Bricsys®

BricsCAD V21

Documentação do produto



Bricsys®

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1. O Que há de novo

1.1 Overview

BricsCAD® V21 offers a plethora of new and improved tools to help maximize your productivity.

This section gives an overview of all the new features and improvements in BricsCAD® **V21 Core and Civil tools**:

User interface na página 32

Productivity na página 35

Drafting na página 49

Modeling na página 51

Civil tools na página 60

Performance and compatibility na página 65

For information about the new features and improvements in BricsCAD® **V21 BIM**, go to the What's new in BricsCAD® V21 BIM section. It contains the following articles:

Overview na página 961

Modeling techniques na página 962

Building data na página 969

Project collaboration na página 978

Design documentation na página 979

Point clouds na página 988

For information about the new features and improvements in BricsCAD® **V21 Mechanical** tools, go to the What's new in BricsCAD® V21 Mechanical section. It contains the following articles:

Visão Geral na página 1532

Gerenciador de BOM na página 1533

Bibliotecas de peças na página 1536

Parâmetros e Restrições na página 1537

Peças padrão na página 1537

Chapa metálica na página 1537

1.1.1 Release notes

Click here for an overview of the release notes.

1.1.2 Diagnostics and Usage Data Collection

Many of the improvements were inspired by requests and suggestions from you, BricsCAD® users. At Bricsys, we continue to expand our methods of gathering user feedback with new data collection tools.

When you launch BricsCAD® for the second time, a dialog box displays, inviting you to participate in the Diagnostics and Usage Data Collection program. The data collection process enables you to share

anonymous diagnostic and usage data with Bricsys to help improve BricsCAD®. You can withdraw your consent and have your data removed from our databases at any time.

1.2 User interface

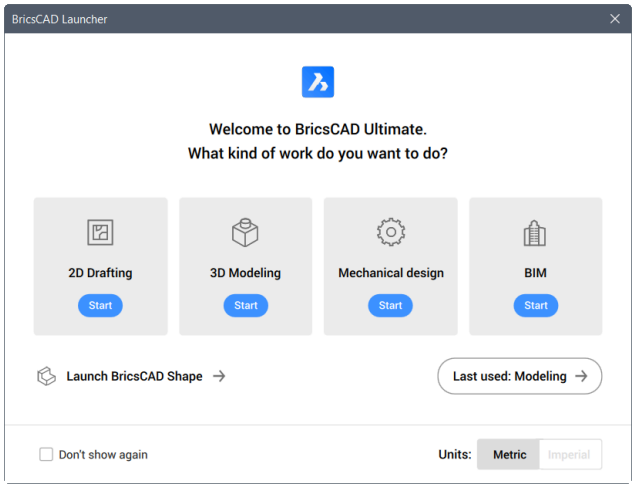
BricsCAD® V21 offers a variety of user interface enhancements to help increase your productivity without disrupting your existing workflow.

1.2.1 BricsCAD Launcher

The redesigned BricsCAD Launcher enables you to launch BricsCAD® in an environment that is most relevant for your design tasks and license level.

The available environment options depend on your license level. You can change the license level using the new SETLICENSELEVEL command which controls the RUNASLEVEL system variable. If you are using a trial license, you may choose any environment. Otherwise, you may choose any environment supported by the license level you purchased. For example, an Ultimate license supports all environments. Whereas, a Lite license supports only the 2D Drafting environment.

The new Units option enables you to set the measurement units (MEASUREINIT system variable) that determine which hatch pattern and linetype definition files to use. This control also specifies the drawing units for new drawings created with no template file.



1.2.2 Commands

The Command window now supports clickable command options enabling you to select options at the Command line without having to type at the keyboard.



The COMMANDS command allows you to look up available commands. In BricsCAD® V21 it's updated to list only the commands that are available with your current license level.

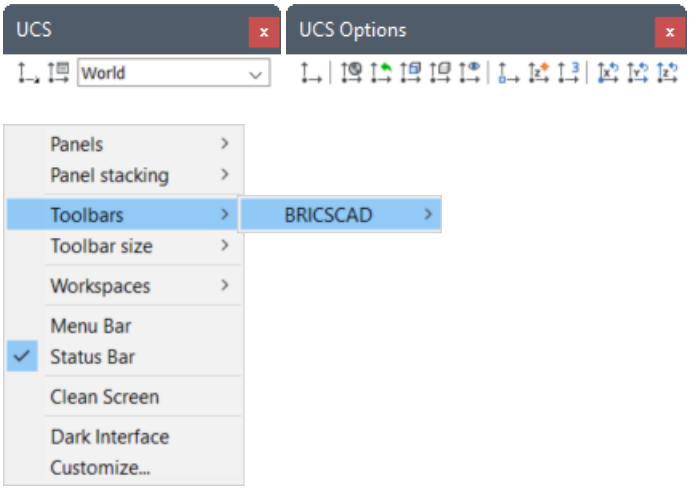
1.2.3 Ribbon

The ribbon enables you to scroll through ribbon tabs using the mouse wheel. Simply place the cursor anywhere on the ribbon and roll the wheel.

The RIBBONTOOLSIZE system variable sets the size of the tool buttons in the ribbon. The options are **Small**, **Large** and **Extra Large**.

1.2.4 Toolbars

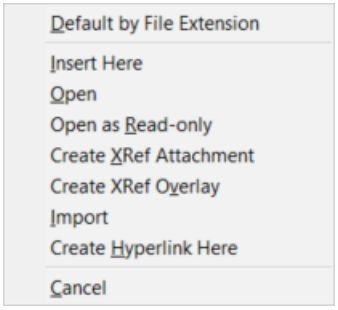
New UCS and UCS Options toolbars offer easy access to UCS commands and options. You an access these toolbars by right-clicking on a toolbar, a panel, or the ribbon and expanding the **Toolbars** > **BricsCAD** menu.



The TOOLBUTTONSIZE system variable applies to toolbars only, no longer to ribbon tool buttons (see above).

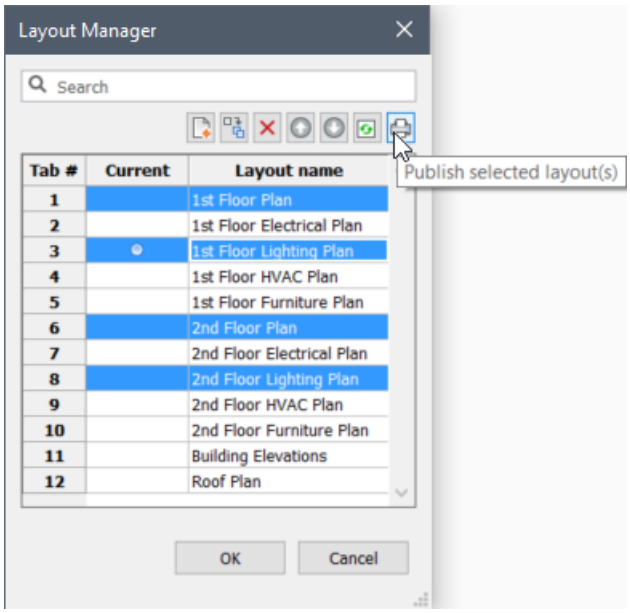
1.2.5 Menus

The menu that displays when you drag and drop a dwg file into BricsCAD®, using the right mouse button, offers a new option to **Open as Read-only**.

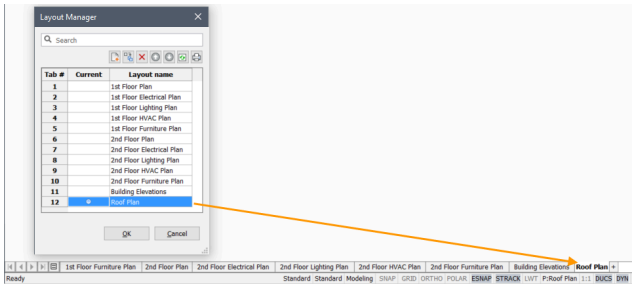


1.2.6 Layouts

The Layout Manager is extended to enable publishing of multiple layouts.

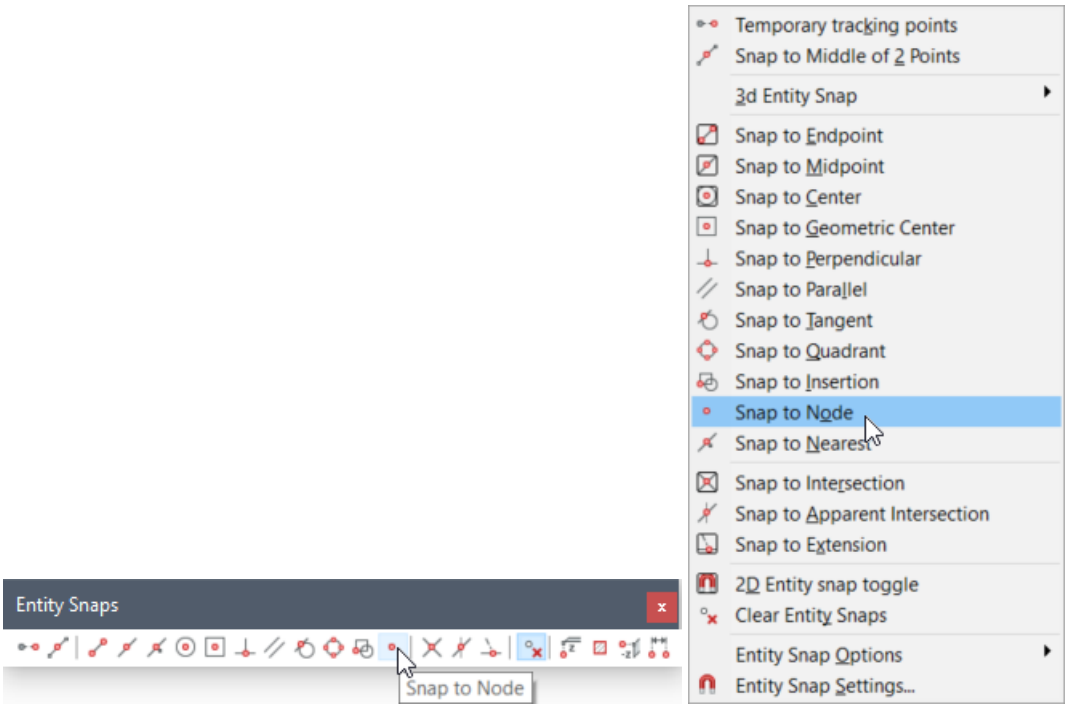


When you set a layout current in the Layout Manager, the active layout tab automatically scrolls to remain visible in the layout tab list.



1.2.7 Entity Snaps

The **Snap to Point** option in the Entity Snaps toolbar and menu is renamed to **Snap to Node**.

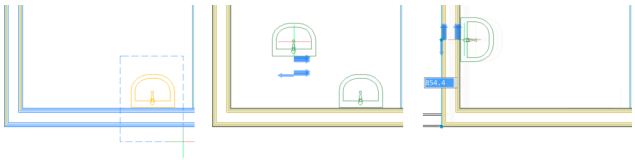


1.3 Productivity

1.3.1 Move with guide curves

The new MOVEGUIDED command enables you to move selected entities using reference curves.

When you launch the MOVEGUIDED command and select the entitiesyou want to move, entities that fall completelywithin the selection windowwill bemoved. Entities crossing the selection windowareused as reference curves.Blue vectors on theselectedentitiesrepresent the reference curves.As you move the selected entities near geometry that matches the reference curves, they automatically snap into place. Relevant geometry in the new location is automatically trimmed and geometry in the original location is automatically healed. This significantly reduces the amount of manual editing required!

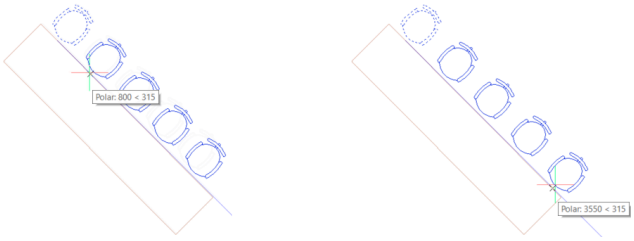


For more flexibility, you can first select the entities you want to move and then launch the MOVEGUIDED command. The pre-selected entities are used for the Move selection set and you are prompted to select entities to use as reference curves.

1.3.2 Copy in a linear array

The COPY command now includes a new **Array** option enabling you to copy selected entities in a linear array. The new Array option is available after you specify the base point for the copy operation.

You specify the number of copies to array then pick a point to specify the distance and angle between the first two sets of entities or use the **Fit** option to specify the distance and angle between the first and last sets of entities.



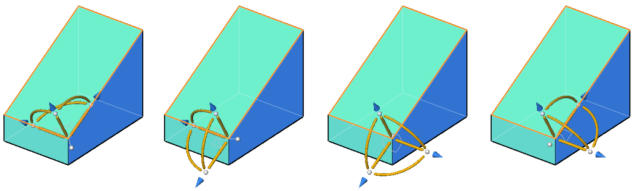
1.3.3 Undo per entity

A new History control in the Properties panel enables you to undo editing operations for a selected entity. The History property only displays when you select a single entity. Open the History drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since.



1.3.4 Manipulator

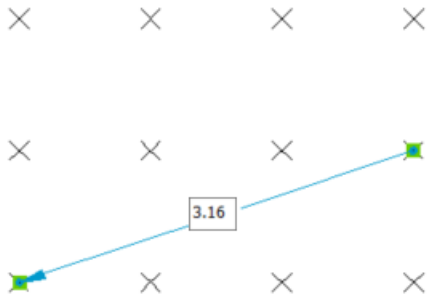
Press the SHIFT key to rotate the Manipulator 90 degrees about its normal axis.



The Copy and Repeat options in the Manipulator are now visible, clickable, and localized.

1.3.5 Nearest Distance

The NEARESTDISTANCE system variable offers improved performance and behavior including support for Point entities.

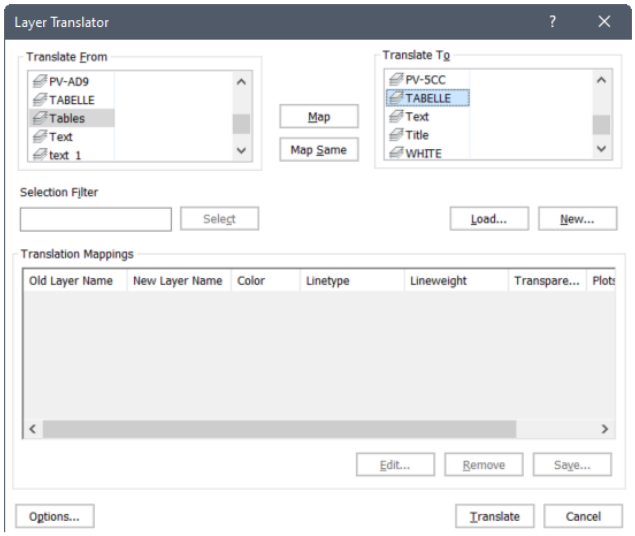


1.3.6 Layers

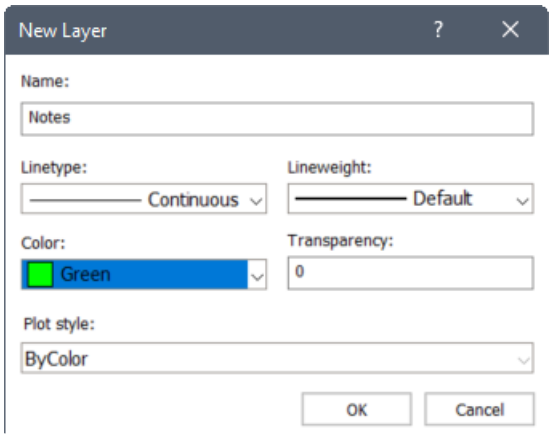
BricsCAD® V21 offers several layer enhancements to help increase your productivity.

Layer Translator

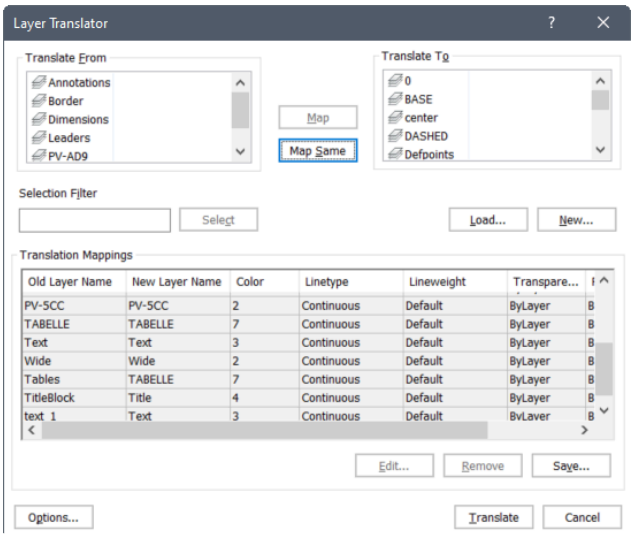
The new LAYTRANS command enables you quickly apply layer names and properties from another drawing file to the current drawing. When you launch the LAYTRANS command, the Layer Translator dialog displays. All the layer names in the current drawing are displayed in the Translate From list. You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.



You can also create new layers. Simply enter the layer name and properties to which you want to map existing layer.

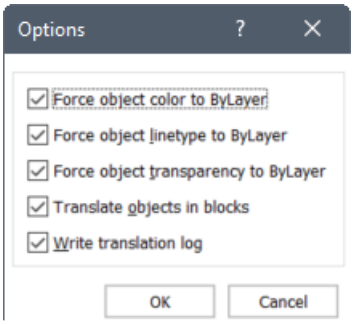


You can select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose **Map Same**, any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right. The layer mappings you select are displayed in the Translation Mappings list.



Regardless of how you add layers to the Translation Mappings list, you can modify their mapping properties. Double-click on a layer or choose **Edit** to open the Edit Layer dialog box with the same options at the New Layer dialog. You can also remove layers from the Translation Mappings list or save the mapping list to use again in the future.

The Options dialog offers additional controls for layer mappings.



Set by Layer

The new SETBYLAYER and -SETBYLAYER commands enable you quickly change property overrides of selected entities to ByLayer.

Use the new SETBYLAYERMODE system variable to control which properties are affected by the SETBYLAYER commands. The value is stored as a bitcode using the sum of the values of all selected options.

- 1: Color
- 2: Linetype
- 4: Lineweight
- 8: Material
- 16: Plot style
- 32: ByBlock
- 64: Blocks

- 128: Transparency

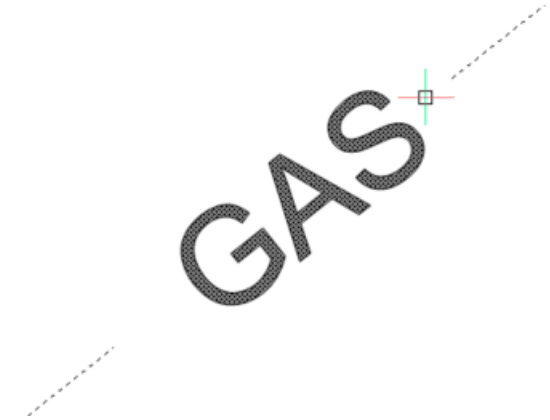
Layer properties by Viewport

The VPLAYER command is enhanced in 21 allowing you to select entities on layers for which you want to apply viewport overrides. After selecting which layer property you want to modify for the viewport, you can type the name of the layer or press enter to specify the layer by selecting an entity on that layer.



1.3.7 Linetypes

BricsCAD® now supports selection of linetype gaps with the addition of the LTGAPSELECTION system variable. With LTGAPSELECTION enabled, entities are highlighted and selected even when you pick in the gap.

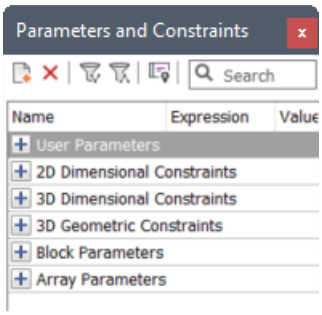


1.3.8 Parameters & Constraints

BricsCAD® V21 offers powerful enhancements for creating parametric drawings and block definitions.

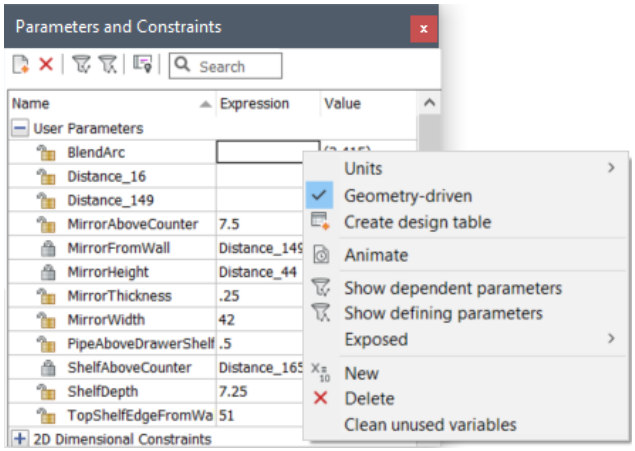
Parameters & constraints panel

All BricsCAD® license levels now support 2D parameters and constraints. View and modify parameters and constraints in the Parameters and constraints panel. The panel is extended to display 3D geometric constraints, 2D dimensional constraints, parametric blocks, and arrays.



The right-click menu for parameters and constraints enables you to specify if parameters are geometry-driven. Expressions for such parameters cannot be edited (you can switch geometry-driven status off to

edit them) – instead they are automatically computed from 3D geometry via 3D dimensional constraints that depend, directly or indirectly, on geometry-driven parameters. Simple arithmetic expressions for such 3D dimensional constraints are solved in dynamics when you change your geometry with direct editing tools such as DMPUSHPULL or DMMOVE/DMROTATE.

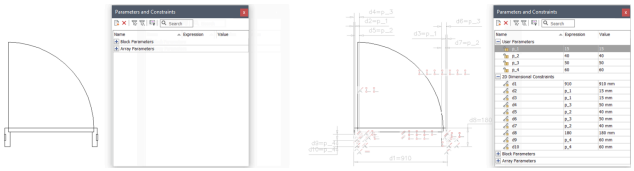


The right-click menu also enables you to control if a parameter is exposed for editing when inserted as a block.

If you open a 3D parametric model with a BricsCAD® Lite license, 3D constraints are shown in magenta. You can view and edit their expressions with BricsCAD® Lite. However, the 3D model only reflects the changes after it's opened with a BricsCAD® Pro license or above.

Automatic constrain

The new AUTOCONSTRAIN command enables you to automatically apply 2D geometric and dimensional constraints to selected 2D geometry. BricsCAD® fully constraints the geometry, ensuring not to over-constrain it. The geometric and dimensional constraints are displayed on the geometry. You can view, add, remove, and modify the parameters and constraints in the Parameters and Constraints panel.

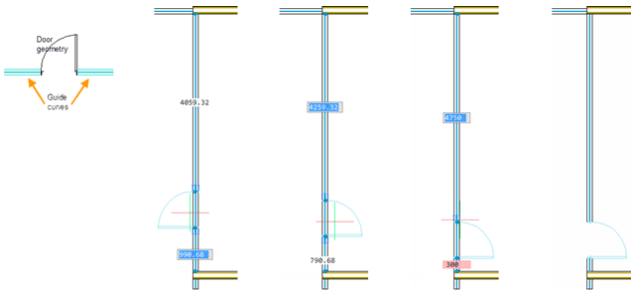


Reference curves

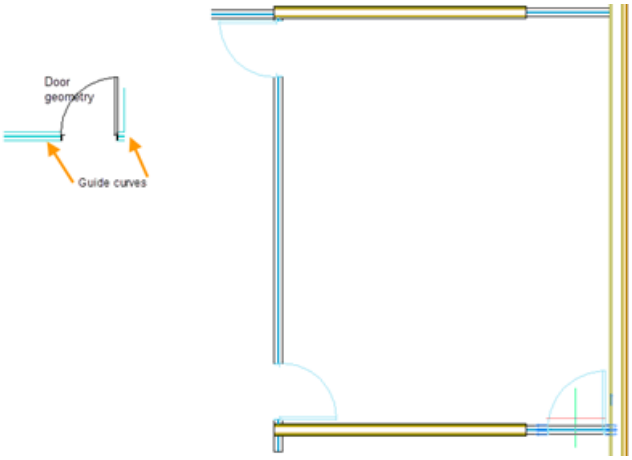
The new REFERENCECURVES command enables you to specify reference geometry for blocks. When you select geometry to use for reference, BricsCAD® creates a REFERENCE_CURVES layer (if it doesn't already exist) and adds the selected entities to that layer.

With reference curves specified in a drawing or block definition, you can automatically align the block or drawing to relevant geometry when you insert it. The number of reference curves and the distance between them determines with which geometry it can align. As the cursor approaches relevant geometry, the block can automatically flip, offering multiple insertion options. Distances between the ends of the

relevant geometry and block are displayed, enabling you to enter specific values if you wish. And, if the reference curves include gaps, relevant geometry is automatically trimmed to produce matching gaps.



You can also use reference curves to automatically align with corners. The following example includes 4 parallel reference curves and a single corner reference curve enabling the door block to align with geometry that matches the four parallel lines and single perpendicular line.



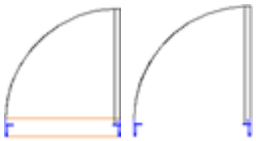
Visibility states

The new VISIBILITYSTATES command enables you to define visibility parameters and states. You can create multiple visibility parameters in a drawing or block definition. And, you can create multiple visibility states in each visibility parameter. When you insert the drawing or block, you can change the visibility of each instance. For example, you might create a door block with two visibility parameters, SwingAngle and Header.

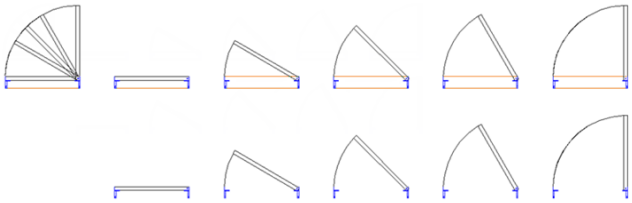
The SwingAngle parameter includes 5 visibility states: Closed, 30, 45, 60, and 90.



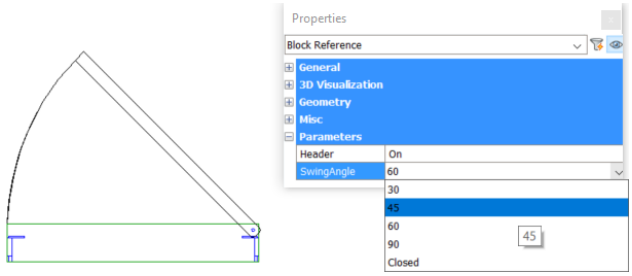
The Header parameter includes two visibility states: **On** and **Off**.



You create all the geometry for each of the visibility states and then select the appropriate geometry to display in each state of each parameter. Geometry that doesn't belong to a visibility state is always displayed. This door example enables 10 variations of block insertions.

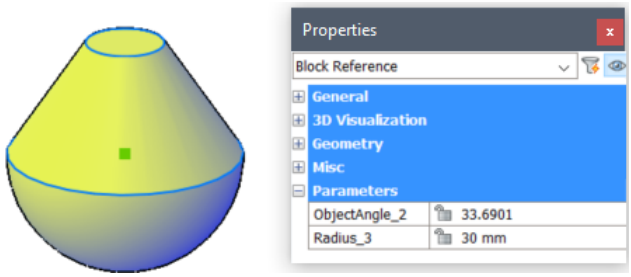


You can easily change the visibility states for a selected block insertion using the Properties panel.



3D Radius constraints

The DMRADIUS3D command enables you to create geometry-driven 3D radius constraints to measure the radius of entities inside blocks.

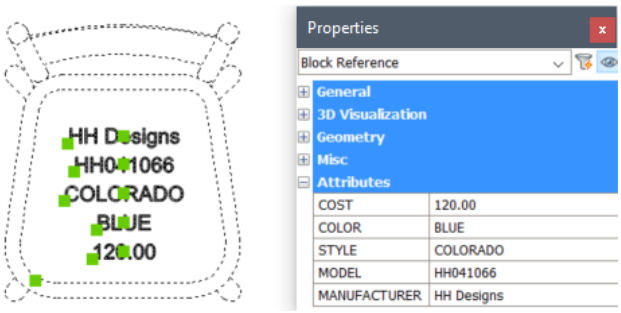


1.3.9 Blocks

BricsCAD® V21 offers many enhancements to increase your productivity when working with blocks.

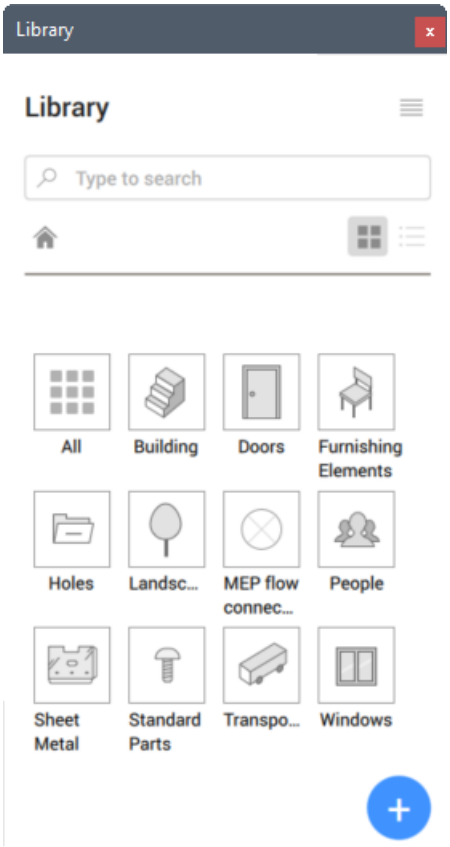
Block attribute properties

You can now access block attribute properties in the Properties panel. This allows you to view and edit multiple attributes at the same time. Press the Ctrl key to select one or more attributes within block instances. Open the Properties panel to display the Attributes entity type with all the relevant attribute properties.

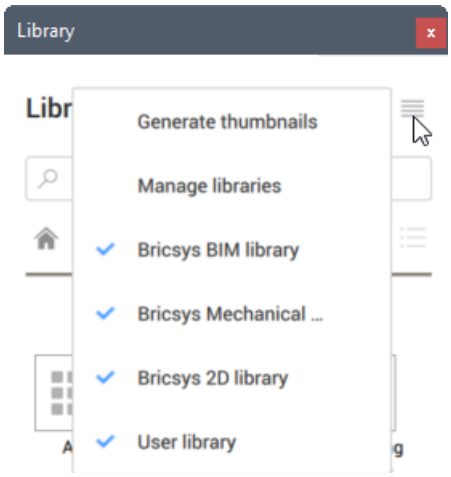


Library panel

The new Library panel offers a central location to access 2D and 3D block libraries. It replaces both the Components and the 2D Parametric Blocks panels from previous releases.

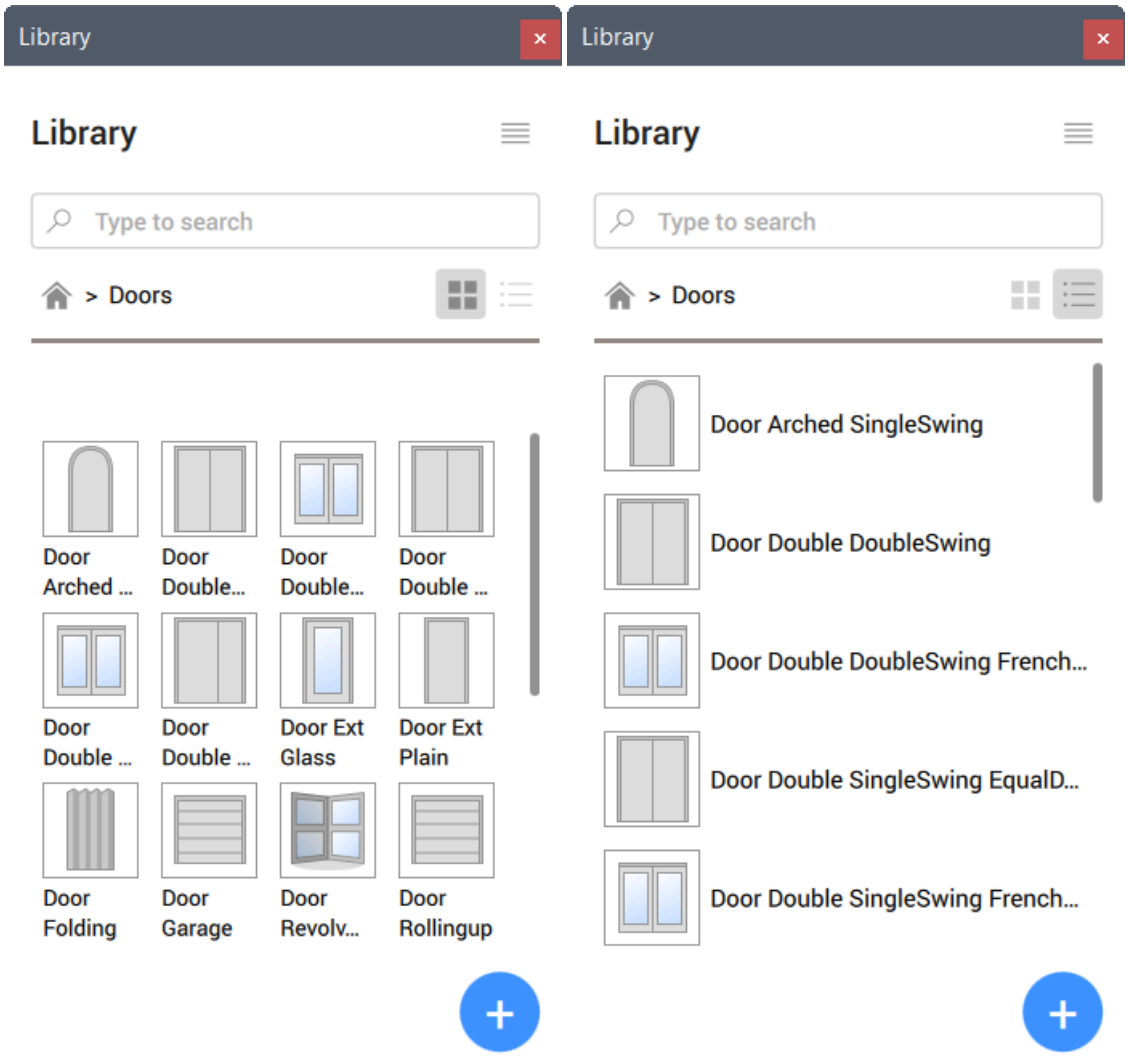


A menu in the upper right corner allows you to control which content is displayed in the Library panel. It may include pre-defined content from Bricsys libraries as well as the User Defined library with your own content.

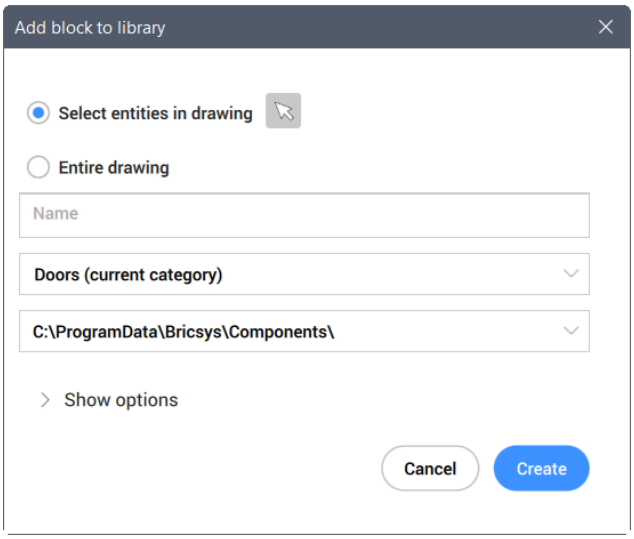


You can specify where BricsCAD® looks for library content by selecting the Manage libraries option. It offers quick access to the COMPONENTSPATH setting so you can view and modify library folder locations. Another menu option automatically generates thumbnail images for any that are missing.

You can search for content by typing key words in the Search box or expand a category to see all the content within that category. And you can display the content in a grid or list view.



You can easily add new block definitions to the library using the Add button at the bottom of the Library panel. Select specific entities from the drawing or use the entire drawing.



Block icons

The new BLOCKICON command creates or updates thumbnail images that enable you to preview blocks with various tools including Drawing Explorer and the Library panel.

Non-uniformly scaled blocks

The EXPLODE command and the XPLODE command now honor the EXPLMODE system variable when exploding non-uniformly scaled blocks.

Insert

The INSERT command is enhanced to support parametric blocks. Block definitions containing parametric entities with exposed parameters can now be parametrically changed after insertion by the INSERT Command. To insert parametric BIM and Mechanical components, use the BMINSERT command.

1.3.10 Drawing Optimizations

A new Drawing optimizations ribbon panel offers tools to clean up and optimize your drawings.

Simplify

The new SIMPLIFY command reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Options allow you to switch between simplifying and smoothening. Additional options enable you to control the amount of deviation from the original entity. Simplified entities are easier to manipulate and can significantly reduce file size.

Purge

The PURGE command is enhanced so the Batch, All, and Orphaned data options remove more items from the drawing.

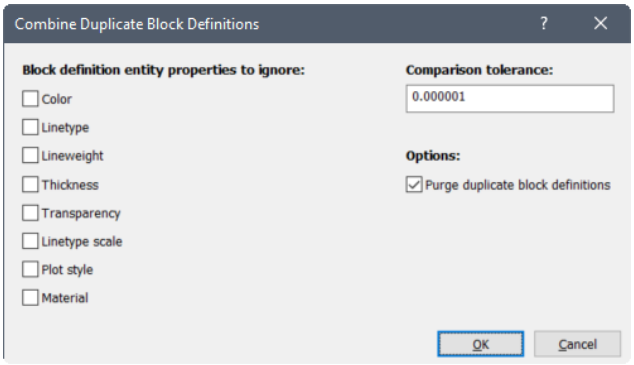
Audit

The AUDIT command is enhanced to clean up duplicate ACIS attributes. A combination of AUDIT and CLEANUNUSEDVARIABLES now cleans a drawing from multiple orphan parameters.

Delete Duplicate Entities

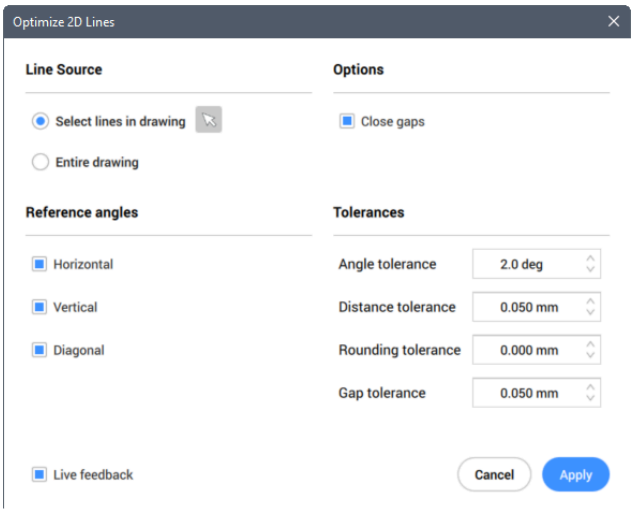
The OVERKILL command enables you to choose if you want to delete duplicate entities or move them to a specified layer. You can specify the layer name using the new OVERKILLLAYER setting or leave the default name, Duplicate Entities.

A new option in the OVERKILL command allows you to combine duplicate block definitions. Choosing this option displays the Combine duplicate blocks dialog box where you can specify any block properties you want to ignore. You can specify how closely the geometry must match by entering a comparison tolerance. If BricsCAD® finds more than one block definition whose geometry meets the tolerance and properties criteria, it uses the most recent definition to replace any instances of the older definitions. The older definitions become unused and you can enable the Purge duplicate blocks option to automatically purge them.

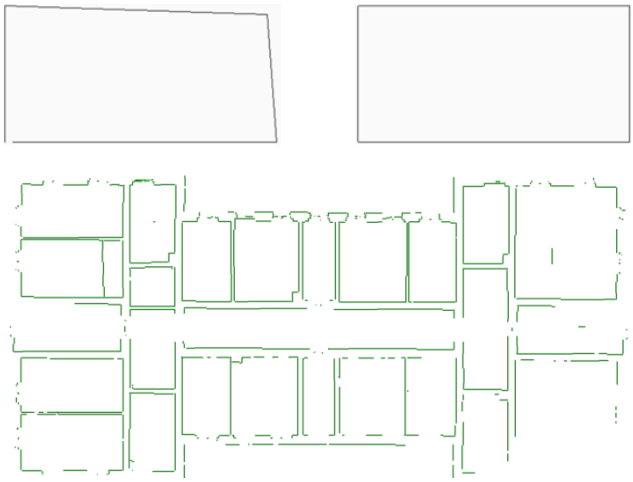


Optimize

The new OPTIMIZE command helps you clean up 2D lines in your drawings. It displays the Optimize 2D Lines dialog box where you can control which lines are affected. Specify which lines BricsCAD® should consider for optimization. You can select specific line entities or have BricsCAD® analyze and optimize all line entities in the drawing. It can find and fix lines that are within a specified angle tolerance relative to horizontal, vertical, or 45 degrees, in the current UCS. An additional option automatically closes gaps within a specified tolerance.

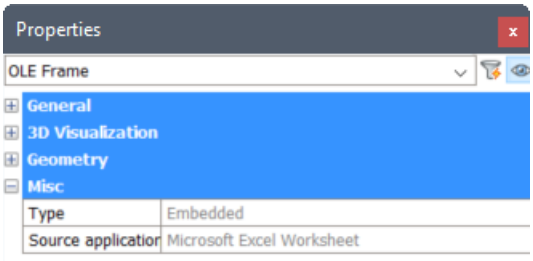


The Optimize tool is useful for cleaning up any 2D lines. For example, you can use it to quickly clean up line geometry extracted from point clouds.



1.3.11 Object linking and embedding

The Properties panel for OLE objects is enhanced to provide additional information for the selected OLE entity. The additional information may include Type, such as Embedded or Static, Plot quality, and Source application.

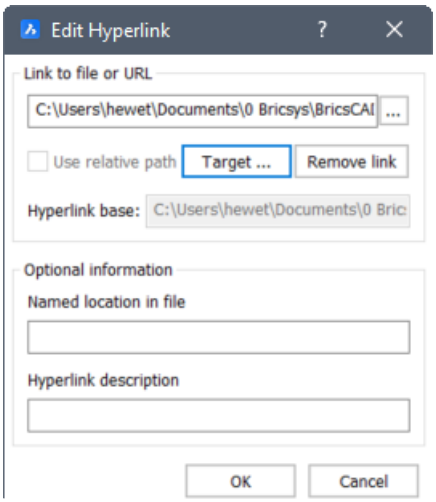


1.3.12 Images

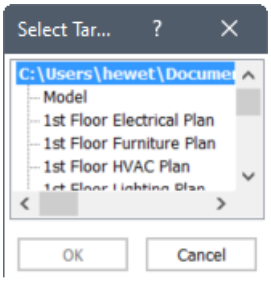
Image manipulation is enhanced in BricsCAD® V21. When you manipulate an attached image using grips and other editing tools the image content displays and updates as you edit the image frame. This enables you to position, align, and scale the image using the image content for reference. The selected image temporarily displays above any overlapping objects as you edit it.

1.3.13 Hyperlinks

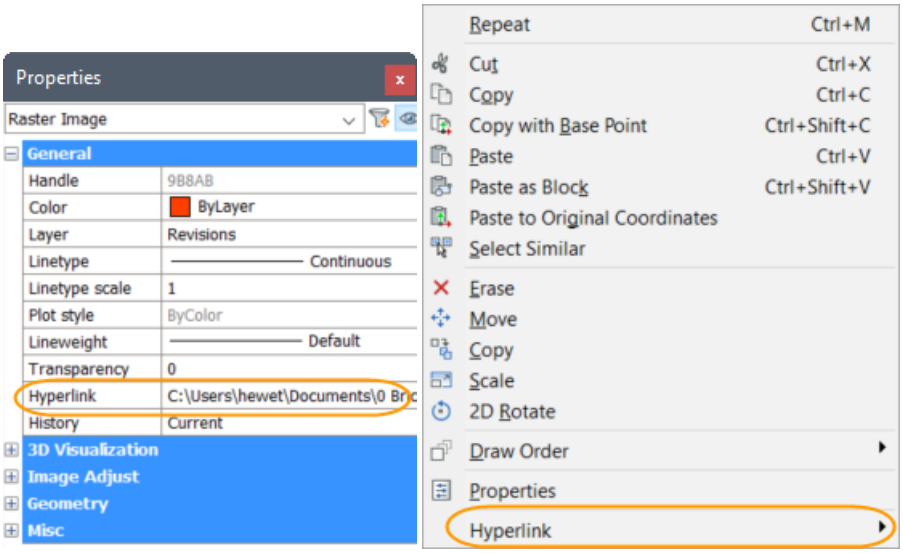
The Edit hyperlink dialog box now enables you to link to layouts and views inside a drawing. After selecting the drawing to which you want to link, choose **Target** to open the Select Target View dialog box.



The Select target view dialog box displays a list of all named views and layouts in the linked drawing.



To access a hyperlink, double-click on the hyperlink in the Properties panel for the selected entity or press and hold the right mouse button and choose **Hyperlink** from the menu.

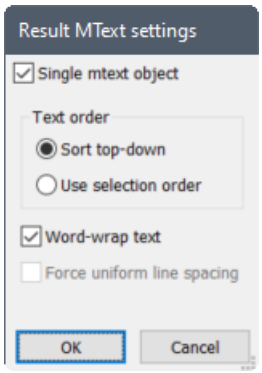


1.4 Drafting

BricsCAD® V21 offers drafting enhancements to help increase your 2D drawing productivity.

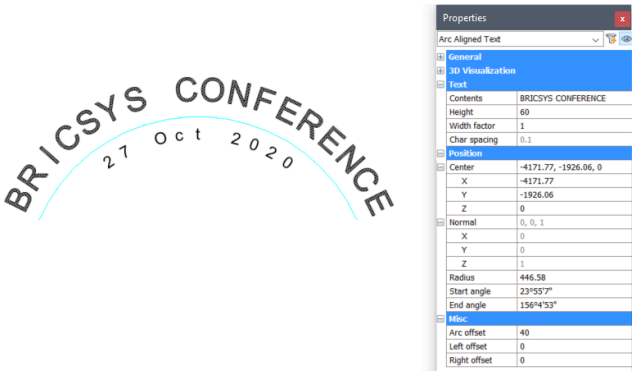
1.4.1 Combine Text to Mtext.

The new TXT2MTXT command (alias COMBINETEXT) enables you to combine single- and multi-line text into a single MText entity. You can control the behavior for combining text in the Result MText Settings dialog box.



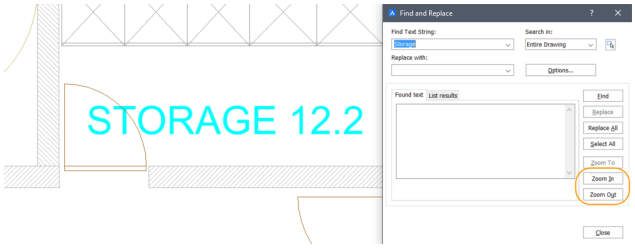
1.4.2 Arc Aligned Text

The Properties panel enables you to view and edit properties for selected Arc Aligned Text entities.



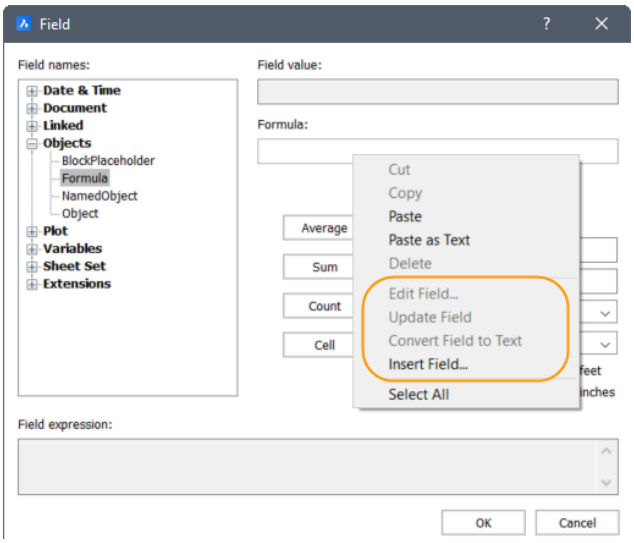
1.4.3 Find and Replace Text

The Find and Replace Text dialog box includes new buttons **Zoom In** and **Zoom Out**buttons. These enable you to take a closer look at the text within the context of the drawing without having to end the Find and Replace operation.



1.4.4 Field

The Field dialog box supports nested formula fields. When you right-click in the Formula field of the Field dialog box, new options enable you to Insert, Edit and Update a field. You can also convert a field to text.



1.5 Modeling

BricsCAD® V21 offers many enhancements to 3D modeling tools to help simplify and automate the modeling process.

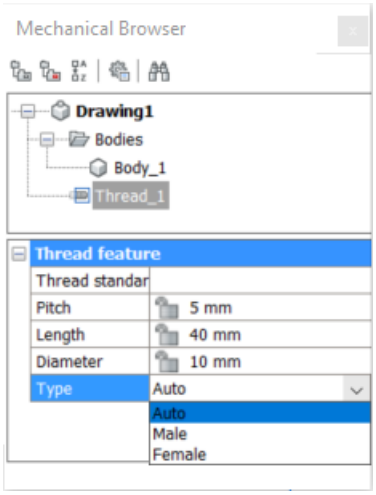
1.5.1 Dynamic UCS

Dynamic UCS is now supported by the PLAN command. This allows you to select a face for the current UCS and immediately display it in plan view.

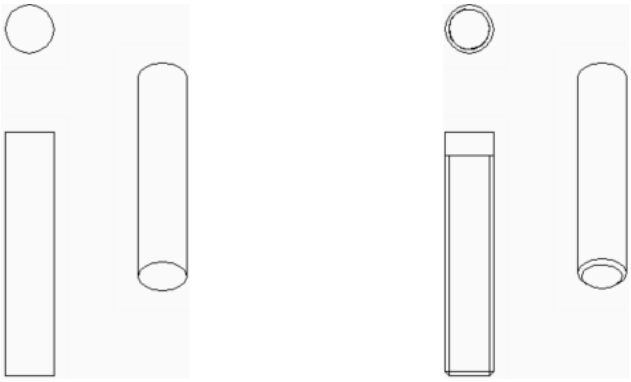
1.5.2 Direct modeling

Threads

The new DMTHREAD command enables you to add a standard parametric thread to a cylindrical face of your 3D solid. You can edit the properties of the thread using the Mechanical Browser.

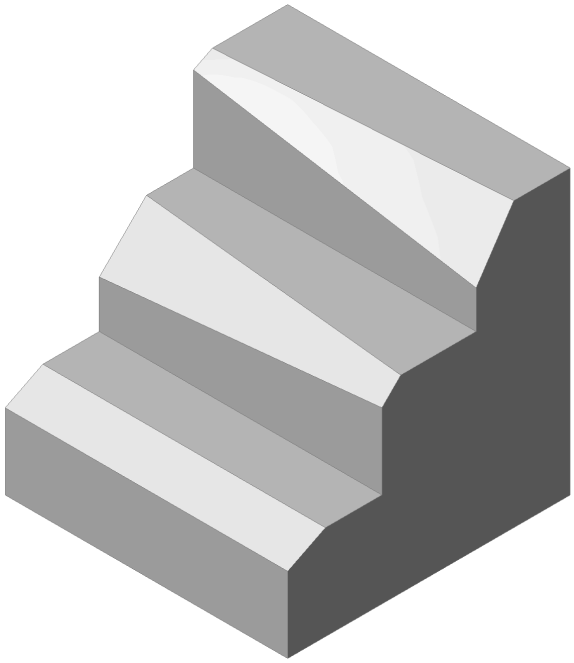


Drawing views created with the VIEWBASE command, the VIEWPROJ command or the VIEWSECTION command represent the threads following drafting standards. The following image show views of a cylinder before and after adding threads.



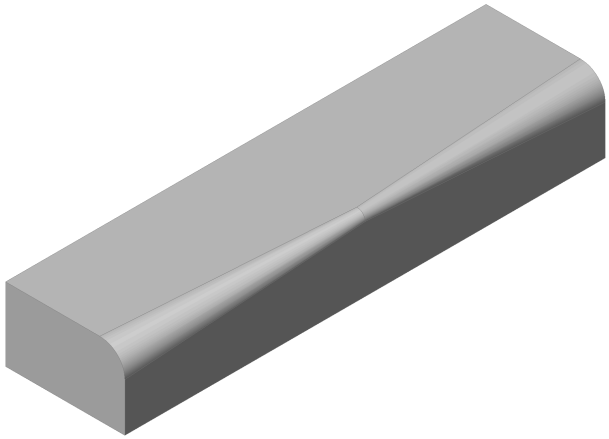
Chamfers

The DMCHAMFER command offers more flexibility with new asymmetrical, angular, variable symmetrical, variable asymmetrical and variable angular modes.



Fillets

The DMFILLET command offers more flexibility with a new variable radius mode.



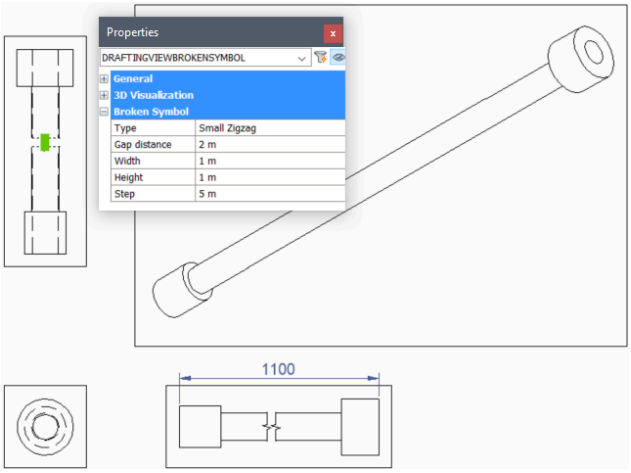
Push Pull

The DMPUSHPULL command includes an option to specify subtract mode (on/off), which is also controlled by DMPUSHPULLSUBTRACT system variable.

It also allows you to switch the starting point of the dynamic dimensions on cylindrical faces. Press the Tab key to switch between radial and offset dimensions.

1.5.3 Broken views

The new VIEWBREAK command enables you to break an associative drawing view of your 3D model created with the VIEWBASE command, the VIEWPROJ command, or the VIEWSECTION command and automatically include break symbols. You can change the appearance of a selected broken view symbol using the Properties panel. The VIEWBREAK command correctly supports dimensions you put on the drawing view before breaking. They remain associative with your 3D model and the drawing view.

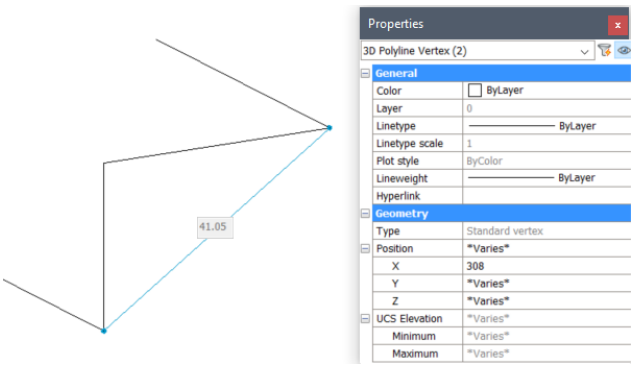


1.5.4 3D Entity snaps

Easily toggle 3D entity snaps on and off by pressing the F4 function key.

1.5.5 3D Polyline

The vertices of 3D polylines now support sub-entity selection, the Properties panel, and nearest distance.



Additional enhancements to 3D Polylines enable you to apply 3D constraints to their vertices and edges.

1.5.6 Parametric blockify

The new PARAMETRICBLOCKIFY command enables you to quickly convert model entities to parametric blocks. Select a parametric block or parametrized entities as input geometry to compare against other geometry in the drawing. Matching geometry is automatically converted to parametric blocks.

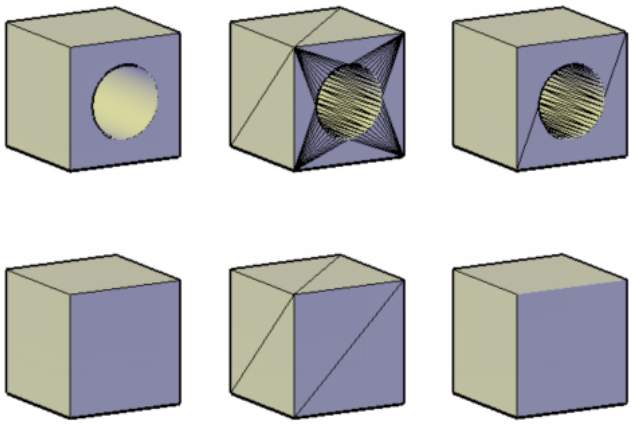
1.5.7 3D display

Material glossiness has been adjusted to ensure entity details are distinguishable, even with glossy materials. The glossy effect is less intense for light materials and more intense for dark materials.

1.5.8 Solids and meshes

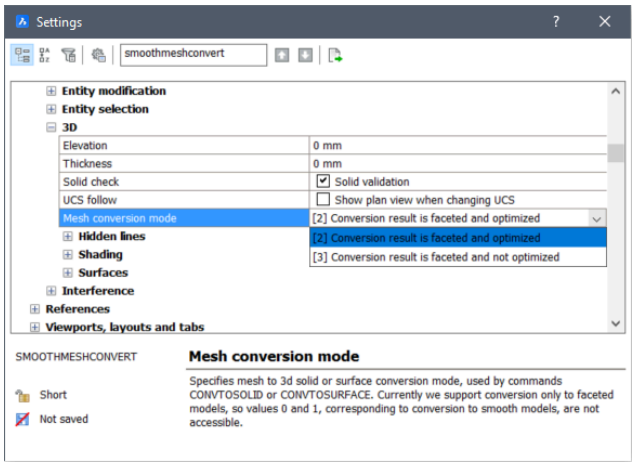
Convert to mesh

The CONVTOMESH command is optimized to triangulate planar faces only when necessary and with as few faces as possible. As an example, the original solids on the left are converted to meshes in BricsCAD® V20 (center) and BricsCAD® V21 (right).

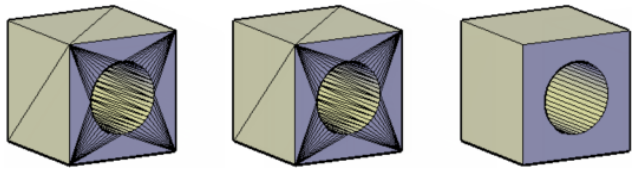


Convert to solid

The CONVTOSOLID command enables you to convert a mesh to an optimized 3D solid based on the new SMOOTHMESHCONVERT system variable.

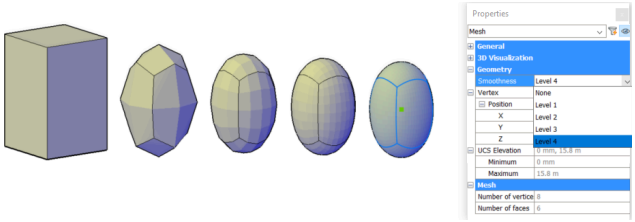


When set to optimized, coplanar and adjacent faces are merged into a single face. As an example, the original mesh on the left is converted to a solid in BricsCAD® V20 (center) and BricsCAD® V21 (right).

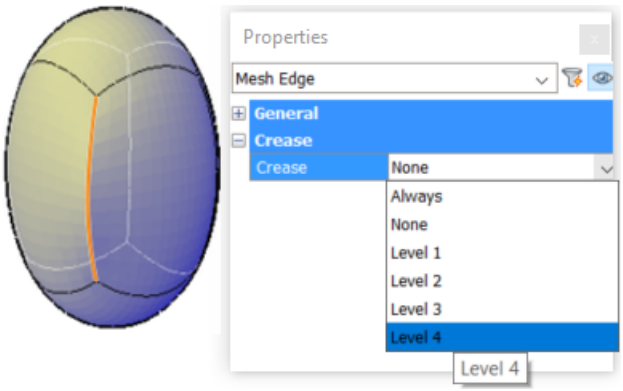


Mesh properties

The Properties panel offers access to additional properties for selected mesh entities. Mesh entities now include Mesh and Geometry properties including the ability to specify smoothness.



The Properties panel also supports mesh sub-entities including faces, edges and vertices enabling you to apply different levels of creasing.



1.5.9 Associative data

The new GENERATEASSOCATTRS setting controls generation of associative data on 3D solids during modeling. When set to On, 3D solids contain associative data from the moment of creation. This allows BIMSECTIONUPDATE command and VIEWBASE command commands to produce drawings for which dimensions and tags can be automatically updated after the 3D model changes.

1.5.10 View Twist

The Twist option is added to the -VIEW command.

1.5.11 Animation Editor

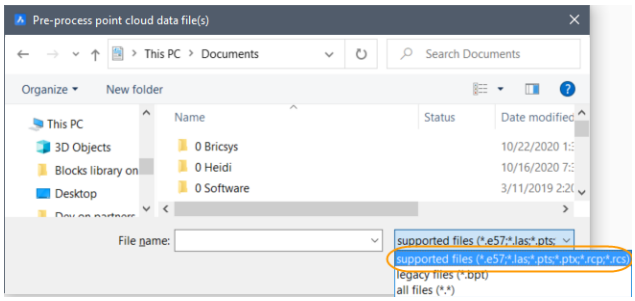
The Animation Editor is enhanced to support the animating of exploded views in model space.

1.5.12 Point clouds

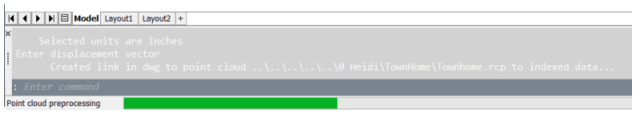
Point cloud functionality is significantly faster and more powerful in BricsCAD® V21.

Pre-processing

The E57 file format is added to the supported files for pre-processing.



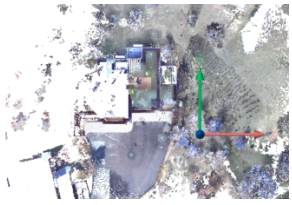
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD® V21 than in V20. A progress meter at the bottom of the display shows the conversion progress.



Alignment

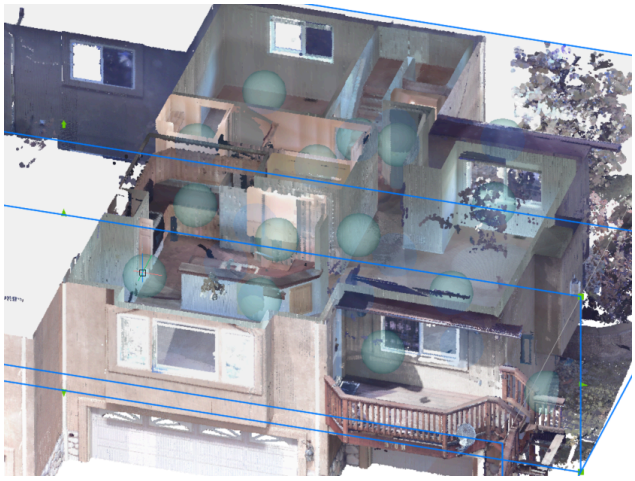
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment it can analyze the entire point cloud or you can specify the most relevant area.





Bubble Viewer

The point cloud displays bubbles at all the scan locations. It's at those locations where you will experience the most realistic visual representations.



Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location.



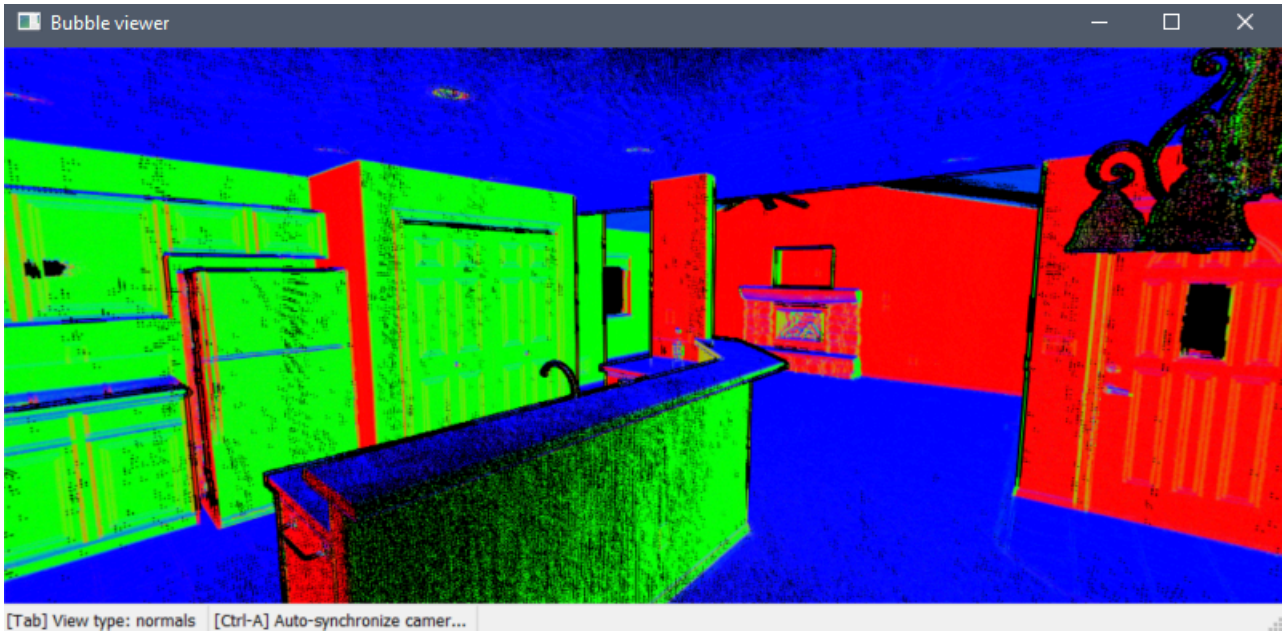


Press the Tab key to cycle between three different visual modes.

The first mode displays the points as their actual colors or in grayscale, depending how the data was scanned.

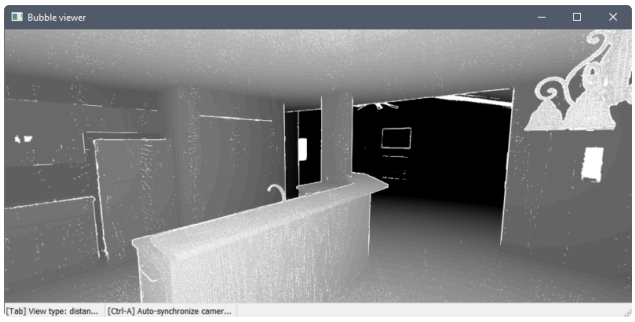


The second mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.





The third mode displays the points from light to dark as the distance from the scan location increases.



You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.



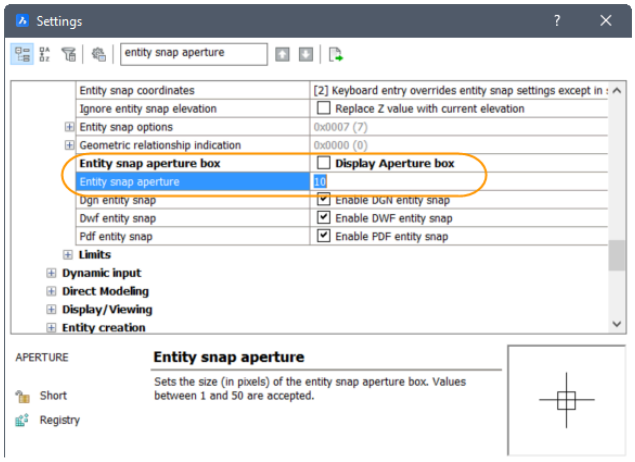
Entity snaps

The new Point Cloud Closest Point entity snap significantly improves your ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.

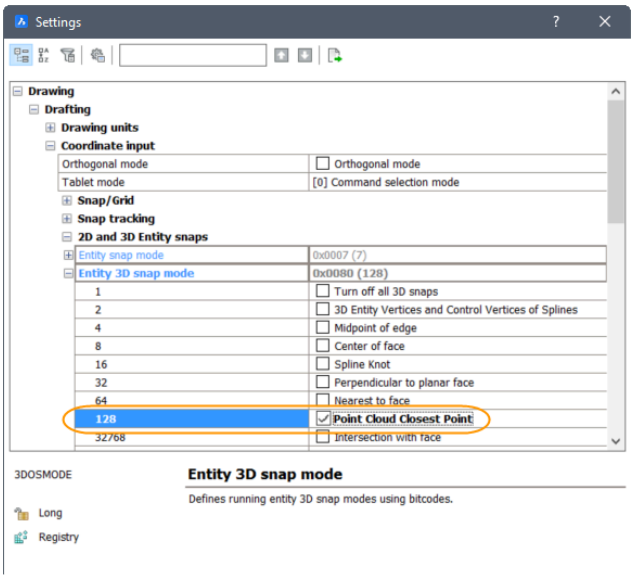




The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.



Enable the new Point Cloud Closest Point entity snap along with other 3D entity snaps in the Entity Snap menus, toolbar, and settings.



Export

The new POINTCLOUDEXPORT command allows you to export a cropped selection of a point cloud to a .pts file.

1.6 Civil tools

BricsCAD® V21 offers many advanced tools for linear infrastructure modeling.

1.6.1 Import/Export

New and enhanced tools for importing and exporting civil data help accelerate collaboration and design processes.

Civil 3D Data

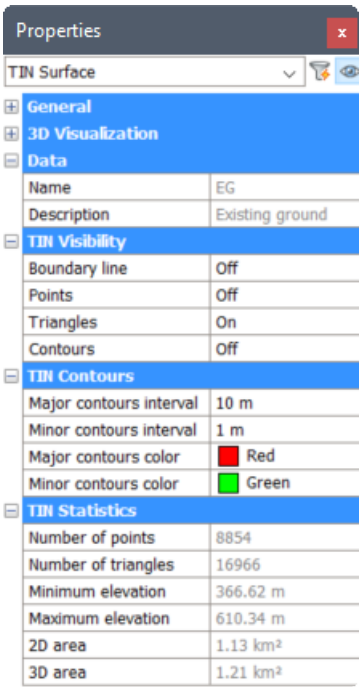
The new CIVIL3DIMPORT command enables you to create multiple civil entities in your BricsCAD® drawing, from an external Autodesk® Civil 3D drawing, in one operation.

LandXML Data

The LANDXMLIMPORT command and LANDXMLEXPORT command tools offer more flexibility. They support new alignment curves including spiral-curve-spiral combinations and parabolas for vertical alignments. Imported vertical alignments automatically create 3D Alignments. And, when importing a surface with breaklines, a new option allows you to specify whether the breaklines should be drawn as 3D polylines or not drawn at all.

1.6.2 Civil Entity Properties

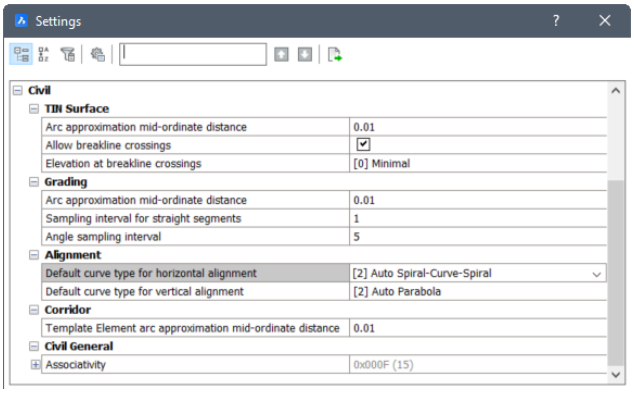
All Civil entities now include Name and Description properties in the Properties panel. Name and description are in the read-only mode in the Properties panel. They can be edited with the API.



Selected sub-entities of (Civil) custom entities are also displayed in the Properties panel.

1.6.3 Civil settings

A new Civil node is included in the Settings dialog box for easy access to Civil settings.



1.6.4 TIN Surfaces

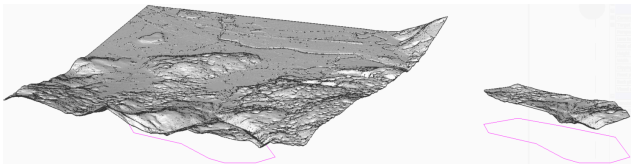
New and enhanced tools for creating and editing TIN surfaces provide more flexibility and a faster workflow.

Associativity

Points and blocks are dynamically linked to the TIN surface. If a TIN surface is created from points or blocks in the drawing and some points/blocks are moved, the TIN surface automatically updates to reflect the changes.

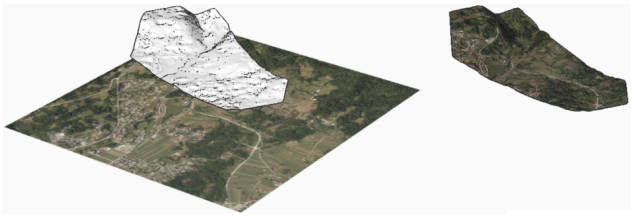
Clipping

A new option in the TIN command and TINEDIT command enables you to clip a TIN surface according to a given polyline boundary.



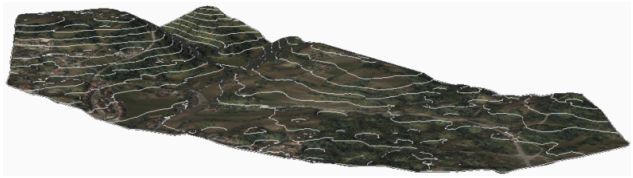
Raster Images

The new TINASSIGNIMAGE command enables you to assign a raster image as a TIN Surface material to create a realistic site model.



Extract

The TINEXTRACT command enables you to extract faces, points, contours, and the border from a TIN surface, in addition to the previous ability to extract a mesh or solid.



You may also extract a TIN surface from a Grading or TIN Volume Surface using the TINEXTRACT command.

Project

The new TINPROJECT command enables you to project point-based entities including points, blocks, and text or linear entities, including lines, polylines, and circles, to a TIN Surface.





Water Path

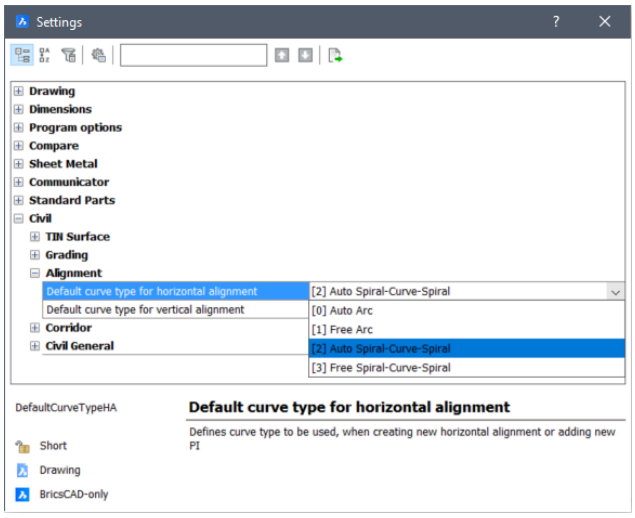
The new TINWATERDROP command enables you to draw a water drop's path from a point on a TIN Surface.

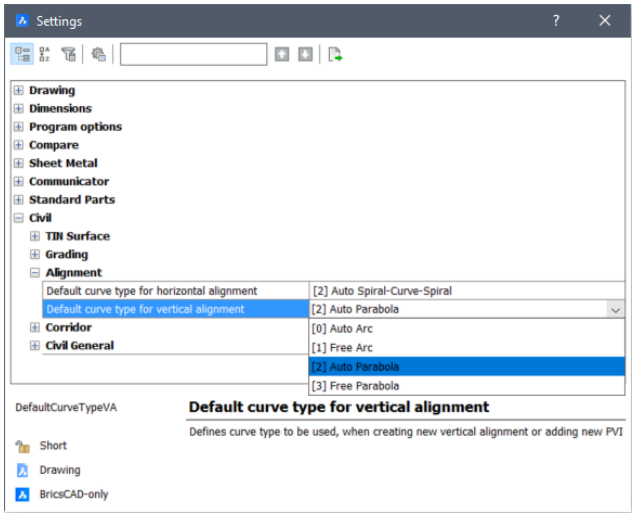


1.6.5 Alignments

The workflow for creating alignments is simplified while still offering powerful functionality and additional flexibility.

Horizontal and vertical alignment geometry is enhanced. Horizontal alignments now support spirals and vertical alignments now support parabolic curves. The new DEFAULTCURVETYPEHA and DEFAULTCURVETYPEVA alignment settings enable you to specify the default curve type for creating new horizontal and vertical alignments.





The Properties panel displays properties for Spiral-Curve-Spiral or parabola elements.

1.6.6 Corridors

Powerful corridor tools help you create a 3D model of linear infrastructure defined by a template and 3D alignment.

The new CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from closed polylines.



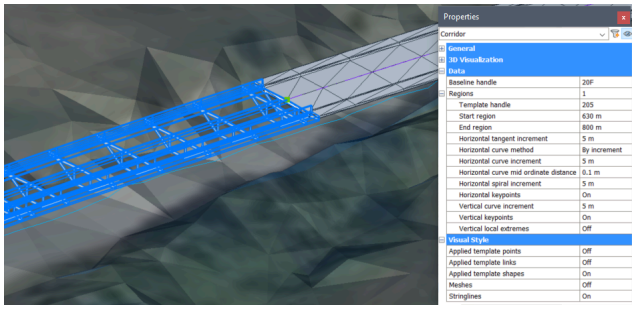
Then use the new CORRIDORTEMPLATE command to create a corridor template from template elements.



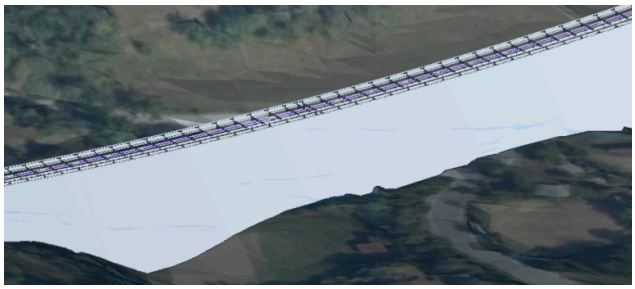
Finally, use the new CORRIDOR command to create a corridor from a corridor template, 3D alignment and/or TIN Surface. You can create multiple regions using different corridor templates. And, you can add or remove regions using the new CORRIDOREDIT command.



Use the Properties panel to modify data and visual style properties of the selected region.



The new CORRIDOREXTRACT command enables you to extract a 3D solid, 3D mesh, or 3D polyline from a corridor to use for various purposes. For example, you can extract polylines from a corridor to use as edges for the grading. If you grip edit the alignment, the corridor and its associated geometry, such as the grading, automatically follows.



1.6.7 Gradings

The new GRADINGBALANCE command enables you to balance grading cut and fill volumes.

1.6.8 TIN Volume Surfaces

The ARRAYASSOCIATIVITY system variable includes a new control for TIN Volume Surface associativity. TIN volume surfaces automatically rebuild when their source surfaces change.

1.6.9 API Samples

The following API samples are provided: Horizontal alignment report, Color surface triangles by elevation, Color surface triangles by slope.

1.7 Performance and compatibility

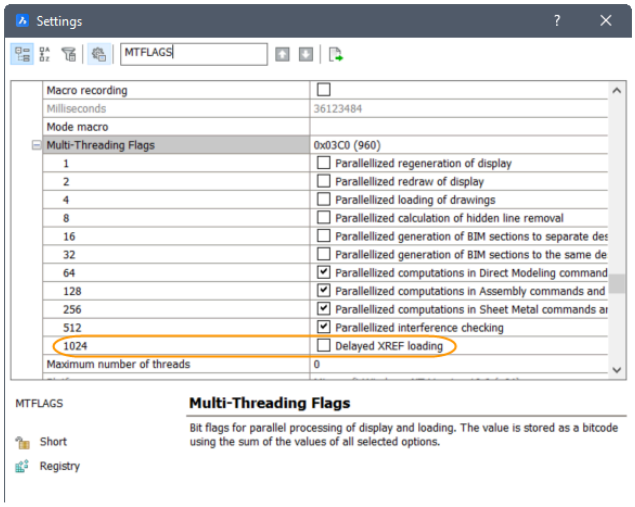
BricsCAD® V21 offers new and enhanced tools to improve performance and compatibility as you work individually or in a collaborative environment.

1.7.1 Selection Preview

Selection preview highlighting is deactivated during Pan operations, resulting in a smoother user-experience.

1.7.2 Xref Loading

A new MTFLAG is added to delay the loading of Xrefs to idle time.



1.7.3 Multiuser Version Control (BETA)

BricsCAD® V21 introduces new multi-user version control functionality as a beta technology preview. This functionality enables multiple users to work on the same drawings, simultaneously. Drawings are stored in the cloud using Bricsys 24/7 and checked in and out from each user's local machine.

Access

Since this is Beta functionality, you must request access by submitting a support request.

1 Log into your Bricsys account.

2 Choose New Support Request.

3 Open the BricsCAD® menu and choose **BricsCAD** > **Version Control**.

4 In the Subject field enter: Access to VERSIONCONTROL Beta.

5 Fill in the rest of the fields as needed.

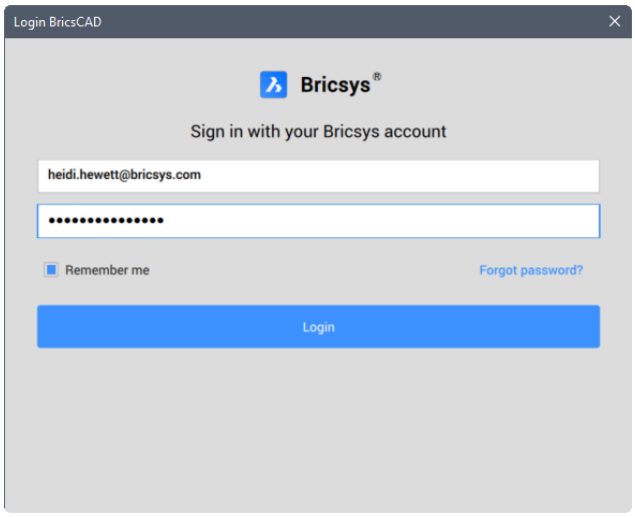
6 Choose Send Support Request.

VERSIONCONTROL Command

The new VERSIONCONTROL command offers the following options:

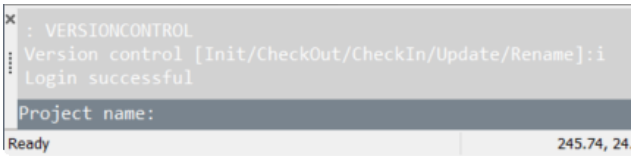
- Init
- Checkout
- Checkin
- Update
- Rename

The first time you launch VERSIONCONTROL in a new BricsCAD® session and choose one of the options, it prompts you to log into your Bricsys account.



If your account has not been granted access to the beta functionality, it directs you to a web page with instructions to request access.

If your account has access to the beta functionality, your successful login is indicated in the Command window and it prompts you to enter the name of the project on which you want to work.



Init

The **Init** option enables you to create a new multi-user project. Enter a new project name, keeping in mind project names are case-sensitive. If you enter a name that already exists, it prompts you to enter a different name.

Checkout

The **Checkout** option enables you to check out a drawing from your cloud-based projects. You must specify from which project you want to check out a drawing. Enter the project name, keeping in mind project names are case-sensitive. If you enter a project name that doesn't exist, BricsCAD® displays a list of valid project names and prompts you to reenter the project name.

After entering a valid project name, you must specify a version control folder for that project. The version control folder is the location on your local machine where your drawings are stored and synced with the cloud. The default location is *drive:\Users\username\Documents\Bricsys247\projectname*. However, you can specify any location. If the location doesn't exist, it's created and set under version control. If the location already exists it's set under version control. If the location is already under version control, you are prompted to specify a different location.

Checkin

The **Checkin** option enables you to check in drawings from your local, version control folder to the cloud-based project. You can check in all files in the version control folder or you can commit modified and untracked files.

After specifying what to check in, you can enter a relevant Check In message.

Update

The **Update** option enables you to sync project drawings from your local folder with the cloud.

Rename

The **Rename** option enables you to rename a drawing that's under version control.

1.7.4 DGN Export

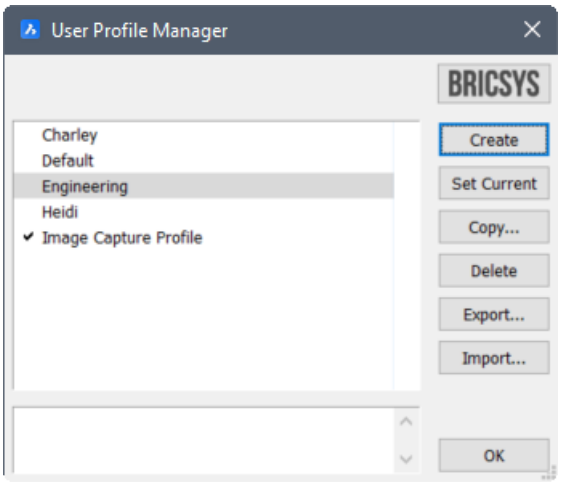
A new DGNEXPORT command enables you to export the current drawing to DGN file format.

1.7.5 3D DWF Output

The 3DDWF command enables you to create 3D DWF or 3D DWFx files based on the value of the new DWFFORMAT system variable.

1.7.6 User Profile Manager

The User Profile Manager is now more powerful. It can now delete or replace existing user profiles, including the current one. Deleting the current profile doesn't completely remove it, but it does reset all settings to factory defaults.



1.7.7 Start Menu Shortcuts

Start menu shortcuts now set the program folder instead of personal documents folder as the initial working directory. This change solves problems with installation under a system account where personal folders are not available.

2. Instalar e licenciar

2.1 Instalar o BricsCAD®

2.1.1 Requisitos Mínimos e Recomendações do Sistema

Requisitos e recomendações do sistema para computadores Microsoft Windows e Linux baseados em Intel e AMD e Apple Macs baseados em Intel.

Sistemas Operacionais Suportados	• Windows 10 (x64) • Windows 8.1 (x64) • Ubuntu V18.04 or superior • openSUSE builds, depois de abril de 2018 • Fedora builds, depois de abril de 2018 • Mac OSX 10.13 ou superior	***Nota:*** Windows 8.1 chegará ao final do programa de suporte estendido da Microsoft em 10 de Janeiro de 2023. ***Nota:*** O Mac OSX é suportado somente no hardware baseado em Intel neste momento.
Sistemas Operacionais Obsoletos	• Windows Server 2008 R2 • Windows Server 2008 (com SP2 ou superior) • Windows 7 (x64)	***Nota:*** não recomendamos executar software CAD de missão crítica em sistemas operacionais não suportados, pois seu PC pode se tornar mais vulnerável a riscos de segurança. • Em 14 de Janeiro de 2020, a Microsoft encerrou o suporte para Windows Server 2008 e 2008 R2. • Em 14 de Janeiro de 2020, a Microsoft encerrou a iniciativa Suporte Estendido do Windows 7.

CPU mínima: • Intel® Core™ i5 • AMD Ryzen™ 5 • 2,5 GHz ou superior	**CPU recomendada:** • Intel® Core™ i7 • Intel® Core™ i9 • AMD Ryzen™ 7 • AMD Ryzen™ 9 • 3.0 GHz ou superior	A CPU com a mais alta classificação de desempenho de thread-único sempre se destacará com **BricsCAD®**. Núcleos de CPU adicionais podem ser um bônus, porque o **BricsCAD®** suporta o conceito de “threads auxiliares” para muitas tarefas-intensivas de dados e cálculos. **Nota sobre a Intel® Xeon® e AMD Ryzen™ Threadripper™ CPUs**: essas CPUs de estações de trabalho de nível superior oferecem suporte a grandes cache de memória no chip, mais PCIe® Faixas de I/O e arquiteturas de memória multicanais e de alta velocidade. Eles ajudarão a **BricsCAD®** executar mais rápido? Talvez. Você precisará fazer testes em seu ambiente específico, para determinar se o custo adicional desses sistemas é justificável.
Memória Mínima do Sistema: 8 GB	**Memória de Sistema Recomendada:** • 16 GB • 32 GB	Para obter o máximo desempenho, determine a configuração de memória do seu PC e certifique-se de que cada canal de memória na placa-mãe esteja preenchido com pelo menos um módulo de memória (SIMMS/DIMMs). ***Nota:*** o número de canais de memória disponíveis pode não correlacionar 1:1 com o número de soquetes RAM na placa-mãe do seu PC. Questões? Por favor, confira com seu fornecedor de hardware antes de comprar a memória do sistema.
Espaço Mínimo em Disco: 2 GB para **BricsCAD® Ultimate**, instalado	**Espaço em disco recomendado:** (Conforme necessitar)	Recomendamos unidades de estado sólido (SSDs) como unidades de sistema, tanto para **BricsCAD®** quanto para Windows. Desenhos e arquivos de suporte podem residir em unidades de disco rígido mecânicas mais lentas, se necessário. ***Nota:*** discos de solid-state podem ser até 10 vezes mais rápidos na leitura e 20 vezes mais rápido na gravação de dados do que uma unidade de disco rígido de médio alcance (ex. Seagate Barracuda 5.4k RPM, SATA 6.0).

Sistema de Monitores Mínimo: • Telas com 1920 x 1080 em True Color e uma placa gráfica com VRAM de 1 GB • Apple Mac Monitores	**Sistema de Monitores Recomendado:** • Vários monitores True Color 3840 x 2160 (4k), com um PCIe® placa gráfica com GPUs e VRAM de 4 GB • Telas Apple Mac Retina (4K e 5K)	Em sistemas Microsoft Windows, a maioria das placas gráficas & GPUs da NVIDIA, AMD e Intel® são suportadas. Nos sistemas Linux, a maioria das placas gráficas & GPUs da NVIDIA e da AMD são suportadas. Em sistemas Mac, o hardware de vídeo nativo Apple é suportado. **BricsCAD®** os modos de exibição não-wireframe são gerados pela tecnologia RedSDK da Redway3D®. Para o desempenho máximo do RedSDK nesses modos, é necessária uma GPU. **Nota:** Sugerimos que você instale os drivers mais recentes disponíveis para seu hardware de vídeo, conforme recomendado pelo fabricante. **Nota aos usuários do Linux:** a aceleração de hardware de gráficos 3D em sistemas Linux NÃO é suportada em: • Intel® chipsets gráficos/placas • Laptops com adaptadores gráficos duplos. No entanto, recentes desenvolvimentos de drivers pela NVIDIA mostram sinais promissores nesta área. Para obter mais informações sobre hardware gráfico suportado para **BricsCAD®**, consulte a página de referência do chipset GPU Redway3D Redway3D GPU chipset reference page.

2.1.2 Download do BricsCAD®

2.1.2.1 Baixando a versão mais recente de BricsCAD®

1 Se você ainda não fez isso, acesse o site da Bricsys, Registre-se e faça Login.

2 Clique em **Baixar BricsCAD**.

3 Selecione a plataforma, idioma e versão (64-bit).

4 Marque a caixa de seleção **Eu concordo com os termos de uso**.

5 Clique no botão **Download**.

2.1.2.2 Baixando uma versão antiga de BricsCAD®

1 Se você ainda não fez isso, acesse o site da Bricsys, Registre-se e faça Login.

2 Clique em **Baixar BricsCAD**.

3 Clique no link **Versões Anteriores**.

4 Selecione a plataforma e o idioma.

5 Selecione a versão **BricsCAD®**.

6 Selecione a versão (64-bit).

7 Marque a caixa de seleção **Eu concordo com os termos de uso**.

8 Clique no botão **Download**.

2.1.3 Visão geral das opções de instalação

- **BricsCAD®** vem como um único arquivo de download para todos os níveis de licença, inclusive **BricsCAD® Shape**.
- Quando você instala uma nova versão do **BricsCAD®** pela primeira vez, você pode executar um modo de teste de 30 dias.
- Você pode instalar cada nova versão principal ao lado de uma versão principal anterior. Essas não se auto-influenciam.

2.1.3.1 Instalação Interativa

No caso de uma instalação normal (não silenciosa), o usuário fornece a entrada de instalação necessária através de caixas de diálogo.

A interface do usuário do Windows Installer consulta o sistema de destino e exibe o Assistente de Configuração. O Assistente de Configuração permite que o usuário altere várias opções que afetam a instalação.

2.1.3.2 Instalação Silenciosa

No caso de uma instalação silenciosa a interface do usuário do Assistente de Configuração é desativada e as ações durante a sequência da interface do usuário não são executadas.

2.1.3.3 Instalação Interativa

1 Clique duas vezes no arquivo do instalador.

2 Clique em **Próximo**.

3 Marque a caixa de seleção para aceitar o **Contrato de Licença**.

4 Clique em **Próximo**.

5 Siga um dos procedimentos:
   - Aceitar a pasta padrão de instalação.
   - Clicar no botão **Alterar...** e selecione uma pasta de instalação.

6 Clique em **Próximo**.

7 Clique em **Instalar** para iniciar a instalação.

8 Escolha se deseja ou não criar um atalho na área de trabalho e/ou exibir as Notas de Release da versão quando o processo de instalação estiver terminado.

9 Clique em **Próximo**.

10 Clique em **Concluir** para concluir o processo de instalação.

Result: **BricsCAD®** está instalado.

2.1.3.4 Instalacao Silenciosa (somente Windows)

Para uma instalação silenciosa, você precisa dos direitos de Administrador.

1 Abra o **PowerShell** ou **Prompt de Comando** e altere o diretório (cd\) para o local do arquivo instalador .msi **BricsCAD®** .

Nota: No Windows 10, você pode acessar o menu de contexto **janela aqui Abrir PowerShell** ao clicar o botão-direito do mouse numa pasta no **Explorer**. Isso permite que você abra o **PowerShell** na janela atual, sem ter que alterar o diretório.

2 Digite **msiexec /i**, e então faça um dos seguintes:

- Digite o nome do arquivo do instalador **BricsCAD®** (por exemplo, BricsCad-V21.2.06-1-pt_US(x64).msi.
- Arraste o arquivo do instalador **BricsCAD®** da caixa de diálogo do Windows Explorer, para a janela de comando.

3 Digite **/qn** no prompt de comando.

4 (opção) Adicionar **ADDDESKTOPSHORTCUT=""** para suprimir a adição de um atalho de área de trabalho.

5 (opção) Adicionar **SHOWRELEASENOTES=""** para suprimir a exibição das Notas de Release quando a instalação for concluída.

6 (opção) Adicione **APPLICATIONFOLDER="your_app_folder"** para instalar **BricsCAD®** em uma pasta personalizada.

7 (opção) Adicionar **BRXLICENSEDESTFILE="path_to_lic_file"** para especificar uma pasta para o arquivo de licença se alguma das propriedades **BRXLICENSEKEY**, **BRXLICENSEFILE** ou **BRXLICENSESERVER** estiver definida.

O padrão é **APPLICATIONFOLDER\BricsCad.lic**.

8 (opção) Adicionar **"BRXLICENSEKEY="sua_chave_de_licença"** para ativar a licença e armazená-la na pasta especificada por **BRXLICENSEDESTFILE**.

9 (opção) Adicionar **BRXLICENSEFILE="seu_arquivo_de_licença"** para copiar o arquivo de licença para a pasta especificada por **BRXLICENSEDESTFILE**.

10 (opção) Adicionar **BRXLICENSESERVER="host"**, ou **BRXLICENSESERVER="port@host"** (*) para criar um arquivo de licença na pasta especificada por **BRXLICENSEDESTFILE**, contendo a especificação do host do servidor.

(*) Se o servidor de licenças estiver usando uma porta diferente da porta padrão 5053.

11 (opção apenas para 32-bit) Adicionar **INSTALLVBA="0"** ou **INSTALLVBA=""** para não instalar o VBA.

A entrada completa no prompt de comando pode ser: **msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn ADDDESKTOPSHORTCUT="" SHOWRELEASENOTES=""**.

12 Pressione **Enter** para iniciar a instalação.

Nota:

- Para mais opções, digite: **msiexec /?** no prompt de comando.
- No site da Microsoft, você pode encontrar mais informação sobre o comando **msiexec**.

2.1.4 Modo trial (teste)

BricsCAD® pode ser executado no modo de teste (trial) por 30 dias. Após esse período, você precisa ativar uma licença comercial válida.

2.1.4.1 Executar BricsCAD no modo Trial

Cada vez que você começa **BricsCAD®** no modo de teste, a caixa de diálogo **BricsCAD - Teste Gratuito** é aberta, indicando os dias restantes.

Siga um dos procedimentos:

- Clique no botão **Continuar** para abrir **BricsCAD®** no modo trial.
- Clique o botão **Entrar Licença...** para ativar uma licença Individual ou em Volume, ou para usar uma Licença de Rede.

2.2 Ativar e licenciar BricsCAD®

2.2.1 Visão geral das opções de licenciamento

BricsCAD® oferece várias opções de licenciamento.

2.2.1.1 Níveis de licença

Existem 5 níveis de licença para **BricsCAD®**:

- **BricsCAD® Lite**
- **BricsCAD® Pro**
- **BricsCAD® BIM**
- **BricsCAD® Mechanical**
- **BricsCAD® Ultimate**

1 BricsCAD® Lite

Se você criar apenas projetos 2D gerais, **BricsCAD® Lite** pode ser a escolha certa para você. Em comparação com **AutoCAD LT®**, este também permite visualizar, mover, copiar, ou espelhar modelos 3D. No entanto, você não poderá editar o modelo, ou criar novas entidades em 3D. **BricsCAD® Lite** também executa rotinas Lisp. A linguagem Lisp permite que usuários de **BricsCAD®** carreguem rotinas personalizadas ou pequenas aplicações que serão executados no **BricsCAD®**.

2 BricsCAD® Pro

BricsCAD® Pro inclui modelagem direta 3D com um completo sistema de restrição 3D, renderização com materiais e iluminação, e acesso a muitas aplicações de terceiros.

3 BricsCAD® BIM

BricsCAD® BIM inclui todos os recursos de **BricsCAD® Pro**. Você pode usar suas habilidades atuais em CAD para passar facilmente à criação de modelos reais de BIM em tempo recorde. Um produto, um fluxo de trabalho, tudo no formato *.dwg, padrão na indústria.

4 BricsCAD® Mechanical

Além de todos os recursos de **BricsCAD® Pro**, **BricsCAD® Mechanical** inclui um kit de ferramentas de design mecânico completo, tudo no formato *.dwg, padrão do setor, incluindo funcionalidade de Sheet Metal.

5 BricsCAD® Ultimate

BricsCAD® Ultimate combina os recursos de **BricsCAD® BIM** e **BricsCAD® Mechanical**.

2.2.1.2 Tipos de licença

Existem 2 tipos de licença para **BricsCAD®**:

- Licença perpétua.
- Licença de aluguel.

1 Licença perpétua

Quando você compra uma licença perpétua, você é proprietário permanente da licença do software.

BricsCAD® As edições executam no Microsoft Windows, macOS e Linux.

Existem 2 opções de Manutenção:

- A opção com Manutenção inclui atualizações automáticas e suporte.
- Comprar upgrades a seu critério, quando for conveniente.

2 Licença de aluguel

Quando você escolhe a opção de aluguel, você tem acesso à versão mais recente do **BricsCAD®** por 1 ano. Esta inclui Suporte Prioritário e acesso a todas as novas versões lançadas durante o período de vigência da sua assinatura.

2.2.1.3 Opções adicionais de licença

As opções de licença adicionais incluem licenciamento acadêmico, de usuário único, em rede e em volume.

1 Licença Acadêmica

Estudantes, professores, instrutores, e laboratórios das instituições educacionais tem acesso gratuito a todos os softwares **BricsCAD**® por 12 meses, renováveis. Você pode renovar anualmente mostrando seu vínculo atual com a instituição. Registre-se como estudante professor, ou escola, com suas informações acadêmicas.

2 Licença de usuário único

O contrato de licença isolada é para um único usuário. Cada licença de usuário único permite 2 ativações. Você pode ativar o BricsCAD em duas máquinas, por exemplo, uma estação de trabalho no escritório e um laptop para o campo. No entanto, apenas uma licença pode ser usada de cada vez. A licença isolada está disponível tanto para uso perpétuo como para aluguel. As licenças acadêmicas são sempre licenças de usuário único.

3 Licenca de Rede

O acordo de licenciamento em Rede permite que múltiplos usuários acessem o **BricsCAD**® dentro de uma LAN (Rede Local). O número de estações adquiridas para operar na Rede define o número de usuários que podem acessar o software simultaneamente. A opção de licença de Rede está disponível para licenciamento perpétuo ou de aluguel, e inclui um contrato de manutenção de um ano. O licenciamento de Rede também está disponível para instituições que usam uma licença de Uso Acadêmico.

4 Licença em Volume

O contrato de licenciamento por volume permite que você use **BricsCAD**® dentro de uma base de usuários definida. Você pode instalar o software usando uma única chave de licença válida, que vai ativar cada estação de usuário. A opção de licença por Volume está disponível para licenciamento perpétuo ou de aluguel, e inclui um contrato de manutenção de um ano. É a forma padrão usada para licenças acadêmicas institucionais.

2.2.2 BricsCAD® and AutoCAD® feature comparison

BricsCAD® versus AutoCAD® feature comparison.

Drawings							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Native DWG file format (thru AutoCAD 2021)	√	√	√	√	√	√	√
Password Protection	√	√	√	√	√	o	o
eTransmit	√	√	√	√	√	√	√
Sheet Set Manager	√	√	√	√	√	√	√
Geographic Location	√	√	√	√	√	√	√
Drawing Compare	√	√	√	√	√	√	√
Cloud Collaboration	√	√	√	√	√	√	√

Interface							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Dark and Light Color Themes	√	√	√	√	√	√	√
Industry standard commands, aliases, and variables	√	√	√	√	√	√	√
Command Line (Autocomplete, Midstring, Clickable options)	√	√	√	√	√	√	√
Start tab	√	√	√	√	√	√	√
File (Drawing) tabs	√	√	√	√	√	√	√
Model and Layout tabs	√	√	√	√	√	√	√
Layout Manager	√	√	√	√	√	o	o
Menu bar	√	√	√	√	√	√	√
Toolbars	√	√	√	√	√	√	√

Interface							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Ribbon	√	√	√	√	√	√	√
Tool Palettes	√	√	√	√	√	√	√
Contextual Right-click Menus	√	√	√	√	√	√	√
Contextual Quad Menu	√	√	√	√	√	o	o
Status Bar	√	√	√	√	√	√	√
Workspaces	√	√	√	√	√	√	√
Searchable Settings (Variables) Dialog Box	√	√	√	√	√	o	o
Lockable Interface	√	√	√	√	√	√	√
Clean Screen Mode	√	√	√	√	√	√	√
Drawing Explorer	√	√	√	√	√	o	o

Interface							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Customizable User Interface (CUI)	√	√	√	√	√	√	√
Structure Browser	√	√	√	√	√	o	o

Selection, snap & track							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Selection Filtering (QSELECT)	√	√	√	√	√	√	√
Add Selected	√	√	√	√	√	√	√
Select Similar	√	√	√	√	√	√	√
Selection Cycling	√	√	√	√	√	√	√
Entity Snaps (Osnaps)	√	√	√	√	√	√	√
Polar Tracking	√	√	√	√	√	√	√

Selection, snap & track							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Zintersection 3D snap	√	√	√	√	√	o	o
Nearest Distance	√	√	√	√	√	o	o
Adaptive Gridsnap	√	√	√	√	√	o	o
Real-time boundary detection	√	√	√	√	√	o	o
2D Dynamic UCS	√	√	√	√	√	o	o
3D Dynamic UCS	√	√	√	√	√	o	√

Drawing tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Line, Polyline, 3D Polyline, Spline, Ray, Xline	√	√	√	√	√	√	√

Drawing tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Multiline and Multiline Styles	√	√	√	√	√	o	√
Circle, Arc, Ellipse, Elliptical Arc	√	√	√	√	√	√	√
Polygon, Rectangle, Region, Donut, Point	√	√	√	√	√	√	√
Helix	√	√	√	√	√	o	√
2D Parameters, Constraints	√	√	√	√	√	o	√
2D Autoconstrain	√	√	√	√	√	2	2

Annotation tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Dimensions, Dimension Styles	√	√	√	√	√	√	√
Multi-leaders, Multi-leader Styles	√	√	√	√	√	√	√
Single and Multi-line text, Text Styles, Text Editor	√	√	√	√	√	√	√
Hatches, Gradients, Hatch/ Gradient Editor	√	√	√	√	√	√	√
Wipeouts	√	√	√	√	√	√	√
Revision Clouds	√	√	√	√	√	√	√
Attributes, Attribute Editor	√	√	√	√	√	√	√
Tables, Table Styles	√	√	√	√	√	√	√
Fields	√	√	√	√	√	√	√

Annotation tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Date Extraction	√	√	√	√	√	o	√
Data Linking	√	√	√	√	√	√	√
Annotation Scaling	√	√	√	√	√	√	√
Spell Check	√	√	√	√	√	√	√

Editing tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Move, Copy, Rotate, Scale, Stretch, Mirror, Align	√	√	√	√	√	√	√
Trim, Extend, Lengthen, Break, Join	√	√	√	√	√	√	√
Fillet, Chamfer	√	√	√	√	√	√	√

Editing tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
CopyGuided	√	√	√	√	√	o	o
MoveGuided	√	√	√	√	√	o	o
Grip Edit	√	√	√	√	√	√	√
Properties panel	√	√	√	√	√	√	√
Manipulator (3D Gizmo)	√	√	√	√	√	o	√
Drawordder	√	√	√	√	√	√	√
Undo Per Entity (Property History)	√	√	√	√	√	o	o
Optimize	√	√	√	√	√	o	o
Simplify	√	√	√	√	√	o	o

Reference tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Blocks, Block Editor	√	√	√	√	√	√	√
Dynamic Blocks	1	1	1	1	1	√	√
Block Face Camera	√	√	√	√	√	o	o
Blockify	√	√	√	√	√	o	o
Associative Arrays	√	√	√	√	√	√	√
Dwg (Xref) Attach, Clip, Bind, Edit	√	√	√	√	√	√	√
PDF Attach, Clip	√	√	√	√	√	√	√
Image Attach, Clip, Adjust, Transparency	√	√	√	√	√	√	√
Point Clouds Colormap	√	√	√	√	√	o	√

Visualization tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Layers, Layer States	√	√	√	√	√	√	√
Transparency	√	√	√	√	√	√	√
Visual Styles	√	√	√	√	√	o	√
LookFrom (ViewCube™)	√	√	√	√	√	o	√
Match Perspective	√	√	√	√	√	o	o
Lights	o	√	√	√	√	o	√
Sun Properties	o	√	√	√	√	o	√
Materials	o	√	√	√	√	o	√
Render	o	√	√	√	√	o	√
Animation Editor	o	√	√	√	√	o	o
Walkthrough navigation	o	√	√	√	√	o	√

Visualization tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
ANIPATH command	o	√	√	√	√	o	√

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
ACIS Modeling	o	√	√	√	√	o	√
Solid and Surface Modeling	o	√	√	√	√	o	√
Solid Primitives	o	√	√	√	√	o	√
Mesh Primitives	o	√	√	√	√	o	√
Polysolid	o	√	√	√	√	o	√
Union, Subtract, Intersect, Interfere	o	√	√	√	√	o	√

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Extrude, Revolve, Loft, Sweep, Thicken	o	√	√	√	√	o	√
Twist	o	√	√	√	√	o	o
Fillet, Chamfer	o	√	√	√	√	o	√
Slice	o	√	√	√	√	o	√
Multi-slice	o	√	√	√	√	o	o
T Connect, L Connect, Connect with Nearest	o	√	√	√	√	o	o
Extract Edges	o	√	√	√	√	o	√
Direct modeling	o	√	√	√	√	o	√
Deformable (Freeform) Modeling	o	√	√	√	√	o	√
Audit, Simplify	√	√	√	√	√	o	√

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Stitch	o	√	√	√	√	o	√
3D Parameters, Constraints	o	√	√	√	√	o	√
3D Autoparametrize	o	√	√	√	√	o	o
3D Blockify and ParametricBlockify	o	√	√	√	√	o	o
3D Parametric Design Tables	o	√	√	√	√	o	o
Section planes	o	√	√	√	√	o	√
Drawing Views	o	√	√	√	√	o	√
3D Threads	o	√	√	√	√	o	o
Point Cloud Align	o	√	√	√	√	o	o

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Point Cloud Bubble Viewer	o	√	√	√	√	o	o

Civil tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
TIN Surface Modeling	o	√	√	√	√	o	o
TIN Volume Surface	o	√	√	√	√	o	o
Grading	o	√	√	√	√	o	o
Alignments	o	√	√	√	√	o	o
Corridors	o	√	√	√	√	o	o
Civil 3D Object Enabler	√	√	√	√	√	o	√
Architectural Desktop Object Enabler	√	√	√	√	√	o	√

Civil tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
CADWorx Object Enabler	√	√	√	√	√	o	√

Architectural tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
BIM features	o	o	√	o	√	o	√
Structural Modeling Toolset	o	o	√	o	√	o	√
Project Database	o	o	√	o	√	o	√
Library Database	o	o	√	o	√	o	√
Site, Building and Stories	o	o	√	o	√	o	√
Windows and Doors	o	o	√	o	√	o	√

Architectural tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Fully Parametric BIM Component Creation	o	o	√	o	√	o	o
3D Spaces	o	o	√	o	√	o	√
Quickdraw	o	o	√	o	√	o	o
Quickbuilding	o	o	√	o	√	o	o
Point Cloud Floor Detection	o	o	√	o	√	o	o
Point Cloud Projection	o	o	√	o	√	o	o
Point Cloud Geometry Detection	o	o	√	o	√	o	o

Architectural tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Point Cloud FitPlanar (geometry detection)	o	o	√	o	√	o	o
Propagate	o	o	√	o	√	o	o
Custom BIM Properties	o	o	√	o	√	o	o
Drawing Customizations	o	o	√	o	√	o	o
Analytical Model	o	o	√	o	√	o	o
Python Scripts (Beta)	o	o	√	o	√	o	o

Mechanical tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Mechanical Browser	o	√	√	√	√	o	o

Mechanical tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Parts Library	√	√	√	√	√	o	o
Piping	o	√	√	√	√	o	o
Assembly Modeling	o	o	o	√	√	o	o
Exploded Views	o	o	o	√	√	o	o
Kinematic Analysis	o	o	o	√	√	o	o
Bill of Materials	o	o	o	√	√	o	o
Sheet Metal Design	o	o	o	√	√	o	o
3D Solids to Sheet Metal Parts Conversion	o	o	o	√	√	o	o
Corner Relief for Adjacent Edge Flanges	o	o	o	√	√	o	o
Lofted Bends	o	o	o	√	√	o	o

Mechanical tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Coloring	o	o	o	√	√	o	o
Diagnostic for Invalid Sheet Metal Features	o	o	o	√	√	o	o
Batch Processing	o	o	o	√	√	o	o

Import, export, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Print/ Plot	√	√	√	√	√	√	√
Publish/ Batch Plot	√	√	√	√	√	√	√
CTB & STB Table Files	√	√	√	√	√	√	√
PC3 Printer Configuration Files	√	√	√	√	√	√	√

Import, export, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Named Page Setups	√	√	√	√	√	√	√
PDF Import, Export	√	√	√	√	√	√	√
DAE Import, Export	√	√	√	√	√	o	o
IFC Import, Export	o	o	√	o	√	o	√
3DM Import, Export	o	√	√	√	√	o	o
LandXML Import, Export	o	√	√	√	√	o	o
RFA Import	o	o	√	o	√	o	o
SKP Import	o	√	√	√	√	o	o
DGN Import	√	√	√	√	√	√	√
RVT Import	o	o	√	o	√	o	o
DWF Export	√	√	√	√	√	√	√

Import, export, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
SVG Export	√	√	√	√	√	o	o
FBX Export	√	√	√	√	√	o	o
STL Export	√	√	√	√	√	o	√
Point Cloud Export	o	√	√	√	√	o	o
Additional formats available with BricsCAD Communicator	o	√	√	√	√	o	o

Programming							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
ActiveX	√	√	√	√	√	o	√
Entity data editor	√	√	√	√	√	o	o
Script Recorder	√	√	√	√	√	o	√

Programming							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Full LISP Support (vl, vlr, vla and vlax)	√	√	√	√	√	o	√
Solutions Development System (SDS/ ADS)	√	√	√	√	√	o	√
DCL engine	√	√	√	√	√	o	√
COM API	o	√	√	√	√	o	√
Visual Basic for Applications (VBA)	o	√	√	√	√	o	o
VBA DVB projects support	o	√	√	√	√	o	o
.NET	o	√	√	√	√	o	√
BRX (ARX)	o	√	√	√	√	o	√
TX (Teigha eXtensions)	√	√	√	√	√	o	o

Programming							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
CUI menu files support, with Diesel expressions	√	√	√	√	√	√	√

Licensing							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Multi-platform Licenses	√	√	√	√	√	o	o
Perpetual	√	√	√	√	√	o	o
Perpetual + Maintenance	√	√	√	√	√	o	o
Subscription	√	√	√	√	√	√	√
Network Multi-user	√	√	√	√	√	o	o
Volume (Perpetual)	√	√	√	√	√	o	o

Licensing							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Academic	√	√	√	√	√	√	√

(1) You can manipulate dynamic block insertions but you cannot create dynamic block definitions.

(2) You can automatically apply geometric constraints but you cannot automatically apply a combination of geometric and dimensional constraints.

2.2.3 Trial: ativação online

Antes de Iniciar: A ativação da licença Trial de **BricsCAD®** requer uma conexão com a Internet.

1 Lançar **BricsCAD®**.

2 Clique em **Iniciar Avaliação** na caixa de diálogo **Lançador do BricsCAD**.

Result: A licença de avaliação é ativada através da internet.

2.2.4 Trial (teste): ativação offline (manual)

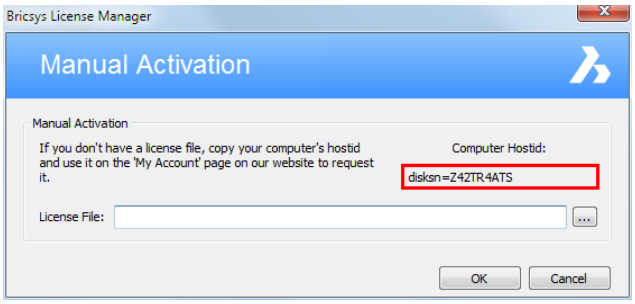
Se o seu computador não estiver conectado à Internet, você poderá ativar a licença manualmente por meio de outro computador conectado à Internet.

1 Lançar **BricsCAD®**.

2 Clique em **Ativar Licença** na caixa de diálogo Lançador do BricsCAD.

3 Clique em **Ativar Manualmente...** na caixa de diálogo **Gerenciador de Licenças Bricsys**.

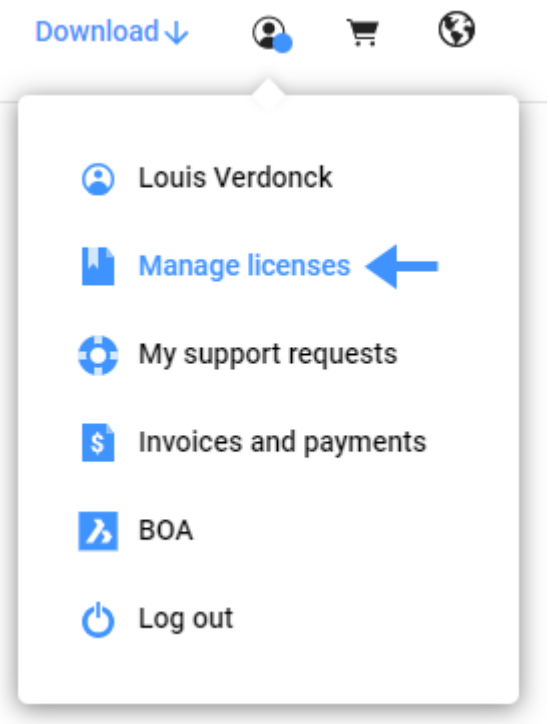
Result: A caixa de diálogo **Ativação Manual** é aberta.



4 Copie todo o conteúdo do campo **Computer Hostid**.

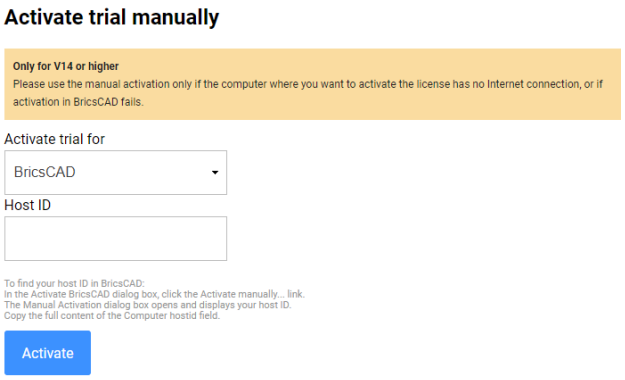
5 Faça login no website da Bricsys, com o computador conectado à internet.

6 Clique no ícone **Minha Conta** ( ) e selecione **Gerenciar Licenças** no menu.



7 Escolha **Ativar BricsCAD Manualmente**.

8 No campo **ID do host**, cole o ID do host do computador no qual você deseja instalar e clique**BricsCAD®** no botão **Ativar**.



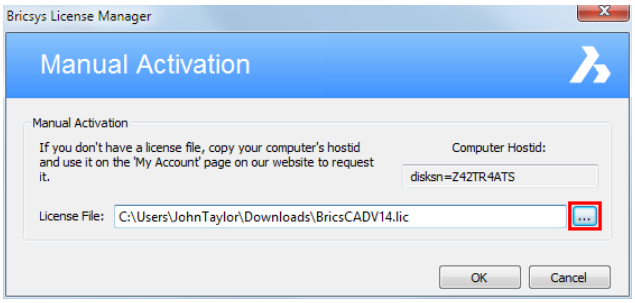
Result: Um arquivo de licença é gerado e disponibilizado para download.

9 Clique no botão **Baixar Arquivo de Licença**.

Result: O arquivo de licença é baixado para sua pasta de download padrão.

10 Na caixa de diálogo **Ativação Manual**, clique no botão **procurar** (...).

11 Abra a pasta para a qual você copiou o arquivo de licença e selecione o arquivo.



12 Clique o botão **OK**.

Result: **BricsCAD®** é lançado.

2.2.5 Licença única ou por volume: ativação online

Antes de Iniciar: A ativação da licença Trial de **BricsCAD®** requer uma conexão com a Internet.

1 Lançar **BricsCAD®**.

Nota: Quando o período de avaliação expirar, clique em **Ativar Licença** na caixa de diálogo **Lançador BricsCAD** e prossiga para a etapa 5.

2 Inicie o Comando LICENSEMANAGER.

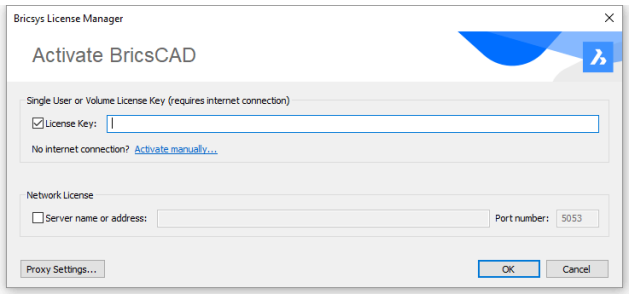
Result: A caixa de diálogo Gerenciador de Licenças Bricsys é exibida.

3 Clique em **Gerenciar Licença**.

Result: A caixa de diálogo **Informação da Licença** é exibida.

4 Clique no botão **Modificar**.

Result: A **Gerenciador Licenças Bricsys** > **Ativar BricsCAD** a caixa de diálogo exibe:



5 Cole a sua chave de licença no campo **Chave de Licença**.

6 Clique o botão **OK**.

Result:

- Sua chave de licença está registrada pela Internet.
- Uma caixa de diálogo é exibida quando a chave de licença é ativada com êxito.

7 Leia o texto na caixa de diálogo e clique no botão **Eu entendo**.

2.2.6 Licença isolada ou de Volume: ativação off-line (manual)

Se o seu computador não estiver conectado à Internet, você poderá ativar a licença manualmente por meio de outro computador conectado à Internet.

Nota: Você não pode desativar licenças ativadas manualmente.

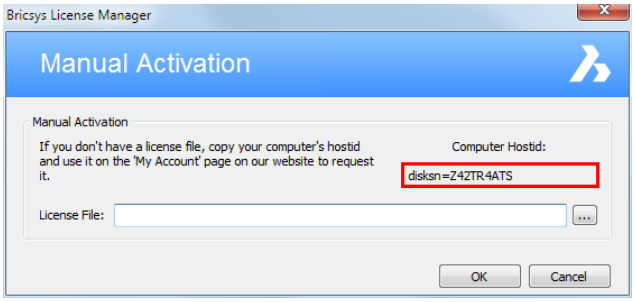
1 Lançar **BricsCAD®**.

2 Siga um dos procedimentos:

- Se você começar **BricsCAD®** pela primeira vez, vá para a próxima etapa.
- Se você começar **BricsCAD®** no modo Trial (avaliação), clique em **Entrar Licença...** na caixa de diálogo **Avaliação Gratuita**.
- Se você substituir ou atualizar uma licença existente, inicie o comando LICENSEMANAGER, clique em **Gerenciar licença** e clique em **Modificar...**.

3 Clique em **Ativar Manualmente...** na caixa de diálogo **Ativar BricsCAD**.

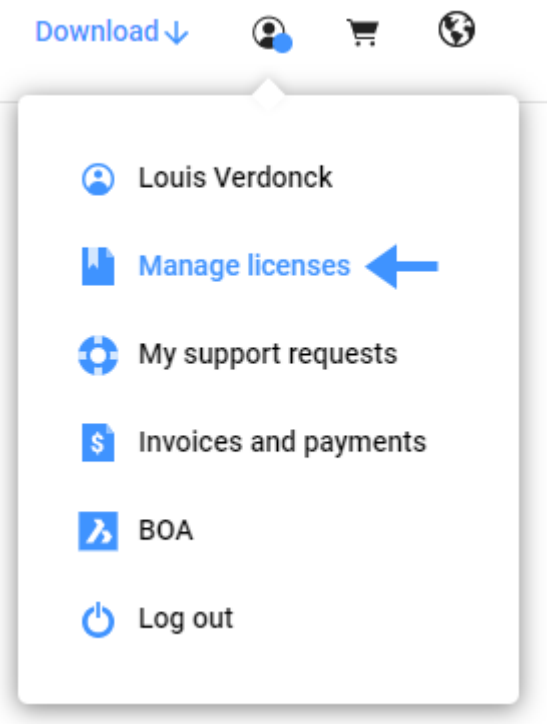
Result: A caixa de diálogo **Ativação Manual** é aberta.



4 Copie todo o conteúdo do campo **Computer Hostid**.

5 Faça login no website da Bricsys, com o computador conectado à internet.
Certifique-se de fazer login com a conta Bricsys que contém suas licenças.

6 Clique no ícone **Minha Conta** ( ) e selecione **Gerenciar Licenças** no menu.



7 Passe o cursor do mouse sobre a chave da licença que você deseja ativar e clique no ícone do lápis (✎).



Result: Os **Detalhes da Licença** são exibidos.

8 Clique em **Ativação Manual**.

9 No campo **ID do host**, cole o ID do host do computador**BricsCAD®** no qual você deseja instalar.

10 Clique o botão **Ativar**.

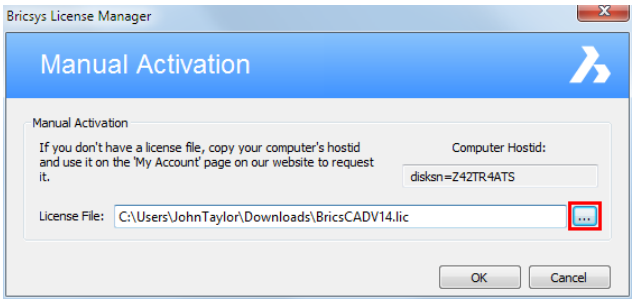
Result: Um arquivo de licença é gerado e disponibilizado para download.

11 Clique no botão **Baixar Arquivo da Licença**.

Result: O arquivo de licença **BricsCADVxx.lic** é baixado para sua pasta de download padrão.

12 Copie o arquivo de licença para o computador no qual você deseja instalar o **BricsCAD®**.

13 Na caixa de diálogo **Ativação Manual**, clique no botão de procura (...). Abra a pasta onde você copiou o arquivo de licença **BricsCadvxx.lic** e selecione o arquivo.



14 Clique o botão **OK**.

Result: **BricsCAD®** é lançado.

2.2.7 Usando uma licença de Rede em um computador-cliente

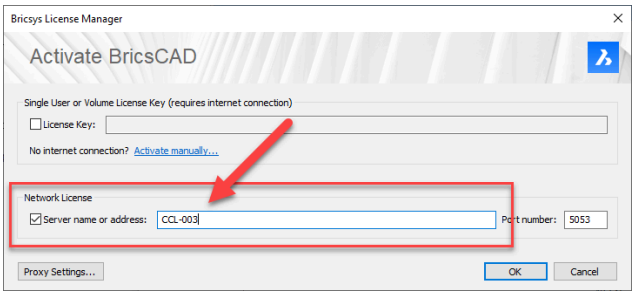
Uma vez que a licença de Rede está ativada no servidor de licenças, esta pode ser usada nos computadores-cliente.

1 Siga um dos procedimentos:

- Se você começar **BricsCAD®** pela primeira vez, clique no botão **Ativar Licença** na caixa de diálogo Lançador BricsCAD.
- Se você substituir ou atualizar uma licença existente, inicie o comando LICENSEMANAGER, clique em **Gerenciar licença** e clique em **Modificar**.

2 Marque a opção **Licença de Rede** na caixa de diálogo **Ativar BricsCAD**.

3 Digite o nome do host ou IP address do servidor de licenças no campo **Server name or address** (Nome do Servidor e endereço).



4 (Opcional) Digite o número da porta no campo **Número da Porta**.

5 Se seu computador se conecta ao servidor de licenças, através de um servidor proxy, clique no botão **Configurações de Proxy...** para definir as configurações de proxy para uma licença de Rede.

6 Clique o botão **OK**.

2.3 Configurar um Servidor de Licenças de Rede

2.3.1 Servidor de licenças de Rede

Para disponibilizar uma licença de Rede no **BricsCAD®** em um computador-cliente, o administrador da rede precisa instalar primeiro um Network License Server (servidor de licenças de rede) da **Reprise Software**.

Bricsys Network License Manager é um programa separado que instala o servidor. Depois de iniciar o servidor de licenças de Rede, você pode configurá-lo através de uma interface web.

Assim como **BricsCAD®** precisa de um arquivo de licença para ser executado, o servidor de licenças de Rede precisa de um arquivo de licença de rede que você possa ativar online ou manualmente.

Você pode ativar uma licença de Rede apenas uma vez, e a partir daí esta fica bloqueada para uso neste servidor.

Uma vez que a licença de Rede está ativada e o servidor está iniciado, você pode usar a licença de Rede no **BricsCAD®** em um computador-cliente.

Nota: Reprise Software não oferece um servidor de licenças de Rede para computadores Mac. No entanto, você pode instalar um servidor de licenças de Rede em Windows ou em Linux, para servir as licenças em computadores-cliente Mac.

2.3.2 Guia rápido sobre como usar uma licença de rede

Em primeiro lugar, certifique-se de que o administrador de rede/TI esteja envolvido no processo.

2.3.2.1 1. Ações da máquina atuante como servidor

1 Baixe o Bricsys Network License Manager e instale com as configurações padrão:

https://www.bricsys.com/bricscad/tools/Bricsys-NetworkLicenseManager.msi

Informações relacionadas às políticas de segurança do servidor:

após a instalação, o RLM.exe deve ter permissão para usar 3 portas: **5053** para licença, **5054** para interface da Web, uma **porta alocada dinamicamente** pelo Windows para o servidor ISV que pode ser definida como fixa após a ativação, conforme descrito no artigo Configurações Avançadas.

2 Quando a caixa de diálogo de ativação for exibida, ative usando a chave de licença da sua conta Bricsys.

Informações relacionadas às políticas de segurança do servidor:

a aplicação ativadora é *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager \actNetworkLicense.exe* e precisa alcançar **license.bricsys.com/actpro** port **80**

3 A página de administração do gerenciador de licenças RLM está no **localhost:5054**

2.3.2.2 2. Ações da máquina cliente

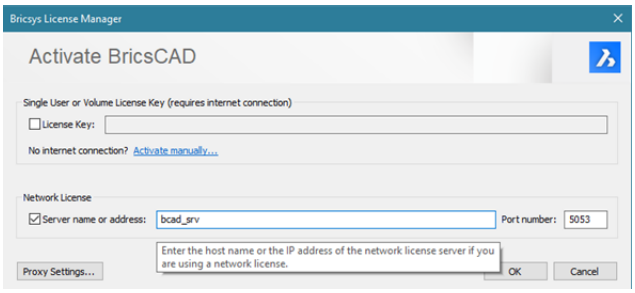
A máquina cliente deve ser:

- Na mesma **LAN** com a máquina RLM, fisicamente ou por conexão VPN.
- Capaz de se comunicar com a máquina RLM na porta **5053** pelo protocolo **TCP**.

1 Instale **BricsCAD®** com as configurações padrão.

2 Iniciar **BricsCAD®** e ativar a licença de Rede: <servername>/<ip number> e <port>

(pode não haver necessidade disso, pois **BricsCAD®** vai verificar a rede em busca de um servidor de licenças no início)



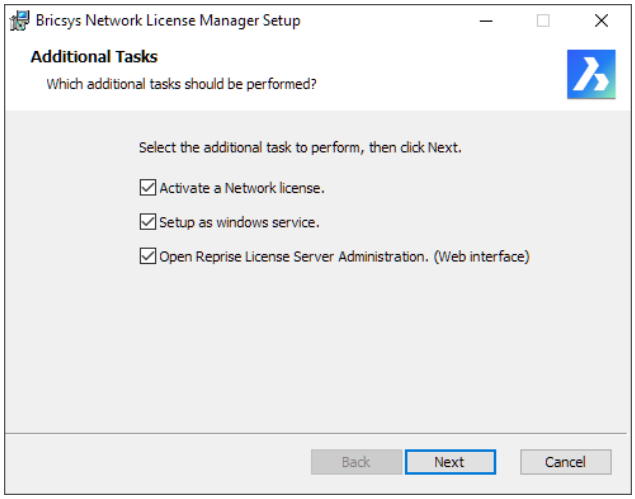
2.3.3 Configurar um servidor de licenças de Rede em Windows

2.3.3.1 Baixando o Bricsys Network License Manager

Faça o download do Bricsys Network License Manager (Windows) a partir do seguinte link:
https://www.bricsys.com/bricscad/tools/Bricsys-NetworkLicenseManager.msi

2.3.3.2 Nova instalação do Bricsys Network License Manager

Na caixa de diálogo **Additional Tasks** (Tarefas Adicionais), certifique-se de marcar todas as caixas de seleção (consulte a imagem do Bricsys Network License Manager Setup).



- **Ativar uma licença de rede**

  Após a instalação, o **Bricsys Network License Manager** abre e pede para ativar uma licença de rede. Você pode abrir o **Bricsys Network License Manager** mais tarde clicando no atalho Ativação de licença no menu Iniciar.
- **Configurar como um 'serviço' do Windows**

  Ao ser configurado como um serviço, cada vez que você iniciar o Windows, o servidor de licenças será automaticamente iniciado. Você pode fazer isso mais tarde com o seguinte comando no prompt de comando elevado: rlm.exe-dlog rlm.log-install_service
- **Abrir o Reprise License Server Administration (interface Web)**

Após a instalação, a interface da Web Reprise License Server Administration é aberta. Permite que você gerencie a licença de Rede. Mais tarde, você pode abrir a Administração do Servidor de Licenças da Reprise clicando no atalho Administração do Servidor de Licenças no menu Iniciar.

2.3.3.3 Atualizar o Bricsys Network License Manager

1 Baixe e instale o Bricsys Network License Manager no computador que será usado como servidor de licenças de rede.

2 Certifique-se de desmarcar todas as caixas de seleção ao lado das **Additional Tasks** (Tarefas Adicionais).

Nota: A instalação reinicia o servidor de licenças.

2.3.3.4 Instalação a partir da linha de comando

Você também pode iniciar o Gerenciador de licenças de rede a partir de uma janela de comando.

Por padrão, as opções **Ativar uma licença de Rede**, **Configurar como um Serviço Windows** e **Abrir Reprise License Administrator** (interface Web) estão desmarcadas quando instalado através da linha de Comando. Para habilitar essas opções, use os respectivos parâmetros ACTIVATELICENSE, INSTALLSERVICE e OPENWEBINTERFACE.

Por exemplo, **msiexec /qb /i Bricsys-NetworkLicenseManager.msi ACTIVATELICENSE=1 INSTALLSERVICE=1** vai apenas mostrar o diálogo de ativação e instalar como um serviço, sem mostrar a interface web.

Nota:

- Para mais opções, digite: **msiexec /?** no prompt de comando.
- No site da Microsoft, você pode encontrar mais informação sobre o comando **msiexec**.

2.3.4 Configurar um servidor de licenças Linux

Antes de Iniciar: Baixe e instale o Bricsys Network License Manager no computador que será usado como servidor de licenças de rede.

1 Baixar Bricsys Network License Manager (Linux)

2 Extraia **Bricsys-NetworkLicenseManager.tgz** para qualquer pasta (linha de Comando: tar xvzf Bricsys-NetworkLicenseManager.tgz).

3 Inicie a aplicação RLM naquela pasta (linha de comando na pasta: ./rlm).

4 No seu Navegador de Internet vá até a página Web **http://localhost:5054**.

5 Clique no botão **System Info** (Informações do sistema) à esquerda na página Web. Copie um hostid ethernet e use isso para ativar sua licença na página **Minha Conta** no site da Bricsys.

6 Quando você baixar o arquivo da licença do site da Bricsys, armazene-o na pasta onde você extraiu o arquivo acima citado Bricsys-NetworkLicenseManager.tgz.

7 Clique no botão **Reread/Restart Servers** (Reler/Reiniciar).

8 Clique o botão **Status** para ver o status do servidor, na próxima página, clique no botão **bricsys** na tabela, para ver as licenças Bricsys.

9 Clique no botão **RLM Manual...** para aprender em detalhes como configurar o servidor de licenças.

Entre outras coisas, a página explica como iniciar o RLM automaticamente quando o servidor é inicializado.

Result: Uma vez que a licença de Rede estiver ativada e o servidor iniciado, o servidor está pronto para servir as licenças. Agora é possível usar a licença de Rede no **BricsCAD®** em um computador cliente.

2.3.5 Licença de rede: ativação online

A ativação da licença de **BricsCAD®** requer uma conexão com a Internet.

1 Clique no atalho **Ativação da Licença** no menu **Iniciar** do sistema operacional.

C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe

A caixa de diálogo **Bricsys License Manager - Activate Network License** abre.

2 Entre a nova licença de rede no campo **Chave da Licença** e clique a **OK**.

3 Clique no atalho de **License Server Administration** no menu **Iniciar** do sistema operacional para abrir o **Reprise License Server Administration**.

4 Clique na opção **Reread/Restart Servers** no menu da esquerda.

2.3.6 Licença de rede: ativação off-line (manual)

Se o seu computador não estiver conectado à Internet, você poderá ativar a licença manualmente por meio de outro computador conectado à Internet.

Nota: Você não pode desativar licenças ativadas manualmente.

1 Clique no atalho **Ativação da Licença** no menu **Iniciar** do sistema operacional.

C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe

A caixa de diálogo **Bricsys License Manager - Activate Network License** abre.

2 Copie todo o conteúdo do campo **Computer Hostid**.

3 Faça login no website da Bricsys com o computador conectado à internet.

4 Clique no ícone **Minha Conta** () e selecione **Gerenciar Licenças** no menu.

Certifique-se de fazer login com a conta Bricsys (na sua empresa) onde está anotada a licença de Rede. Entre em contato com seu gerente de conta Bricsys, se for necessário.

5 Passe o cursor do mouse sobre a chave da licença que você deseja ativar e clique no ícone do lápis ().

Os **Detalhes da Licença** são exibidos.

6 Clique em **Ativação Manual**.

7 No campo **Host ID**, cole o ID do host do computador **BricsCAD®** no qual você deseja instalar.

8 Clique o botão **Ativar**.

Um arquivo de licença é gerado e disponibilizado para download.

9 Clique no botão **Baixar Arquivo da Licença**.

O arquivo de licença **BricsCAD.lic** é baixado para sua pasta padrão de download.

10 Copie o arquivo de licença para o servidor, na pasta de instalação **Bricsys Network License Manager** (por padrão: *C:\Program Files (x86) \Bricsys\Bricsys Network License Manager.*).

11 Execute a função **Reread/restart servers** na Interface da web do RLM: localhost:5454, para que o gerenciador de licenças Reprise possa usar o arquivo *.lic .

2.3.7 Administracao de servidor de licença de Rede

Quando o servidor de licenças de Rede (também chamado RLM License Server) estiver executando, pode ser configurado usando a interface web do Reprise License Server Administration.

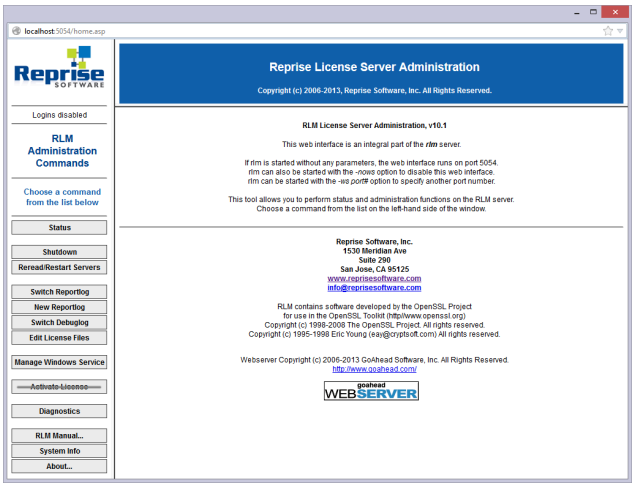
2.3.7.1 Abrir a interface web

1 Se o servidor de licenças de rede não estiver em execução, clique no atalho **License Server** (Servidor de Licenças), no menu Iniciar (no Windows Vista ou superior: clique o botão direito no atalho e escolha > Executar como administrador).

Result: Uma janela de console é aberta.

2 Navegue até **http://localhost:5054**.

Result: O Reprise License Server Administration (interface web) abre.



Nota:

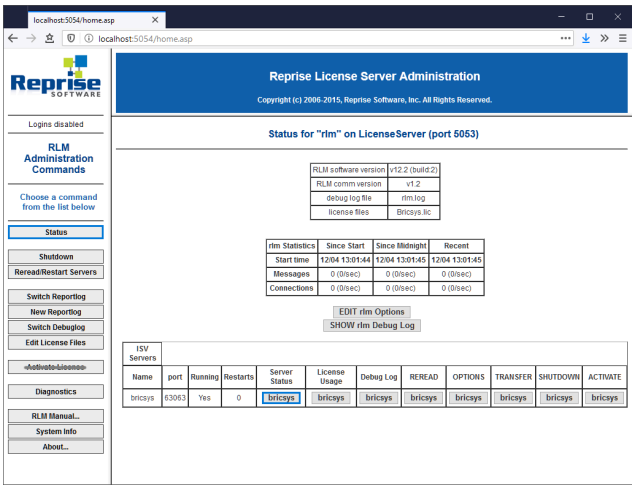
- Não use o botão de **Ativar Licença**!
- Para obter uma visão geral de todas as opções, clique no botão **RLM Manual...** à esquerda ou vá para o RLM Manual de Administração de Licenças.

2.3.7.2 Verificar o status da licença e servidor

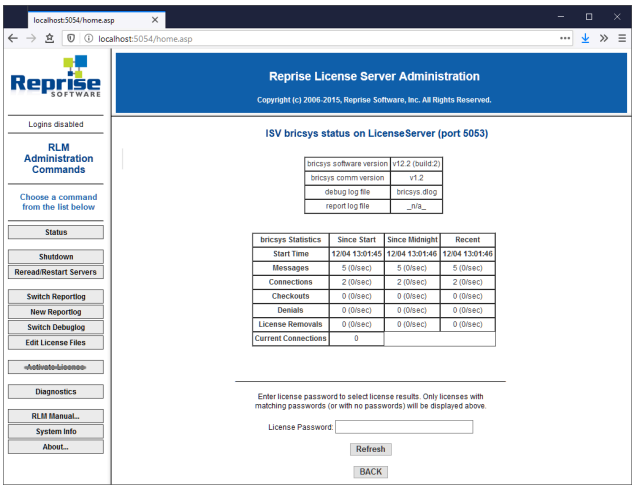
Você pode verificar o status de todas as licenças da Rede no servidor.

1 Clique no botão **Status** à esquerda.

Result: A página **Status for "rlm" on LicenseServer (port 5053)** se abre.



2 Clique no botão **bricsys** na coluna **Server Status** da tabela ISV Servers para ver o status da licença de Rede Bricsys.

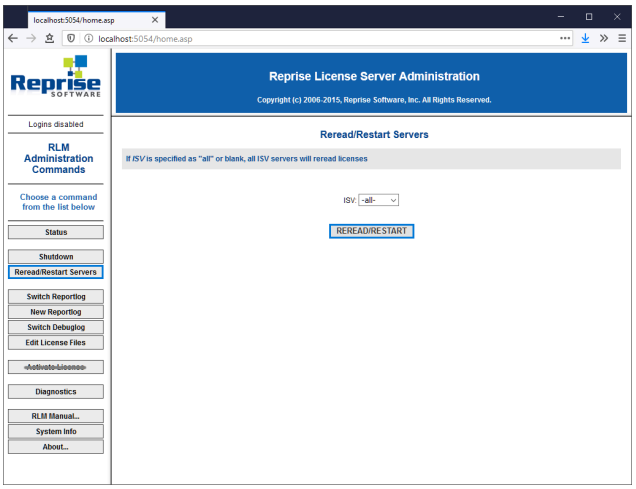


2.3.7.3 Reler a licença / Reiniciar o servidor

Depois de ativar uma licença de Rede enquanto o servidor estava funcionando, você precisa reler a licença / reiniciar o servidor.

1 Clique no botão **Reread/Restart Servers** (Reler/Reiniciar Servidores) à esquerda.

Result: A página **Reread/Restart Servers** é aberta.



2 Clique no botão **Reread/Restart Servers** (Reler/Reiniciar).

Result: O servidor reinicia e relê a licença de Rede.

2.3.8 Configurações avançadas

Assim que o servidor de licenças de rede estiver configurado, **BricsCAD®** poderá relatar o erro -17 ao tentar se conectar ao servidor de licenças.

Os motivos podem ser devidos à presença de software antivírus e/ou firewall com bloqueio, e/ou ainda um servidor em proxy.

2.3.8.1 Antivírus

Se um software antivírus estiver sendo executado no servidor e bloquear o programa do servidor de licenças (**rlm.exe** no Windows, **rlm** no Linux/Mac), você precisará desbloquear o serviço do RLM no seu antivírus.

Ver sua configuração antivírus sobre como desbloquear serviços.

2.3.8.2 Firewall

Se um firewall estiver instalado entre o servidor e o **BricsCAD®**, dois números de porta precisam ser abertos no firewall: o número de porta 5053 e o número da porta atribuída para o servidor de licença Bricsys

Esse número de porta é atribuído dinamicamente quando o servidor de licenças é iniciado. Para abrir esta porta no firewall, essa deve ser definida primeiro no arquivo Bricsys.lic, no servidor:

1 Abra a interface Web de administração do servidor de licenças (License Server Administration) em **http://localhost:5054**.

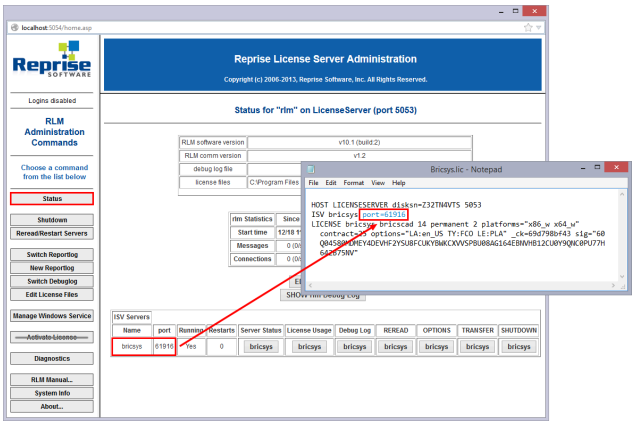
2 Clique no botão **Status** à esquerda. A página **Status for "rlm" on [LicenseServer] (port 5053)** se abre.

3 Procure o número da porta na tabela ISV Servers. O número da porta vai ser exibido na linha 'bricsys', na 2ª coluna.

4 Copie este número de porta.

5 Abra o arquivo Bricsys.lic em um editor de texto (por exemplo, Bloco de notas, Gedit,...). O local padrão do arquivo é C:\Program Files (x86)\Bricsys\Bricsys Network License Manager.

Você pode fixar o número da porta, modificando a linha bricsys ISV em: ISV bricsys port=[port number], onde [port number] deve ser substituído pelo número da porta copiado da tabela ISV Servers.



6 No firewall, abra ambos números 5053 e o número da porta que você adicionou no arquivo Bricsys.lic.

7 Encerre o servidor e reinicie o programa do Servidor de Licenças.

Nota: Se isso não resolver, adicione o programa de servidor de licenças (rlm.exe no Windows, rlm no Linux/Mac) como uma exceção para a lista das exceções do firewall, e então reinicie o servidor de licença novamente.

2.3.8.3 Servidor proxy

Se existir um servidor proxy entre **BricsCAD®** e o servidor de licenças, as configurações de proxy deverão ser configuradas em **BricsCAD®**.

Consulte as configurações de Proxy como definir as configurações de proxy permanentemente para uso da Licença de Rede.

2.4 Gerenciar licenças

2.4.1 Modificar uma BricsCAD® licença

Use este procedimento para fazer upgrade ou substituir uma licença.

1 Inicie o Comando LICENSEMANAGER.

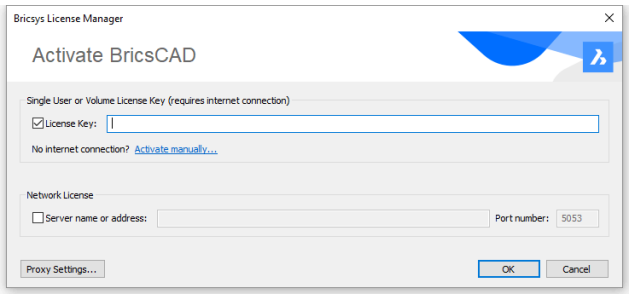
Result: A caixa de diálogo Gerenciador de Licenças Bricsys é exibida.

2 Clique em **Gerenciar Licença**.

Result: A caixa de diálogo **Informação da Licença** é exibida.

3 Clique no botão **Modificar**.

Result: A caixa de diálogo **Gerenciador de Licenças do BricsCAD** - **Ativar BricsCAD é exibida**.



4 Cole a sua chave de licença no campo **Chave de Licença**.

5 Clique o botão **OK**.

Result:

- Sua chave de licença está registrada pela Internet.
- Uma caixa de diálogo é exibida quando a chave de licença é ativada com êxito.

6 Leia o texto na caixa de diálogo e clique no botão **Eu entendo**.

2.4.2 Desativar ou revogar uma BricsCAD® licença

Use este procedimento para:

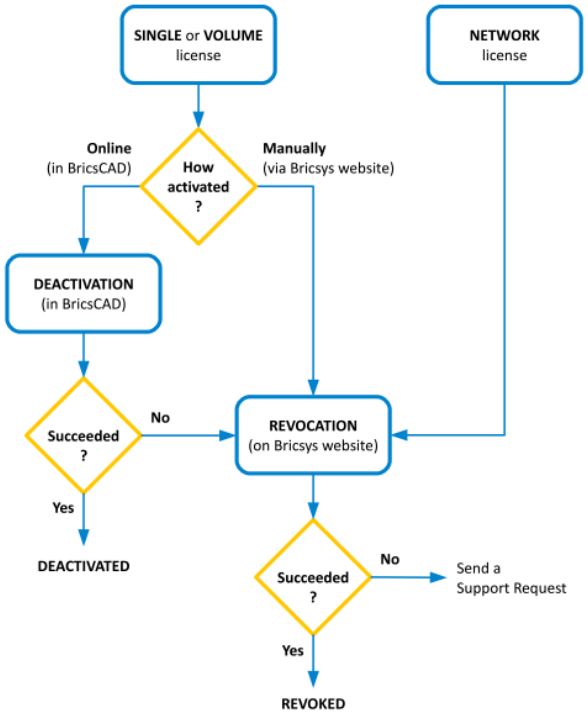
- Migrar uma licença para outro computador.
- Remova permanentemente **BricsCAD®** de um computador.
- Reative **BricsCAD®** após instalar um novo disco rígido, alterar partições, ou atualizar o sistema operacional.

Para desativar uma licença **BricsCAD®**, é necessária uma conexão com a internet. Não é possível desativar manualmente uma licença através de outro computador conectado à Internet.

Você não pode desativar:

- Licenças individuais e de volume ativadas manualmente.
- Licenças de rede.
- Licenças de teste.

O gráfico explica como desativar uma **BricsCAD®** licença. Se não for possível desativar uma **BricsCAD®** licença, você pode revogá-la.



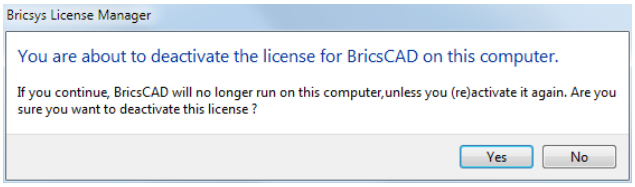
2.4.2.1 Desativar uma licença

1 Inicie o Comando LICENSEMANAGER. Clique na opção **Gerenciar licença** na caixa de diálogo **Gerenciador de Licenças Bricsys**.

Result: A caixa de diálogo **Informação da Licença** é exibida.

2 Se o computador se conectar ao servidor de licenças por meio de um servidor proxy, clique nas **Configurar Proxy...** para definir as configurações de proxy.

3 Clique em **Desativar...**



4 Clique o botão **Sim** na caixa de diálogo de confirmação para desativar a licença.

Result: A licença está desativada e pode ser ativada em outro computador.

2.4.2.2 Revogar uma licença

Apenas revogue uma licença se a desativação não for possível.

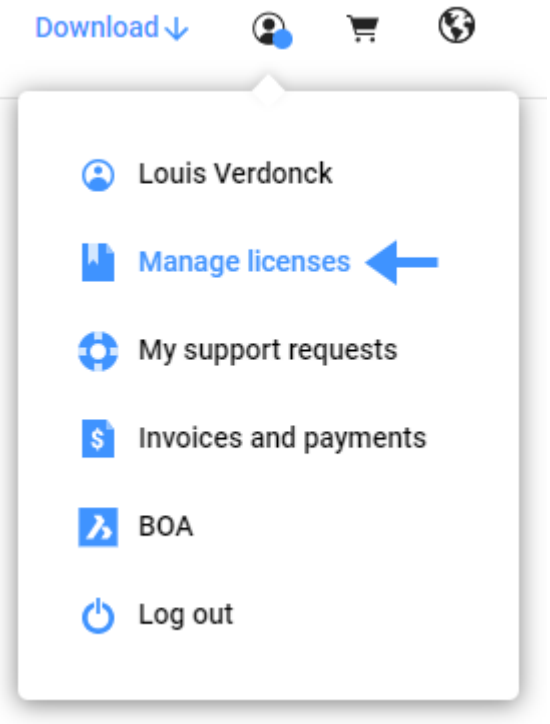
Use este procedimento para:

- Desative manualmente uma licença de Rede ou uma licença Isolada ou de Volume que tenha sido ativada manualmente.
- Reative **BricsCAD®** após uma falha no disco rígido.

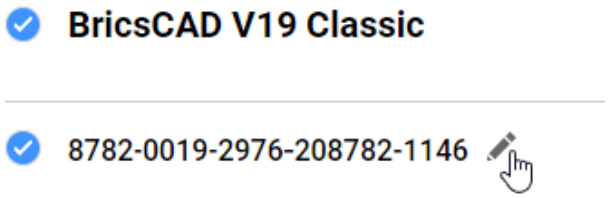
- Reative **BricsCAD**® após instalar um novo disco rígido, alterar partições, ou atualizar o sistema operacional.
- Migrar uma licença para outro computador.
- Remova permanentemente **BricsCAD**® de um computador.

1 Faça login no site da Bricsys.
Certifique-se de fazer o login com a conta Bricsys que contém o registro das suas licenças.

2 Clique no ícone **Minha Conta** ( ) e selecione **Gerenciar Licenças** no menu.



3 Passe o mouse sobre a chave de licença que você deseja revogar, e clique no ícone de um lápis ( ).



4 Role para baixo e clique no ícone **X** para revogar a licença.



2.4.3 Migrando uma licença Isolada ou de Volume para outro computador

Todas as suas licenças válidas podem ser visualizadas no site da Bricsys (requer Log in).

https://www.bricsys.com/protected/account/licenses.

1 Desative a licença no computador 'antigo'.

2 Inicie **BricsCAD**® no novo computador.

3 Ative a licença no computador 'novo'.

2.4.4 Migrar uma licença de Rede para outro servidor

Faça o seguinte:

1 Revogar a licença.

2 Ativa a licença no seu novo servidor.

3 Encerre o gerenciador de licenças no servidor antigo, e exclua o arquivo de licença no servidor antigo.

Nota: O número de revogações é limitado.

2.4.5 Agrupar licencas

2.4.5.1 Editando as opções

Você pode editar as opções de duas maneiras:

- No arquivo bricsys.opt, localizado na pasta de instalação do Network License Manager.
- Na interface web do Gerenciador de Licença de Rede (Network License Manager).
  - Navegue para *http://localhost:5054* no servidor de licenças.
  - Clique em **Status** no menu do lado esquerdo.
  - Clique em **bricsys** na coluna OPTIONS da linha ISV bricsys.

Depois de editado, acione Reread/Restart (reler/reiniciar) o Servidor da licença.

2.4.5.2 Opções comumente usadas

Grupos podem ser usados para definir uma lista de nomes de usuário, nomes de host ou endereços-IP.

O nome poderá ser usado posteriormente para aplicar regras para esse grupo.

		Sintaxe	Exemplo
GRUPO	Define um grupo de nomes de usuário.	GROUP nome lista-nomes-de-usuários	GROUP engenheiros userA userB userC
HOST_GROUP	Define um grupo de nomes de host.	HOST_GROUP nome lista-nomes-de-host	HOST_GROUP PrimPavimento maqA maqB maqC

		Sintaxe	Exemplo
INTERNET_GROUP	Define um grupo de endereços-IP.	INTERNET_GROUP nome list-endereços-IP	INTERNET_GROUP networkGhent 192.168.1.* 192.168.2.* O caractere curinga (*) pode ser usado em endereços IP.
RESERVE	Reserva várias licenças para um usuário específico (ou grupo) de usuários.	RESERVE num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	RESERVE 20 bricscad GROUP engenheiros
MAX	Limita o número de licenças disponíveis para um grupo específico grupo de usuários.	MAX num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	MAX 5 bricscad HOST_GROUP PrimPavimento
INCLUDE	Permite somente licenças de um produto para um grupo específico de usuários. Qualquer usuário não especificado pela linha INCLUDE não tem permissão para usar o produto.	INCLUDE product userhost|group|host_group|internet|internet_group|project who [id=nnn]	INCLUDE bricscad GROUP bricscadusers
EXCLUDE	Desautoriza licenças de um produto para um grupo específico de usuários.	EXCLUDE product user|host|group|host_group|internet|internet_group|project who [id=nnn]	EXCLUDE bricscad INTERNET_GROUP 1.1.*.*

Mais opções podem ser encontradas no manual do usuário final do RLM (em inglês), no parágrafo **The ISV Options File**.

2.4.5.3 Identificar um arquivo de licença

Desde a V15, um novo recurso no gerenciador de licenças (Reprise license manager) permite que você adicione um campo '_id' em seu arquivo de licença.

Com este '_id', você pode identificar cada arquivo de licença como um pool (grupo) de licenças separado.

Ao especificar um parâmetro [id=nnn] para uma OPTION adicionada, essa OPTION só terá efeito no arquivo de licença correspondente.

Exemplo de Uso: Você tem 2 licenças, uma com 10 estações para **BricsCAD® Lite** e outra com 10 estações para **BricsCAD® Pro**, e deseja especificar quais usuários podem usar qual nível.

OS ARQUIVOS DE LICENÇA

Lite 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=""

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: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=""

Agora a licença Lite é definida por id=1, e a licença Pro por id=2.

AS OPÇÕES (OPTIONS)

Se agora, por exemplo, você quiser permitir que somente os engenheiros da sua empresa usem a versão Pro, isso pode ser feito adicionando o seguinte às opções.

GROUP engenheiros userA userB userC

INCLUDE bricscad GROUP engineers id=2

Symptoms

Quero ajustar a forma como minhas licenças são reunidas.

Mais informação: http://www.reprisesoftware.com/RLM_Enduser.html

2.4.5.4 Agrupamento (pooling) de licenças pelo RLM_PROJECT

O seguinte se aplica aos casos em que vários níveis de licenças de rede são ativados no mesmo servidor RLM. Consequentemente, os usuários podem decidir qual nível de licença adquirir ao ativar **BricsCAD®**.

O usuário pode obter um nível específico de licença do **BricsCAD®**, com base na variável de ambiente RLM_PROJECT que deve corresponder às opções do servidor ISV. As estações de licença serão reservadas no servidor ISV para os usuários específicos do PROJECT por um IDentifier.

Nota: Esteja ciente de que as seguintes ações devem ser executadas por uma pessoa qualificada, do departamento de TI.

1. Ações do RLM server

1 Add _id=<number> identifier para arquivos de licença, apenas antes de _ck=xxxxxxx string:

……contract=999-9999-9999-8888 options="LA:xx_XX TY:NFR LE:PRO _id=1 _ck=4b07efdd90….

……contract=999-9999-9999-9999 options="LA:xx_XX TY:NFR LE:LIT _id=2 _ck=4b07efdd90…..

2 Adicione a opção RESERVE para projetos e IDs de licença ao servidor ISV, para todas as estações das licenças:

RESERVE 5 bricscad PROJECT bc_pro id=1

RESERVE 5 bricscad PROJECT bc_lite id=2

2. BricsCAD® ações do usuário

1 Crie um script que Defina uma variável temporária de ambiente RLM_PROJECT na máquina do usuário, o que permitirá que o BricsCAD negocie com o servidor RLM o pool de licenças PROJECT.

2 A execução de um script específico permitirá que o BricsCAD extraia o nível de licença desejado

Pode ser arquivos *.CMD (prompt de comando):

- @echo off
  set RLM_PROJECT=bc_lite
  call "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- @echo off
  set RLM_PROJECT=bc_pro
  call "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- ou arquivos *.PS1 (powershell):
  $env:RLM_PROJECT = "bc_pro"
  Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"
- $env:RLM_PROJECT = "bc_lite"
  Start-Process -FilePath "C:\Program Files\Bricsys\BricsCAD V21 en_US\bricscad.exe"

Desde que todas as estações da licença estão RESERVED nas opções RLM, **BricsCAD**® não poderá adquirir uma licença quando a variável RLM_PROJECT não existir, ou quando tiver um valor que não corresponda às opções do ISV server.

Aumentar as estações disponíveis acima de 5 ou baixar a contagem de RESERVE abaixo de 5 permitiria que algumas licenças estejam disponíveis livremente.

O RLM_PROJECT também funciona com as outras opções (EXCLUDE/INCLUDE/...) para que muitas configurações sejam possíveis.

2.4.6 Relatório de uso de licenças de rede

Existem várias possibilidades de verificar o uso da licença de rede:

- A partir da Interface do Usuário Web da RLM

  ir para localhost:5054/home.asp > Status > License Usage > bricsys
- A partir do arquivo "bricsys.dlog" , que contém a informação IN/OUT. Este arquivo está localizado na pasta RLM *C:\Program Files (x86)\ Bricsys\ Bricsys Network License Manager*
- Opção REPORTLOG. Esta opção é descrita nas páginas 77 e 103 no RLM Manual:

  http://www.reprisesoftware.com/RLM_License_Administration.pdf

  Ele deve ser adicionado ao arquivo **bricsys.opt** como este:

  REPORTLOG +D:\your_path\report_name.log

  A seguir, execute um Reler/Reiniciar Servidores.

  O sinal + significa que os novos dados são anexados ao arquivo, caso contrário, o arquivo será substituído.
- Pacote de Administração de Licenças RLM

  https://reprisesoftware.com/admin/software-licensing-downloads.php
- FlexReport JTB

  https://jtbworld.com/jtb-flexreport

2.5 Usando uma licença de Rede

2.5.1 Roaming

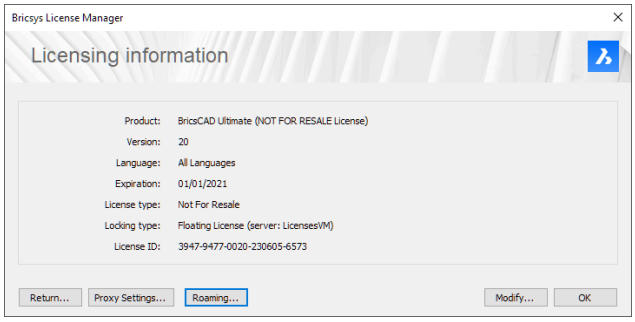
Com uma licença de rede, você pode definir uma licença flutuante para operar num computador que será posteriormente desconectado da rede. A licença resultante pode ser usada para o número de dias especificado quando a licença foi definida para roaming. Após esse período, a licença retorna automaticamente para a rede, de volta. Se necessário, você pode devolver a licença em roaming para o pool de licenças, antecipadamente.

O(s) arquivos de licença no servidor de licenças de Rede contêm um parâmetro *max_roam_count*. Este é definido como o número total de estações em uma licença de Rede, por padrão.

2.5.1.1 Roaming de uma licença de Rede em um computador-cliente

Uma licença de Rede pode ser transportada para ser usada **BricsCAD®** sem conexão com o servidor de licenças (trabalho fora do escritório, em casa,...).

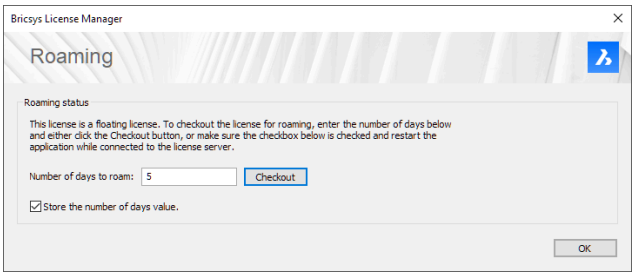
Quando uma licença de Rede é usada em um computador-cliente, o botão **Roaming...** é exibido na caixa de diálogo de informação do licenciamento (Licensing information).



1 Se seu computador se conecta ao servidor de licenças, através de um servidor proxy, clique no botão **Configurações de Proxy...** para definir as configurações de proxy para uma licença de Rede.

2 Clique o botão **Roaming...**.

Result: A caixa de diálogo **Roaming** é aberta.



3 Entre o número de dias que você deseja autorizar a licença para ficar fora da rede, e clique no botão **Check-out**.

4 (Opcional) Marque a caixinha se você quer atualizar automaticamente o período de roaming, cada vez que seu computador é conectado ao servidor.

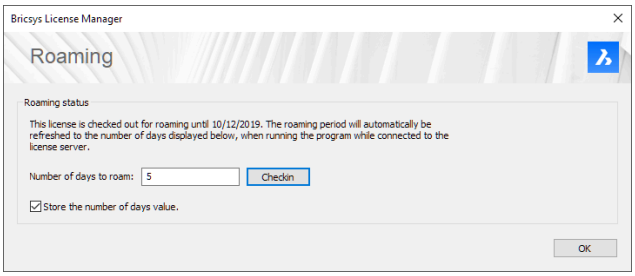
5 Reinicie **BricsCAD®** para aplicar as alterações.

2.5.1.2 Retornar uma licença de Rede em roaming para o servidor

Se você estiver conectado ao servidor de licenças, você pode retornar uma licença roamed para o servidor.

1 click the **Roaming...** button in the Licensing information dialog box.

Result: The **Roaming** dialog box opens.



2 Click the **Checkin** button.

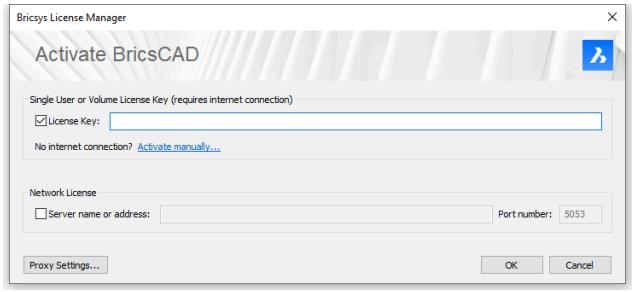
3 Reinicie **BricsCAD®** para aplicar as alterações.

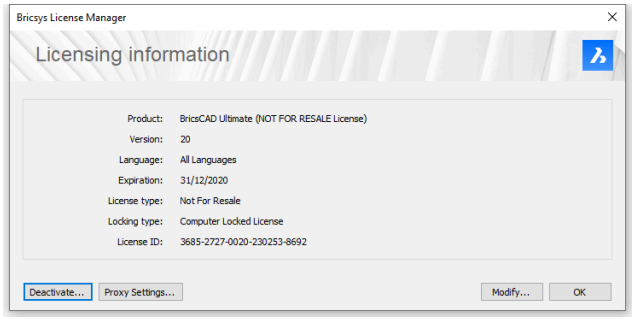
2.5.2 Configurações de proxy em um computador cliente

Antes de Iniciar: Alguns computadores estão conectados a uma rede ou à Internet através de um servidor proxy. Portanto, as configurações de proxy devem estar configuradas antes de ativar ou desativar uma licença.

2.5.2.1 Configurando as configurações de proxy

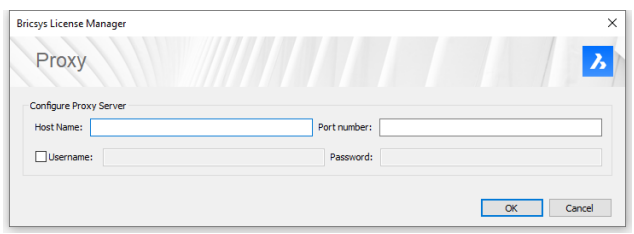
1 Clique no botão **Configurações do Proxy...** na caixa de diálogo **Ativar BricsCAD** ou **Informação de Licenciamento**.





Result: A caixa de diálogo **Proxy** é aberta.

2 Preencha os campos necessários na caixa de diálogo **Proxy**.



3 Clique o botão **OK** para continuar.

Result:

Nota: A configuração de configurações de proxy permanece ativa somente para a sessão **BricsCAD®** atual. Se você precisar reativar ou desativar uma licença em sessão futura, terá que reconfigurar as configurações de proxy.

2.5.2.2 Configurando as configurações de proxy permanentemente para uso de licença de Rede

Antes de Iniciar: Se você usar uma licença de Rede, **BricsCAD®** precisa se conectar ao servidor de licenças sempre que uma sessão for aberta. Portanto, as configurações de proxy devem estar definidas permanentemente.

Prossiga da seguinte forma no computador em que **BricsCAD®** está sendo executado:

1 Crie a variável **HTTP_PROXY**: HTTP_PROXY=[LicenseServer]:[PortNumber]

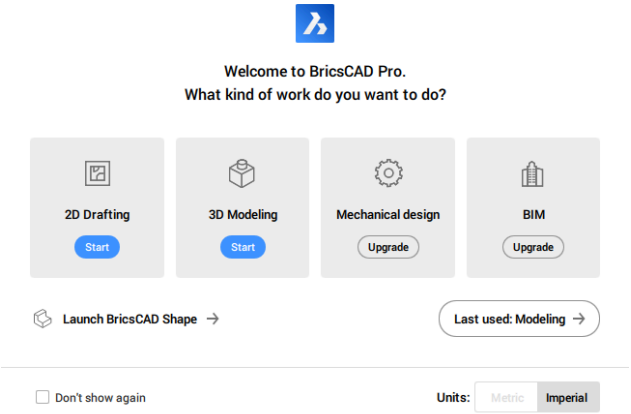
2 (Opcional) Se o servidor proxy suporta autenticação básica, crie a variável **HTTP_PROXY_CREDENTIALS**: HTTP_PROXY_CREDENTIALS=[UserName]:[Password]

3. Começar com BricsCAD®

3.1 Lançar BricsCAD®

3.1.1 Lançador BricsCAD

Se a variável de sistema GETSTARTED estiver Ligada, a caixa de diálogo Lançador BricsCAD será exibida quando você iniciar **BricsCAD®**. Aqui você pode escolher a Area de trabalho para a sessão atual. As áreas de trabalho disponíveis dependem do nível da sua licença.



- **Desenho 2D**: Inicia a área de trabalho Desenho. **BricsCAD® Lite** licença ou superior é necessária.
- **Modelagem 3D**: Inicia a área de trabalho Modelagem. **BricsCAD® Pro** licença ou superior é necessária.
- **Projetos de Mecânica**: Inicia a área de trabalho Mecânica. **BricsCAD® Mechanical** ou licença **BricsCAD® Ultimate** é necessária.
- **BIM**: Inicia a área de trabalho BIM. **BricsCAD® BIM** ou licença **BricsCAD® Ultimate** é necessária.
- **Lançar o BricsCAD Shape**: Inicia **BricsCAD® Shape**. Nenhuma licença é necessária.
- **Último uso**: Inicia a última área de trabalho que foi utilizada anteriormente.
- **Não mostrar novamente**: define a variável de sistema GETSTARTED como 0. Para redefinir a exibição da caixa de diálogo BricsCAD Launcher, digite **Getstarted** na linha de Comando e escolha **1 para Ligar**.
- **Unidades**: Define as unidades para o desenho template padrão. Define a variável de sistema BASEFILE

Nota: A ferramenta gratuita de modelagem de sólidos, **BricsCAD® Shape**, está incorporada ao instalador único **BricsCAD®**. Quando a avaliação de **BricsCAD® Ultimate** por 30 dias expira, esta reverte para uma licença perpétua **BricsCAD® Shape**, que pode ser convertida em uma licença comercial completa ao comprar uma chave de licença. Você não precisa desinstalar primeiro **BricsCAD®**.

3.2 Explorar a Interface

3.2.1 A aba Iniciar

A aba **Iniciar** está localizada na parte superior da área de desenho, antes das abas de documentos.

Na aba **Iniciar**, você pode:

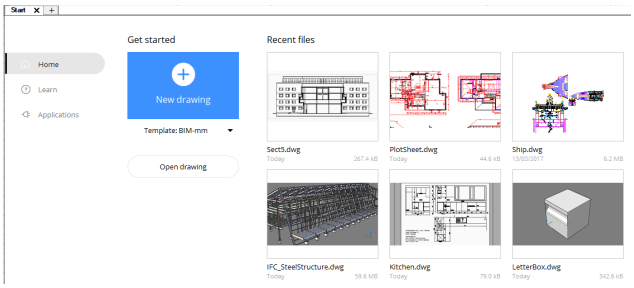
- **Abrir um desenho recente**

  Clique na imagem em miniatura de um arquivo recente.

  O número de arquivos recentes disponíveis é controlado pelo valor da variável de sistema RECENTFILES.
- **Criar um novo desenho**

  Clique no botão **Novo Desenho** para criar um novo desenho usando o template atual. Opcionalmente, abra a lista de **Templates de Desenho** para selecionar um template de desenho diferente.
- **Abrir desenho**

  Exibe a caixa de diálogo **Abrir arquivo**.



A aba **Iniciar** está sempre ativa, por padrão. Se você fechou acidentalmente a aba, poderá reabri-la com o atalho de teclado **Ctrl+Home** (as duas teclas) ou com o comando IRPARAINICIO.

Nota: Ao usar o atalho de teclado **Ctrl+Home**, certifique-se de desligar Num lock em algumas plataformas.

Nota: A variável de sistema SHOWDOCTABS precisa estar **Ligada** para exibir a aba **Iniciar**.

3.2.2 Abas de documentos

Para cada desenho aberto no momento, uma aba de documento é localizada na parte superior da área de desenho.

Abas de documentos permitem a você:

- Alterne facilmente entre todos os desenhos abertos.
- Criar um novo desenho: clique no botão **+** à direita das abas.
- Fechar um desenho: clique no **X** na aba do documento.

3.2.3 Escolher a Area de trabalho

As área de trabalho padrão são:

- **Desenho**: usa a Faixa de opções.
- **Desenho (barras)**: usa menus e barras de ferramentas.
- **Modelagem**: exige um nível de licença **BricsCAD® Pro** ou superior; utiliza a Faixa de opções.

- **Modelagem (barras de ferramentas)**: requer um nível de licença **BricsCAD® Pro** ou superior; usa menus e barras de ferramentas.
- **Mecânica**: requer um nível de licença **BricsCAD® Mechanical** ou **BricsCAD® Ultimate**.
- **BIM**: requer um nível de licença **BricsCAD® BIM**ou **BricsCAD® Ultimate**.

A área de trabalho atual configura o layout geral da janela da aplicação **BricsCAD®**.

Uma área de trabalho define o conteúdo de:

- Menus
- Barras
- Painéis
- A Faixa de opções
- O Básico do Quad

Você pode definir a área de trabalho atual em:

- O Lançador BricsCAD.
- A Area de trabalho Atual na barra de Status.
- A barra Areas de trabalho.
- O comando AreaDeTrabalho.

3.2.4 A área de desenho

Quando você inicia uma sessão de desenho, sua área de trabalho inicial é chamada de **Model Space** (Espaço do Modelo). O Model Space é uma área na qual você cria as entidades bidimensionais e tridimensionais com base no Sistema de Coordenadas Mundo = World (WCS) ou em um Sistema de Coordenadas do Usuário (UCS). **BricsCAD®** fornece uma área de trabalho adicional chamada **Paper Space** (Espaço do Papel). Cada layout do Paper Space representa uma folha de papel. Você pode criar até 255 layouts de Paper Space em cada desenho isolado.

Use a aba **Modelo** para exibir e trabalhar no Model Space. Clique em uma das abas de **Layout** na parte inferior da área de desenho para trocar para o ambiente do Paper Space.

3.2.4.1 Model space

O Model Space consiste em uma única vista que preenche a tela. Se for necessário, você pode criar vistas adicionais, chamadas viewports, que podem mostrar diferentes vistas do seu desenho ou modelo 3D. Você pode gerenciar Viewports do desenho com o comando VPORTS.

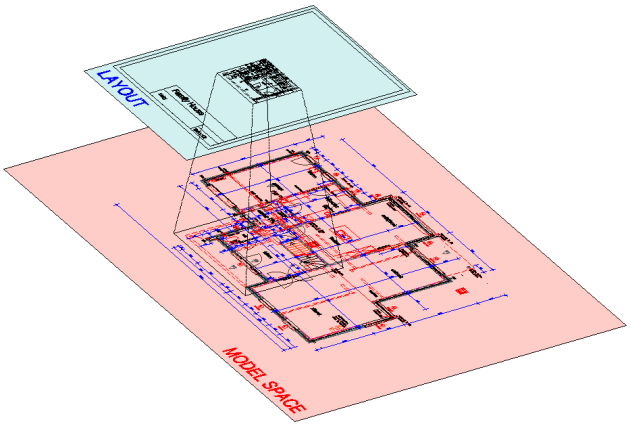
As Viewports são exibidas como mosaicos. Você pode trabalhar em apenas uma dessas viewports de cada vez, mas todas as viewports serão atualizadas simultaneamente para qualquer modificação que você fizer no seu desenho. Clique em uma Viewport, para torná-la a Viewport Atual.

3.2.4.2 Paper Space

O Paper Space representa um layout em papel do seu desenho. Nesta área de trabalho, você pode criar e organizar vistas diferentes do seu desenho, semelhante à maneira como você organizaria seus desenhos

em uma folha de papel. No Paper Space você também pode adicionar notas explicativas, anotações, bordas, blocos de título, e outras entidades relacionadas à impressão, que você não deseja ver no espaço do modelo (Model Space).

Cada desenho tem pelo menos um **Layout** no qual você pode ter uma ou mais viewports de layout. Você pode considerar uma Viewport de layout como uma janela de exibição em seu desenho no espaço do modelo:



Você pode colocar Viewports de layout em qualquer lugar em um layout de Paper Space (do papel). Cada Viewport de layout tem sua própria Escala e visibilidade de Camadas. Você pode imprimir todas as Viewports no mesmo layout simultaneamente.

Você pode gerenciar Viewports de layout com o comando CVISTA.

O Pape Space (espaço de papel) oferece múltiplas vantagens ao imprimir seu desenho:

- Você pode criar várias Viewports para imprimir várias vistas do desenho. Essas vistas podem mostrar uma parte do seu desenho, tanto numa mesma escala como em diferentes escalas.
- Você pode criar vários layouts para imprimir o mesmo desenho com diferentes configurações de impressão, como larguras de caneta, arquivos de configuração da impressora, configurações de espessura de linha, escala de desenho...
- Você pode adicionar entidades relacionadas à impressão que não são essenciais para o próprio modelo, como keynotes, anotações, blocos de título...

3.2.5 A linha de Comando

A linha de Comando é um painel encaixável opcional localizado por padrão na parte inferior da área de desenho.

A linha de comando é um método rápido e eficiente para iniciar comandos e definir variáveis do sistema e preferências do usuário.

BricsCAD® solicita ao usuário a entrada de dados através da linha de Comando, e reporta neste mesmo local o resultado de comandos, como DIST, AREA ou LISTA.

Quando a linha de Comando está fechada, a entrada pelo teclado e os avisos de comando aparecem na barra Status.

3.2.6 Menus

A barra de menus está oculta em todas as áreas de trabalho, por padrão.

O conteúdo do menu é definido na atual área de trabalho.

3.2.6.1 Controlar a exibição da barra de menus

1 Mova o cursor sobre uma barra de ferramentas ou painel da Faixa de opções.

2 Clique com o botão direito e escolha **Barra de Menus** no menu de contexto.

3.2.7 Barras

As Barras de ferramentas são usadas por padrão nas áreas de trabalho: Desenho (barras) e Modelagem (barras), mas podem ser utilizadas em qualquer área de trabalho.

- Você pode escolher ícones pequenos (16 x 16 pixels), grandes (32 x 32) ou extra grandes (64 x 64) nas barras.
- Em alguns botões de ferramentas, uma pequena seta preta no canto inferior direito do ícone indica que está disponível um submenu desdobrável. Um submenu contém um grupo de ferramentas relacionadas entre essas. Pressione e segure o botão esquerdo do mouse para expandir o menu desdobrável e escolha uma das ferramentas do submenu. A ferramenta escolhida por último permanece visível na barra de ferramentas quando estiver recolhida.
- Você pode abrir ou fechar barras de ferramentas a qualquer momento.

3.2.8 A Faixa de opções

A Faixa de opções é um conjunto de barras de ferramentas com abas, preenchidas com botões de ferramentas e outros elementos de controle gráfico, agrupadas por funcionalidade.

O conteúdo da Faixa de opções depende da atual área de trabalho.

As áreas de trabalho: Desenho, Modelagem, Mecânica e BIM usam por padrão a Faixa de opções.

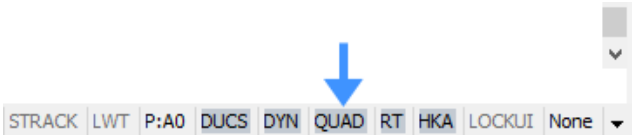
Use o comando FAIXAOPCOES e o comando FECHARFAIXAOPCOES para abrir ou fechar a Faixa de opções.

3.2.9 O Básico do Quad

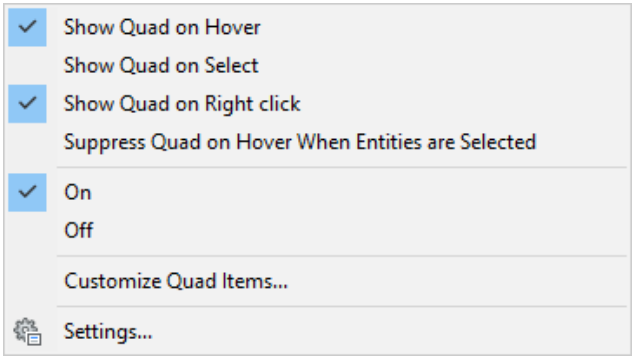
O **Quad** é um sistema de comandos completo, localizado diretamente no cursor do mouse.

Quando você move o cursor pela tela ou quando existe um conjunto de seleção, o Quad detecta o tipo de entidade e apresenta comandos que funcionam nessa entidade, ou na seleção de entidades. Quando nenhuma entidade for detectada ou selecionada, você pode usar o Quad para criar entidades, inserir blocos, anexar subjacências (underlays) ou abrir as caixas de diálogo **Configurações** e **Explorer do Desenho**.

- Para alternar a exibição do **Quad**, clique no campo Quad na barra Status, na parte inferior da janela da aplicação **BricsCAD®**.



- Clique com o botão direito do mouse no campo **Quad** para definir as configurações do Quad em um menu de contexto.

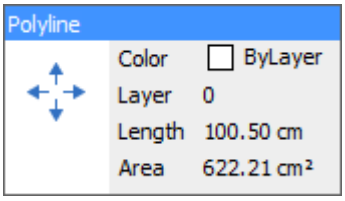


3.2.9.1 Procedimento básico para editar entidades usando o Quad

1 Mova o cursor para sobre a entidade. A entidade é realçada e o Quad exibe o comando usado mais recentemente com este tipo de entidade.

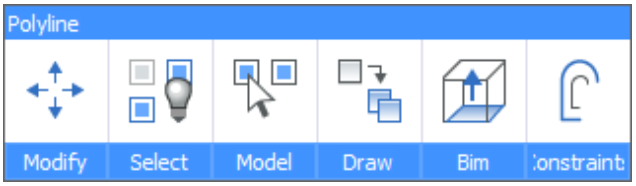


Se a variável de sistema ROLLOVERTIPS = Liga, as propriedades da entidade sob o cursor serão exibidas no Quad.

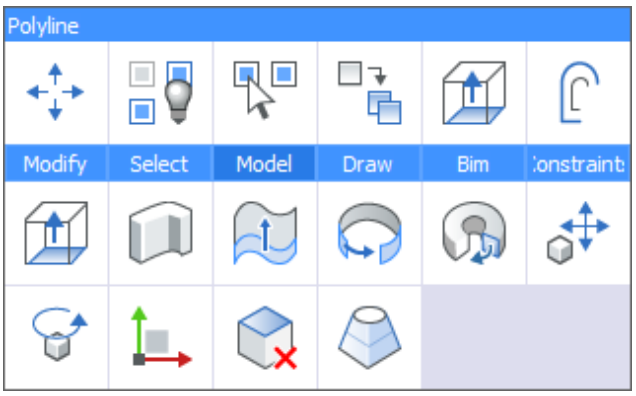


2 Mova o cursor para o Quad para exibir os comandos usados mais recentemente com este tipo de entidade.

Algumas abas de categoria de comando são exibidas na parte inferior.



3 Mova o cursor sobre as abas para expandir o grupo de comandos sob o cursor.



4 Clique em um ícone para lançar o comando.

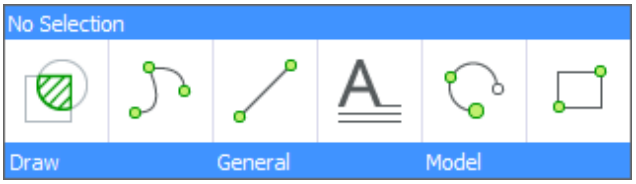
3.2.9.2 Usando o Quad Sem Seleção

Certifique-se de que a opção **Exibir o Quad no clique com o botão direito do mouse** esteja definida.

1 Siga um dos procedimentos:

- Mova o cursor para uma área vazia da janela de desenho e clique com o botão direito do mouse.
- Quando o cursor estiver sobre uma entidade, mantenha pressionado o botão direito do mouse por mais tempo.

O Quad **Sem Seleção** é exibido, mostrando os ícones dos comandos usados mais recentemente.



2 Siga um dos procedimentos:

- Clique em um dos comandos usados mais recentemente.
- Mova o cursor sobre uma aba de comando e selecione um comando quando essa for expandida.

3 Execute o comando.

3.2.10 Clicar botão-direito num Menu

Esta ação traz os menus de Contexto. Ao clicar o botão-direito do mouse é exibida uma lista de comandos ao lado do cursor que permitem a você clicar qualquer comando.

Quando o Quad estiver ativo, mantenha pressionado o botão direito do mouse um pouco mais (= clique longo com botão-direito).

O conteúdo de um menu de contexto depende da entidade atual ou do conjunto de seleção.

Quando o Quad não está ativo e nada está selecionado, um clique com o botão direito vai repetir o comando mais recente. Mantenha pressionado o botão direito do mouse um pouco mais para exibir o menu de contexto.

3.2.11 A barra de Status

A barra de Status está localizada na parte inferior da área de desenho.

- Esta mostra dicas de ferramentas quando você passa o mouse sobre um botão da ferramenta na barra de status.
- Permite entrar nomes de Comando e avisa por opções de comando quando a janela de comando está fechada.
- Esta exibe o estado atual das configurações da entidade: camada, cor, estilo de linha,... e permite editar essas configurações.

  Nota: Essas estão desativados em qualquer área de trabalho, por padrão.
- Exibe o estilo atual de dimensão e estilo de texto, e permite ajustá-los.
- Esta exibe o estado atual dos auxiliares de desenho, como snap de entidade, e permite ligar e desligar esses.
- ...

Clique na pequena seta preta para baixo no lado direito da barra de status para gerenciar a exibição de todos os campos.

3.2.12 Painéis

Painéis são caixas de diálogo encaixáveis.

Os painéis disponíveis são:

- Editor de Animação
- Anexos
- BIM BCF
- BIM Composições
- BIM Perfis
- BIM Navegador do Projeto
- Gerenciador de BOM
- Bricsys 24/7
- Linha de Comando
- Navegador de Conteúdo
- Detalhes
- Personalizações do Desenho
- Camadas
- Biblioteca
- Navegador de Mecânica

- Parâmetros e Restrições
- Propriedades
- Materiais de Renderização
- Relatório
- Faixa de Opções
- Conjs. de Folhas
- Estrutura
- Dicas
- Paletas de ferramenta

3.2.13 O modo de tela limpa

No modo de Tela Limpa, você pode expandir a área de desenho descartando elementos da interface do usuário, como a barra de menus, faixa de opções, barras de ferramentas, painéis, linha de comando, barra de status e as abas do documento.

Você ainda pode usar o Quad para lançar comandos.

Pressione e segure a tecla Ctrl e pressione a tecla 0 (zero) - na faixa superior do teclado, para ativar/desativar o modo de tela limpa.

Nota: Não use o teclado numérico.

Nota: Nos teclados-AZERTY: não pressione a tecla Shift ao pressionar a tecla 0.

3.2.14 Look From (assistente)

O assistente Look From permite controlar a exibição do seu desenho. Você pode selecionar várias vistas predefinidas, como vistas ortográficas e isométricas.



Por padrão, o assistente **Look From** é exibido no canto superior direito da tela gráfica.

O comportamento do assistente **Look From** pode ser controlado por várias variáveis do sistema e preferências do usuário.

3.3 Trabalhar em BricsCAD®

3.3.1 Criar um desenho

Existem várias maneiras de criar novos desenhos no BricsCAD:

- Na aba **Iniciar**.
- Usando o comando NOVOR.
- Usando o comando NOVO.
- Usando o comando NEWWIZ.
- Com um duplo-clique num arquivo *.dwt em uma janela do Windows Explorer.

Nota: Um novo desenho é sempre criado como uma cópia de um desenho de template. O template padrão é salvo por meio da variável de sistema BASEFILE.

3.3.2 Desenhos Template

Um desenho de template (modelo) é um arquivo de desenho com a extensão *.DWT em seu nome.

Este é usado para criar um novo desenho, que parte de uma cópia exata do desenho template. Qualquer desenho com extensão *.DWT é considerado um desenho de template. Os desenhos template do **BricsCAD**® estão localizados na pasta Templates. Use o comando TEMPLATEFOLDER para abrir a pasta. Se você salvar seus desenhos template personalizados na pasta Templates, eles estarão disponíveis na aba Iniciar para criar novos desenhos. Usar templates devidamente preparados é essencial para um uso produtivo do **BricsCAD**®.

Um desenho de template não contém entidades no Model Space, mas contém configurações de acordo com os requisitos específicos de desenho e projeto:

- Unidades - imperial ou métrica e exibir unidades.
- Camadas.
- Estilos de dimensão e de chamadas.
- Estilos de texto.
- Estilos de tabela.
- Múltiplos layouts, cada um com bloco de título, tamanho de papel, impressora e estilo de plotagem.
- Blocos padrão para notas, tabelas, símbolos e outras entidades.

3.3.3 Configurar criação de Entidades.

Novas entidades são criadas na camada atual, usando as configurações atuais para criação de entidades.

- Cor
- TipoLinha
- Escala do Tipo de linha
- EspLinha
- Transparência

Você pode definir configurações de criação de entidade na seção **Geral** do painel Propriedades.

Você pode definir todas as configurações de criação de entidade **PorCamada**. Isso significa que é usada a configuração correspondente para a Camada atual.

Você pode definir as propriedades da camada na seção Camada na caixa de diálogo Explorer do Desenho.

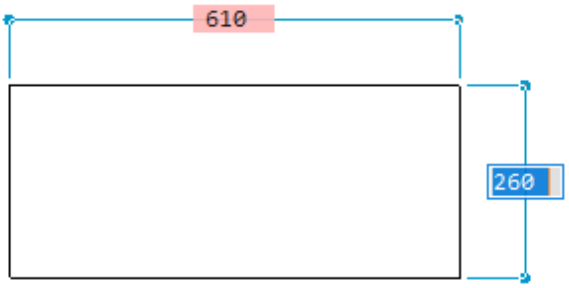
3.3.4 Como desenhar com precisão

Novas entidades são criadas no plano-XY do atual sistema de coordenadas. Por padrão, este é o Sistema Mundo de Coordenadas - World (WCS). Todas as dimensões em um arquivo DWG são expressas nas unidades do desenho (DU). A variável de sistema INSUNITS (Unidades de Inserção) define o valor de 1 DU.

Para novos desenhos, a variável do sistema INSUNITS é definida no desenho de template que você selecionou para criar o desenho. **BricsCAD**® fornece uma série de desenhos de template para unidades decimais e imperiais.

Para desenhar com precisão, **BricsCAD**® fornece vários auxiliares de desenho:

- Snaps de entidade 2D: você pode usar pontos-chave em entidades 2D existentes, como extremidade, ponto médio ou centro, e interseção. Você também pode desenhar perpendicularmente, em paralelo ou tangencialmente a uma entidade.
- Snaps de entidade 3D: você pode usar pontos-chave em entidades 3D, como sólidos.
- Dimensões dinâmicas. Quando as dimensões dinâmicas estão **Ligadas**, você pode inserir distâncias e ângulos em campos de dimensão dinâmica. Quando vários campos de dimensão dinâmica forem exibidos, por exemplo, ao desenhar um retângulo, pressione a tecla Tab para alternar para o próximo campo. Quando você entra um valor e pressiona a tecla Tab, o campo fica vermelho, o que indica que o valor está fixado.



- Rastreamento Polar força a direção do desenho a ângulos especificados em relação aos eixos (X,Y,Z) do atual sistema de coordenadas.
- Rastreamento de Snap a Entidade combina snaps a entidade e rastreamento polar
- Entrada de Coordenadas permite entrar coordenadas XYZ para especificar pontos.
- Snap e grade. Uma grade é um conjunto de linhas ou pontos igualmente espaçados, que servem como referência visual de distância. O cursor se encaixa nos pontos da grade, o que permite desenhar rapidamente em intervalos fixos.
- Limites do Desenho permite limitar a área para desenho, o que impede a criação de entidades fora dos limites de desenho.
- Modo Ortho força o cursor a se mover paralelamente tanto ao eixo-X como Y do sistema de coordenadas atual.
- Sistema de coordenadas do usuário. Você pode definir sistemas de coordenadas de usuário nomeados (UCS) localizados em qualquer lugar no espaço tridimensional. Você pode restaurar a qualquer momento um UCS nomeado.
- Entrada direta de Distância permite entrar comprimentos, raios, diâmetros, e vetores de movimento.

- Distância Mais Próxima permite visualizar e modificar dinamicamente a distância entre duas entidades ou subentidades selecionadas.
- O Manipulator é uma ferramenta versátil para rapidamente mover, rotacionar, escalar e espelhar entidades selecionadas.

3.3.5 Executar comandos

Em **BricsCAD**® você pode iniciar comandos das seguintes maneiras:

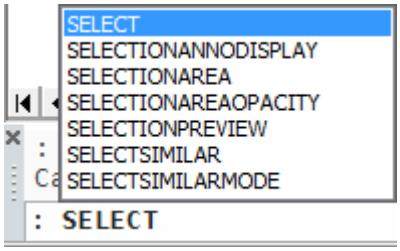
- Digite o comando na linha de Comando.
- Escolha o comando em um Menu.
- Clique no botão do Comando, em uma barra de ferramentas.
- Clique no ícone do Comando, na Faixa de opções.
- Escolha o comando no Quad.

3.3.5.1 Digitar um comando na linha de Comando.

Se você souber o nome do Comando, pode inseri-lo na linha de Comando. Quando a linha de Comando está fechada, a entrada pelo teclado e os prompts de comando aparecem na barra de Status.

Se o sinalizador **Ativar** da variável de sistema AUTOCOMPLETEMODE estiver definido, os nomes de comandos são automaticamente completados enquanto você digita.

Todos os comandos e/ou variáveis do sistema e ainda as preferências do usuário, que contêm as atuais cadeias de caracteres são exibidos na lista de Sugestões:



Siga um dos procedimentos:

- Pressione Enter se estiver exibido o comando atualmente destacado.
- Clique em um comando que apareça na lista de sugestões.

Você também pode iniciar alguns comandos com o 'alias' de comando. Por exemplo, você pode iniciar o comando LINHA inserindo L no aviso de Comando.

3.3.5.2 Escolhendo as opções de comando

Muitos comandos em **BricsCAD**® fornecem várias opções. Essas opções aparecem na linha de Comando ou na barra de Status (quando a linha de comando está fechada) e também em um menu Prompt.

Siga um dos procedimentos:

- Pressione Enter para aceitar a opção padrão. A opção padrão está localizada no final do aviso de comando, entre colchetes de seta (<>).

- Clique sobre a opção na linha de Comando.
- Clique a opção no menu de Prompt
- Digite a letra maiúscula da opção, pressione Enter ou clique o botão-direito do mouse.

3.3.6 Controle da Vista do desenho

Em **BricsCAD**® você pode controlar a exibição do seu desenho de várias formas:

- Usando o mouse e o teclado
- Usando os comandos de Vista
- Usando o assistente Look From

As manipulações de vista mais usadas são:

Ação	Mouse	Comando	Look From
Vista Extensão	Clicar-duplo na rodinha de rolagem	**Zoom** > **Extensao**	Clicando em uma zona de disparo faz rotacionar a vista, e enquadra o desenho na sua extensão.
Zoom	Rodinha de rolagem	**Zoom** > **In Zoom** > **Out**	
Pan	Mantenha pressionada a rodinha de rolagem ***Nota:*** Quando a variável de sistema MBUTTONPAN = Ligada	Pan	
Restaurar Vista em Planta (Vista Sup)		**Planta** > **UCS atual**	Clique na zona central de disparo

3.3.7 Selecionar entidades

Comandos de manipulação, como MOVER, COPIAR e ESPELHAR, solicitam que você selecione uma ou mais entidades. Se a variável PICKFIRST estiver Ligada, você também poderá compor o conjunto de seleção antes de iniciar o comando.

BricsCAD® oferece vários métodos de seleção. Você pode combinar métodos de seleção para compor um conjunto de seleção.

3.3.7.1 Selecionar entidades uma por uma

1 Mova o cursor para sobre a entidade.

Result: A entidade se destaca.

O Quad aparece e exibe o comando usado mais recentemente, e uma seleção das propriedades da entidade.

2 Siga um dos procedimentos:
   - Clique para selecionar a entidade.
   - Pressione repetidamente a tecla Tab para realçar entidades obscurecidas, e clique quando a entidade desejada se destaca.

3 Repita as etapas anteriores para selecionar mais entidades ou lançar um comando.

3.3.7.2 Selecionar múltiplas entidades.

BricsCAD® usa métodos de seleção por janela, para selecionar múltiplas entidades:

1 Clique para definir o canto do retângulo de seleção.

2 Siga um dos procedimentos:
   - Mova o cursor para a direita. Um retângulo em azul é exibido. O ícone **contidas dentro do retângulo** indica que todas as entidades que ficam completamente dentro do retângulo de seleção serão selecionadas.
   - Mova o cursor para a esquerda. Um retângulo em verde é exibido. O ícone **contidas ou cruzadas pelo retângulo** indica que todas as entidades que se encontram dentro do retângulo, ou se sobrepõem a este, serão selecionadas.

3 Clique para definir o canto oposto do retângulo de seleção.

3.3.7.3 Removendo entidades de um conjunto de seleção

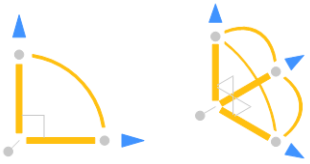
Mantenha pressionada a tecla Shift ao executar um método de seleção, para remover entidades do conjunto de seleção atual.

Pressione a tecla Esc para limpar o conjunto de seleção atual.

3.3.8 Usar o Manipulator

O Manipulator é uma ferramenta versátil para rapidamente mover, rotacionar, escalar e espelhar entidades selecionadas.

Se você mantiver pressionada a tecla Ctrl durante essas ações, as entidades selecionadas serão também copiadas.

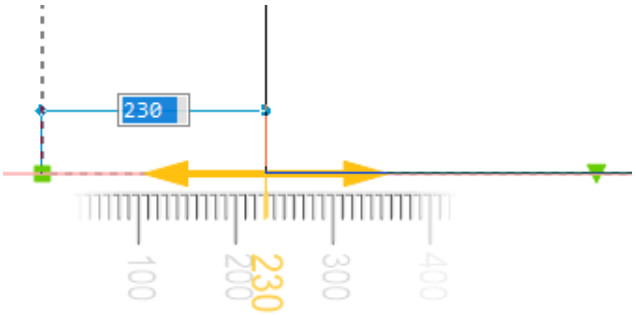


Por padrão, o Manipulator é exibido ao selecionar uma entidade por um longo clique (= manter o botão esquerdo do mouse pressionado um pouco mais tempo).

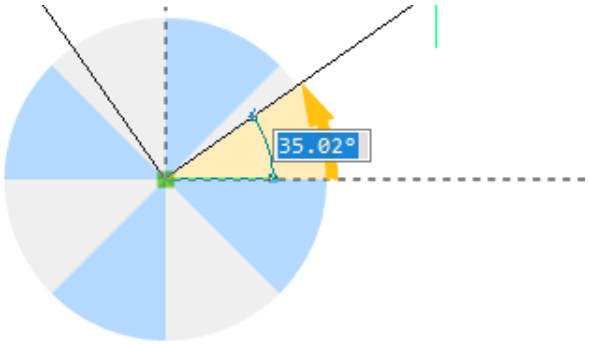
O ponto de ancoragem do Manipulator fica no ponto de aderência mais próximo, em relação ao ponto de seleção.

Coloque o cursor sobre uma das zonas de acionador, para iniciar uma ação:

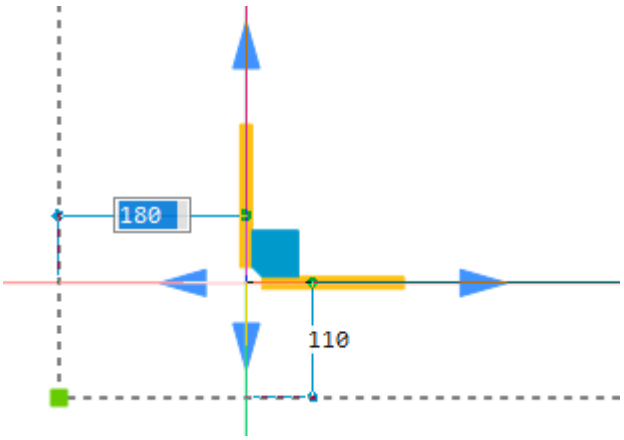
- Eixos, para mover ao longo do eixo:



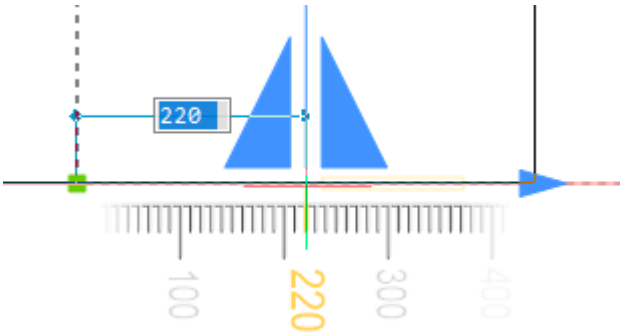
- Arcos, para rotacionar:



- Amostras de planos (= quadrados no ponto de ancoragem), para se mover livremente no plano selecionado (XY, XZ ou YZ):



- Cabeças de seta, para espelhar:



Nota: O tamanho do passo da régua depende do fator de Zoom. Ampliando o Zoom diminui o tamanho do passo, e reduzindo o Zoom aumenta o passo da régua.

3.3.9 Trabalhar com o painel Estrutura

O painel Estrutura exibe uma vista estruturada em árvore do conteúdo do desenho.

BricsCAD® fornece 3 definições de estrutura em árvore (CST):

- **Default** exibe uma estrutura com base em camadas.
- **BIM** exibe uma estrutura com base nas entidades BIM.
- **Mechanical** exibe uma estrutura com base nas entidades.

Clique no menu do hambúrguer (☰) para selecionar uma definição.

No painel Estrutura, você pode:

- Clicar no ícone expandir (**+**) para expandir um nó.
- Clicar no ícone de recolher (**-**) para recolher um nó.
- Selecione um nó, clique com o botão direito do mouse e escolha **Expandir Tudo** para expandir todos os subnós do nó selecionado.

  Selecione o nome do desenho e use esta opção para expandir todos os nós na árvore inteira.
- Selecione um nó, clique o botão direito do mouse e escolha **Recolher Tudo** para recolher todos os sub-nós do nó selecionado.

  Selecione o nome do desenho, então use esta opção para recolher a árvore inteira.
- Clique em um nó ou entidade no painel Estrutura para selecioná-la no desenho.

  Nota: Isso depende das configurações na **árvore da Estrutura**. Para configurar a árvore da Estrutura, clique o botão-direito do mouse em **painel Estrutura** > **Configurar** > **Opções**.

  Mantenha pressionada a tecla Ctrl para compor um conjunto de seleção.
- Clique o botão-direito para exibir um menu de contexto, que permite:
  - **Mostrar**: Exibe a seleção, que havia sido ocultada anteriormente.
  - **Ocultar**: Oculta a seleção.
  - **Isolar**: Isola a seleção, oculta todo o restante.
  - **Zoom**: Faz um Zoom na seleção.

Nota: Selecione o nome do desenho. A seguir, clique o botão-direito e escolha **Mostrar** no menu de contexto, para exibir todo o desenho, ou escolha **Mostrar Entidades** na aba **Geral** do Quad na condição Sem Seleção.

4. Interface do usuário

4.1 The BricsCAD application window

The components of the BricsCAD application window are:

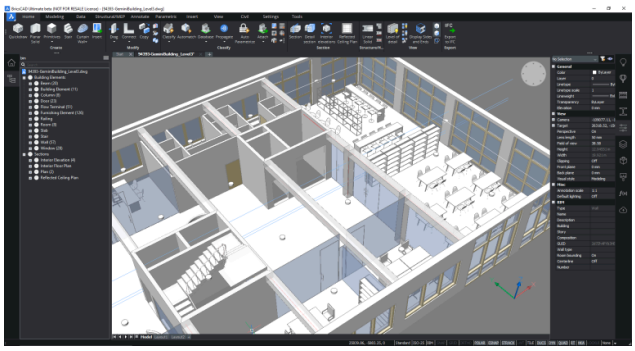
- Menu bar
- Toolbars
- Ribbon Document tabs
- Model space viewports
- Paper space viewports
- Model and Layout tabs
- Dockable panels
- Command line
- Status Bar

The layout of the BricsCAD application window is controlled by the current Workspace.

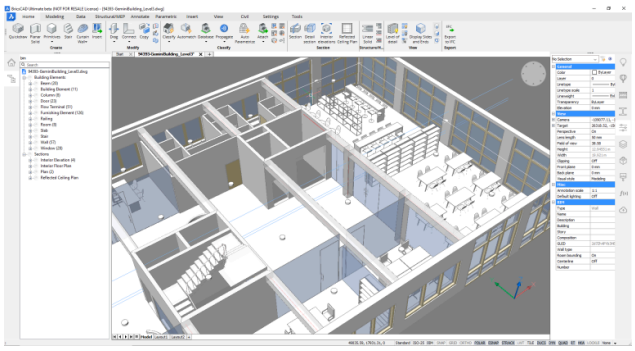
4.1.1 The color theme

The COLORTHEME system variable sets the color theme for the ribbon, panels, dialog boxes, etc. , You can choose between a dark or a light color theme.

Dark color theme (COLORTHEME = 0) :



Light color theme (COLORTHEME = 1):



To set the color theme, do one of the following:

- Type **colortheme** in the command line and choose1for light or0for dark.
- Right-click on a toolbar or ribbon panel and select**Dark Interface**in the context menu.

4.1.2 Menu bar

The menu bar sits at the top of the BricsCAD application window.

Right click a toolbar, ribbon or panel, then choose Menu Bar in the context menu to toggle the display of the menu bar on/off.

The status of the menu bar is saved through the MENUBAR system variable.

4.1.3 The ribbon

To display or hide the ribbon, do one of the following:

- Execute theRIBBONcommand or the RIBBONHIDE command.
- Right click a toolbar, a panel or an empty space in the Layout or Drawing tab area, then choose **Panels** and click **Ribbon**in the context menu.

The size of the tool buttons in the Ribbon is controlled through the RIBBONTOOLSIZE system variable. You can choose between small, large or extra-large buttons.

4.1.4 Document tabs

Document tabs sit at the top of the graphical screen by default. They allow you to easily switch between open drawings and to access the Start tab.

The display of the document tabs is controlled through the SHOWDOCTABS system variable.

4.1.5 Model and layout tabs

At the bottom of the graphic screen sit the Model and one or more Layout tabs which allow you to easily switch between model space and paper space.

4.1.6 Scroll bars

Scrollbars are off by default.

To toggle scroll bars on/off:

- Hold down the Shift key and press the**F4**function key.
- Choose**Scroll Bars**in the**View**menu.
- Launch the SCROLLBAR command.

4.1.7 The Command line

Execute the COMMANDLINE command to display the Command line.

Double click the left hand side of the status bar to toggle the display of the Command line.

4.1.8 The status bar

The Status bar sits at the bottom of the BricsCAD application window.

Right click a toolbar, ribbon or panel, then choose **Status Bar** in the context menu to toggle the display of the status bar on/off.

4.1.9 Hiding a selection of user interface elements

TheCLEANSCREENONcommand allows you to hide a selection of user interface elements simultaneously.

The CLEANSCREENOPTIONS system variable defines which user interface elements are hidden.

The **Ctrl+0** keyboard shortcut toggles the cleanscreen state.

On AZERTY-keyboards:

- Do not use the numeric keypad.
- Do not press the Shift-key when hitting the number key.

4.2 The Command line

To display the Command line, do one of the following:

- Launch theCOMMANDLINEcommand.
- Use the Shift + F2 keyboard shortcut.
- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose**Panels** > **Command line** in the context menu.
- Double-click the far left side of the**Status** bar.

4.2.1 Setting the Command line state

The Command line can either be docked or floating.

By default, the Command line is docked at the bottom of the BricsCAD application window:



To modify the height of the Command line:

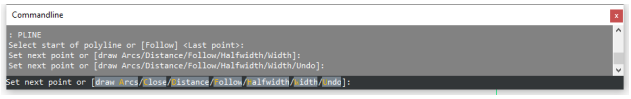
1 Place the cursor over the top edge of the Command line.

2 Press and hold the left mouse button when a double arrow appears.

3 Move the mouse.

Click the**Close**(x) button to close the Command line.

Move the cursor over the dotted left hand side, then press and hold the left mouse button to make the Command line floating:



When the Command line is floating, set the value of theCMDLINEUSENEWFRAME (command line use new floating frame) system variable = 1 to minimize the Command line to a single line. When executing a command, a number of prompt lines are displayed temporarily and transparently on the graphic screen:



The number of prompt lines is set by the CLIPROMPTLINES system variable.

Click the up arrow at the right hand side to display the **Prompt History** window.

When you move the Command line mini-frame near the bottom of the graphic screen, it will stick there.

Nota:

- You can browse the command history with the up arrow and down arrow keys of the keyboard. Press Enter to restart a command.
- When the Command line is closed, command options and keyboard entries are displayed in the **Status** bar.
- Press the F2 function key to toggle the**Command History**window.

4.2.2 Launching a command in the Command line

If the **Enable** flag of theAUTOCOMPLETEMODEsystem 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 are displayed in the**Suggestion**list.

Do one of the following:

- Press Enter to choose the highlighted command.
- Use the down or up arrow key to choose a command, then press Enter.
- Click a command in the list.
- Complete the command name, then press Enter.

4.2.3 Setting the Command line properties

1 Open the**Settings**dialog box.

2 Go to**Program Options** > **User Preferences** > **Command Line**.

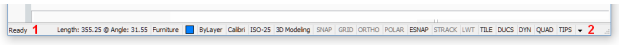
4.3 The Status bar

To display the **Status** bar, do one of the following:

- Move the cursor over a docked toolbar, the ribbon or a panel icon, then right-click and choose **Status Bar**in the context menu.
- Press the **Shift + F3** keyboard shortcut.

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

4.3.1 The Status bar fields



4.3.2 1. Status field

- Displays the status of the software.

- When the cursor is in a menu or on a toolbar, it gives a brief description of the tool or menu item under the cursor.
- When the command window is closed, it displays the command prompts, tool options and keyboard entry.
- Double-click the status field to open or close the **Command line**.

4.3.3 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:

4.3.4 Cursor coordinate values

Displays the coordinates of the current cursor position (depending on the value of the COORDS system variable).

- Click the field to cycle the **Coordinate** setting.

  The sequence is: **Geographic** - **Relative** - **Absolute** - **Off**.
- Right-click and choose an option in the context menu.

4.3.5 Current Layer

Displays the name of the current layer.

- Click to open the **Layer Explorer**.
- Right-click to select the current layer in the context menu.
- Right-click and choose **Properties** in the context menu to open the **Drawing Explorer – Layers** dialog box.

4.3.6 Current Color

Displays the current color.

- Click to open the **Select Color** dialog box.
- Right-click to select one of the basic colors in the context menu.
- Right-click and choose **Select Color...** to select a color in the **Select Color** dialog box.

4.3.7 Current Linetype

Displays the name of the current linetype.

- Click to open the **Drawing Explorer – Linetypes** dialog box.
- Right-click to select the current linetype in the context menu.

4.3.8 Current Text Style

Displays the name of the current text style.

- Click to open the **Drawing Explorer - Text Styles** dialog box.
- Right-click to select the current text style in the context menu.

4.3.9 Current Dimension Style

Displays the name of the current dimension style.

- Click to open the **Drawing Explorer - Dimension Styles** dialog box.
- Right-click to select the current dimension style in the context menu.

4.3.10 Current Work Space

Displays the name of the current workspace.

- Click to set the value of the WSCURRENT system variable in the command line.
- Right-click and choose the current workspace in the context menu.
- Choose **Customize** in the context menu to open the **Customize - Workspaces** dialog box.

4.3.11 Snap

Indicates the status of the SNAPMODE system variable.

- Click the field to toggle SNAPMODE **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Snap and Grid** settings.

4.3.12 Grid

Indicates the status of the GRIDMODE system variable.

- Click to toggle the display of grid in the current viewport.
- Right-click and choose **Settings** in the context menu to edit the **Snap and Grid** settings.

4.3.13 Ortho

Indicates the status of the ORTHOMODE system variable. Turning on **Ortho** automatically disables **Polar**.

- Click to toggle ORTHOMODE **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the ORTHOMODE system variable.

4.3.14 Polar

Toggles the **Polar Tracking** option of the AUTOSNAP system variable.

- Click to toggle Polar Tracking **On**/**Off**. Turning on **Polar** automatically disables **Ortho**.
- Right-click and choose **Settings** in the context menu to edit the **Polar Tracking** settings.

4.3.15 Esnap

Indicates the status of the **Turn of all snaps** option of the OSMODE system variable.

- Click to toggle Entity Snaps **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Entity Snaps** settings.

4.3.16 Strack

Indicates the status of the **Entity Snap Tracking** option of the AUTOSNAP system variable.

- Click to toggle Entity Snap Tracking **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Snap Tracking** settings.

4.3.17 Lineweight (LWT)

Indicates the status of the LWDISPLAY system variable.

- Click to toggle the display of line weights **On**/**Off**.
- Right-click and choose **Settings** to edit the **Lineweights** settings.

4.3.18 Paper/Model

Allows you 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 choose a paper space layout.
- 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.
- Either in Model Space or Paper Space, right-click and choose:
  - **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.

4.3.19 Annotation Scale

Displays the value of the CANNOSCALE (= current annotation scale) system variable. Right-click to display a context menu:

- Select an annotation scale in the list.
- **Custom**: launches the SCALELISTEDIT command.
- **Hide Xref scales**: lists annotation scales of the current drawing only.

4.3.20 Autoscale

Indicates the status of the ANNOAUTOSCALE system variable. When **On**, annotative entities are synchronized with the current annotation scale. Click the field to toggle ANNOAUTOSCALE **On**/**Off**.

4.3.21 Tablet

Initializes the use of a drawing tablet.

4.3.22 Viewport Lock (VPLOCK)

Displays the status of **Display Locked** property of the current viewport or the selected viewport(s). Click the VPLOCK field to toggle the property **On**/**Off**.

The VPLOCK field only appears when one or more paper space viewports are selected or when a viewport is active (Model space with floating viewports).

4.3.23 Dynamic UCS (DUCS)

Indicates the status of the UCSDETECT system variable.

- Click to toggle the UCSDETECT system variable **On**/**Off**.
- Right-click to select the supported entity types in the context menu:
  - **3D solid faces**: 3D solid faces and regions.
  - **Other entities**: 2D linear entities

4.3.24 Dynamic Input (DYN)

Indicates the status of the DYNMODE system variable.

- Click to toggle the display of Dynamic Dimensions **On**/**Off**.
- Right-click and choose **Settings** in the context menu to edit the **Dynamic Dimensions** settings.

4.3.25 Quad

Indicates the status of the QUADDISPLAY system variable.

- Click to toggle the display of the Quad **On**/**Off**.
- Right-click and choose an option in the context menu.
  - **Show Quad on Hover**.
  - **Show Quad on Select**.
  - **Show Quad on Right Click**.
  - **Suppress Quad on Hover When Entities are Selected**.
  - **On**/**Off**
  - **Customize Quad Items…** : Displays the **Customize / Quad** dialog box.
  - **Settings…** : Displays the **Settings** dialog box.

4.3.26 Rollover tips (RT)

Indicates the value of the ROLLOVERTIPS system variable.

- Click to toggle the display of rollover tips.
- Right-click and choose **Customize…** in the context menu to open the **Customize / Properties** dialog box. Here you can define which properties are displayed in the **Rollover Tips** for each entity type.

4.3.27 Hotkey Assistant (HKA)

Indicates the value of the HOTKEYASSISTANT system variable. When switched **On**, an assistant widget at the bottom of the screen reveals options available by pressing the Ctrl key.

- Click to toggle the display of the HKA widget **On**/**Off**.
- Right-click to display a context menu:
  - **On** or **Off** to set the display of the HKA widget.
  - **Configure**: Displays the **Hotkey Assistant Configuration** dialog. Select the commands for

which you want the HKA widget to display when HOTKEYASSISTANT = ON.

4.3.28 Lock User Interface (LOCKUI)

Allows you to lock the location and size of toolbars and dockable panels.

- Click to toggle the lock **On**/**Off**.
- Right-click to display a context menu.
  - Select the UI items to lock: toolbars, panels or both.
  - Choose **Settings...** to set the value of the LOCKUI system variable in the **Settings** dialog box.

4.3.29 GIS Coordinate System

Displays the name of the current GIS coordinate system.

4.3.30 Tray Icon Items

(not implemented) Display icons instead of text for the **Status** bar fields.

4.4 Document tabs

4.5 Toolbars

The availability of a command depends on the license level (**Shape** - **Lite** - **Pro** - **Mechanical** - **BIM** - **Ultimate**) as indicated in the command articles in the Command Reference.

4.5.1 To set the icon size

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

The icon size is saved in the TOOLBUTTONSIZE system variable in the current Workspace.

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

4.5.2 Toolbar Flyouts

On some tool buttons, a small black arrow at the bottom right corner of the icon indicates that 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.

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.

4.5.3 Opening a toolbar

1 In **Windows**: Hover over a toolbar or a blank portion of the toolbar area, then right-click.

In **Linux**/**Mac**: Hover over a blank portion of the menu bar, status bar or toolbar area, then right-click.

A context menu appears.

2 In the context menu, choose **Toolbars**, then choose 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 is displayed.

Toolbars that are already open, are ticked.

If the list does not fit on the screen, go to **More** at the bottom of the context menu.

Some related toolbars are grouped in a flyout menu, e.g. 3D.

3 Click a toolbar on the context menu.

4 (option) To dock a toolbar, drag the toolbar to the top, bottom, left or right edge of the BricsCAD application window.

Nota:

- 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 dotted grip at its left edge (when docked horizontally) or top edge (when docked vertically) to undock.
- To dock a toolbar vertically in **Linux**, the **Position** property of the toolbar must be set to **Left** or **Right**.
- The LOCKUI system variable allows you to lock the location and size of toolbars and dockable panels.
- Click the **LOCKUI** field in the **Status** bar to toggle the locking of the user interface.

4.6 The Quad cursor menu

The Quad cursor menu, aka the Quad is a context sensitive graphic menu which allows you to easily edit, select, show or hide and create entities. The content of the Quad is defined in the workspace and is fully customizable.

4.6.1 Activating the Quad

Activation of the Quad is controlled by theQUADDISPLAYsystem variable.

To toggle the Quad On/Off, do one of the following:

- Click theQUADfield in the**Status** bar.
- Press the**F12**function key.

4.6.2 Setting the Quad display options

Right-click the **Quad** field in the **Status** bar, then choose an option in the context menu.

- **Show Quad on Hover**.
- **Show Quad on Select**.
- **Show Quad on Right Click**: displays the 'No Selection' Quad which allows you to create entities.
- **Suppress Quad on Hover When Entities are Selected**.

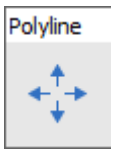
4.6.3 Editing the Quad settings

Right-click the**QUAD**field in the**Status** bar, then choose**Settings**in the context menu.

The**Settings**dialog box opens, showing the**Quad settings** screen.

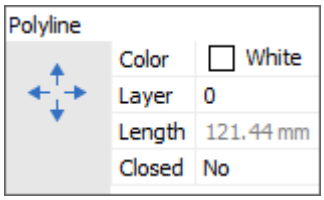
4.6.4 Using the Quad

When the cursor hovers over an entity, the entity is highlighted and the Quad displays the most recently used command with this entity type.



If multiple entities are under the cursor, repeatedly press the Tab key to highlight all entities one by one.

IfRollover Tips(RT) = On, the properties of the entity under the cursor are displayed next to the default command icon in the Quad.

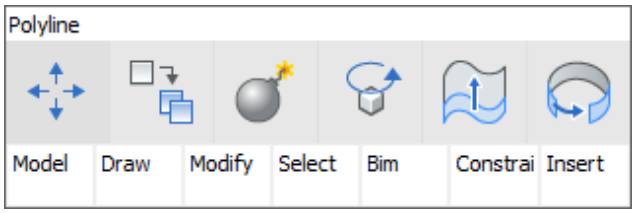


Launching the default command

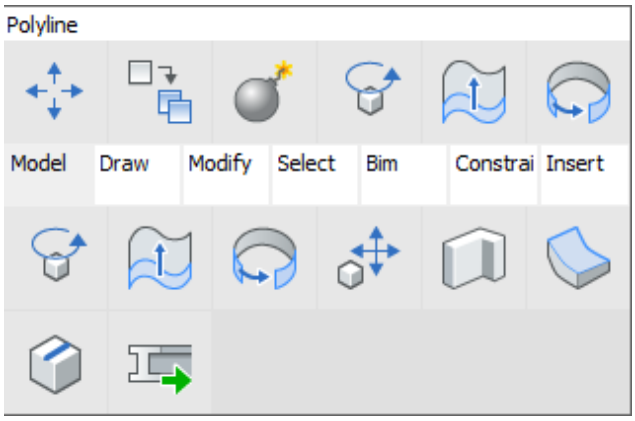
- If QUADCOMMANDLAUNCH = 1, right-click without entering the Quad to launch the command.
- If QUADCOMMANDLAUNCH = 0, click the command button.

Launching a different command

Move the cursor to the default command to expand the Quad: the 6 most recently used commands that apply to the selected entity are displayed. The number of icons that appear is set by the QUADWIDTH system variable (default = 6).



Other commands that apply to the selected entity sit in a number of tabs. Move the cursor over a tab to expand its content.



Click an icon to launch the command.

Nota: Click the title bar of the Quad to go back and forth between the Quad commands and the Rollover Tips.

4.6.5 Selecting multiple entities

1 Move the cursor over the first entity, then click when the entity is highlighted.

1 The entity is selected and remains highlighted.

2 Move the cursor to another entity, then click when the entity highlights.

3 The entity is added to the selection set and remains highlighted.

4 (option) Repeat step 2 to select more entities.

5 (option) Move the cursor to a selected entity, then press and hold the Shift key and click.

6 The entity is removed from the selection set.

Nota:

- When the Quad is active, the value of theSELECTIONPREVIEW system variableis neglected unless a command prompts for entity selection.
- When working with solids, theSELECTIONMODESsystem variable controls which subentities are detected: edges, faces and/or boundaries.
- Holding down the Ctrl key inverts the current selection mode.
- 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.

4.6.6 Using the Quad to create entities

If flag 4 (= Display the Quad on right-click) of theQUADDISPLAYsystem variable is set, entity creation commands are available in the Quad (No Selection Quad). Right-click the**Quad**field in the**Status** barand check the option in the context menu.

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

The available entity creation commands in the Quad are:

- The most recently used entity creation commands.
- The number of available commands depends on theQUADWIDTH system variable.
- A number of command group tabs, depending on the currentworkspace.

4.7 Rollover tips

The Rollover Tips feature displays a selection of entity properties in a popup box near the cursor when hovering over an entity or when an entity is selected. When multiple entities are selected, only their shared properties are displayed.

Which properties are available can be defined per entity type in the **Customize / Properties** dialog box.

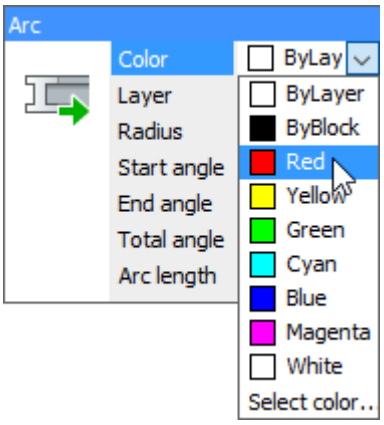
The availability of Rollover Tips is controlled through the ROLLOVERTIPS system variable.

Click the **RT** field in the **Status** bar to toggle Rollover Tips On/Off.

Rollover tips are displayed when:

- The Quad = On.
- The Quad = Off, and SELECTIONPREVIEW system variable = 1 or 3.

Select a property to edit it, much like you would do in the **Properties** panel.



When the Quad is On, you can go back and forth between the Quad commands and the Rollover Tips by clicking the title bar of the Quad/Rollover Tips feature.

4.8 Prompt menus

When executing a command, BricsCAD prompts you in the Command line or the Status bar. Optionally, command options are also available in a prompt menu.

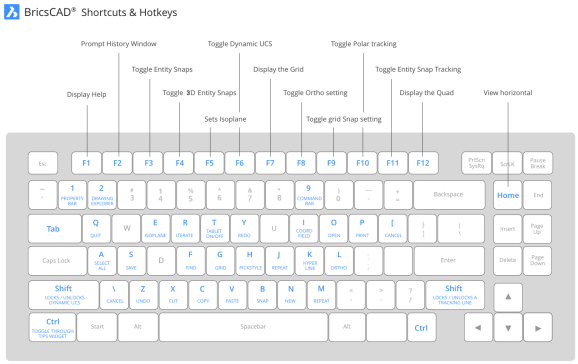
4.8.1 Controlling the display of prompt menus

Whether prompt menus are displayed and where they are located is controlled through thePROMPTMENU system variable.

The PROMPTMENUFLAGS system variable controls the content and behavior of prompt menus.

Nota: These system variables are saved in the registry and therefore apply to all drawings.

4.9 Keyboard shortcuts



Keyboard shortcuts can be a great performance enhancer, especially when toggling settings.

The following keyboard shortcuts are available:

Keyboard	Description
F1	Opens the BricsCAD Help. Executes the HELP command.
F2	Toggles the display of the **Prompt History** window. See the GRAPHSCR command.
F3	Toggles the entity snaps ON/OFF. See the OSMODE setting.
F4	Toggles the 3D entity snaps ON/OFF. See the 3DOSMODE setting.
F5	Sets the isometric plane; toggles the value of the SNAPISOPAIR setting. Applies only when the SNAPSTYL setting = 1.
F6	Toggles the Dynamic UCS feature.
F7	Toggles the display of the grid (see the GRIDMODE setting).
F8	Toggles the ORTHOMODE setting ON/OFF. Disables bitcode 8 of the AUTOSNAP setting, if set.
F9	Toggles Snap ON/OFF (see the SNAP command).

Keyboard	Description
F10	Toggles the Polar Tracking setting. Sets the ORTHOMODE setting OFF, if ON. Toggles bitcode 8 of the AUTOSNAP setting ON/OFF.
F11	Toggles the Entity Snap Tracking setting. Toggles bitcode 16 of the AUTOSNAP setting ON/OFF.
F12	Toggles the display of the Quad cursor menu. See the QUADDISPLAY setting.
Alt + F8	Launches the VBARUN command.
Alt + F11	Launches the VBAIDE command.
Ctrl + 1	***Nota:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key. Toggles the Properties panel.
Ctrl + 2	***Nota:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key. Opens the Drawing Explorer dialog box.
Ctrl + 9	***Nota:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key. Toggles the display of the Command line. Executes the COMMANDLINE command or the COMMANDLINEHIDE command.
Ctrl + 0	***Nota:*** Do not use the numeric keypad. On AZERTY-keyboards: do not use the Shift key when pressing the number key. Toggles the display of user interface entities as specified by the CLEANSCREENOPTIONS setting. Executes the CLEANSCREENON command or the CLEANSCREENOFF command.
Ctrl + A	Selects all entities.
Ctrl + B	Toggles Snap ON/OFF (see the SNAP command).
Ctrl + C	Copies the selection to the clipboard (COPYCLIP command).

Keyboard	Description
Ctrl + Shift + C	Copies the selection along with a base point (COPYBASE command).
Ctrl + E	Sets the isometric plane; iterates the value of the SNAPISOPAIR setting. Applies only when the SNAPSTYL setting = 1.
Ctrl + F	Launches the FIND command.
Ctrl + G	Toggles the display of the grid (GRIDMODE setting).
Ctrl + H	Toggles the value of the PICKSTYLE setting, which controls the selection of groups and associative hatches.
Ctrl + I	Sets the readout of the coordinate field in the Status bar (COORDS setting).
Ctrl + J	Repeats the previously issued command.
Ctrl + K	Attach a hyperlink to an object or modify an existing hyperlink (HYPERLINK command).
Ctrl + L	Toggles the ORTHO setting.
Ctrl + M	Repeats the previously issued command.
Ctrl + N	Creates a new drawing (NEW command).
Ctrl + O	Opens an existing drawing (OPEN command).
Ctrl + P	Prints the current drawing (PRINT command).
Ctrl + Shift + P	Toggles the display of the Properties panel (PROPERTIES command and PROPERTIESCLOSE command).
Ctrl + Q	Quits the application; prompts to save changes (QUIT command).
Ctrl + R	Iterates through viewports.
Ctrl + S	Saves the current drawing (QSAVE command).
Ctrl + T	Turns the tablet on/off (TABLET command).
Ctrl + V	Pastes the contents of the Clipboard. (PASTECLIP command).

Keyboard	Description
Ctrl + Alt + V	Pastes the contents of the Clipboard in a specified format (PASTESPEC command).
Ctrl + Shift + V	Pastes the contents of the Clipboard as a block (PASTEBLOCK command).
Ctrl + X	Cuts the selection and puts it on the clipboard (CUTCLIP command).
Ctrl + Y	Redoes the last action undone (REDO command).
Ctrl + Z	Undoes the last action (UNDO command).
Ctrl + [	Cancels the running command.
Ctrl + \	Cancels the running command.
Ctrl + Home	Creates or opens the Start tab (GOTOSTART command).
Shift + F2	Toggles the display of the Command line (COMMANDLINE command and COMMANDLINEHIDE command).
Shift + F3	Turns the Status bar on/off (SCROLLBAR command).
Shift + F4	Turns the Scroll Bars on/off (SCROLLBAR command).
Shift + F8	Launches the Visual Basic Project Manager (VBAMAN command).
Shift + F11	Launches the VBA COM Add-In Manager (ADDINMAN 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.

Keyboard	Description
Shift + Up Arrow	Moves the view up.
Shift + Down Arrow	Moves the view down.

4.10 Digitizing tablet

A digitizing tablet, also called a graphics tablet, a graphics 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.

The TABLET command configures and calibrates a digitizing tablet and toggles the tablet mode.

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.

4.10.1 Using a digitizing tablet in BricsCAD

In order to prepare BricsCAD for the use of a digitizing tablet, do the following:

- Install the appropriate driver for your tablet.
- Make sure Wintab32.dll is present 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 of it is used 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 Bricscad drawing.

4.10.2 Downloading the tablet overlay and the CUI files

Click here to download the file Tablet.zip.

https://www.bricsys.com/bricscad/tools/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.

4.10.3 Loading the overlay CUI file

1 Extract the overlay CUI files to the **Support** folder of the **roamable root** folder (see BricsCAD User Files).

2 Load **Tablet.cui** as a partial CUI. The next time you start BricsCAD, **Tablet.cui** will be loaded automatically.

Nota:

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

4.10.4 Switching the tablet ON / OFF

Do one of the following:

- Click the **TABLET** field in the Status bar.
- Press the **F4** function key.
- Launch the TABLET command.

4.10.5 Configuring the tablet

1 Launch the TABLET command.

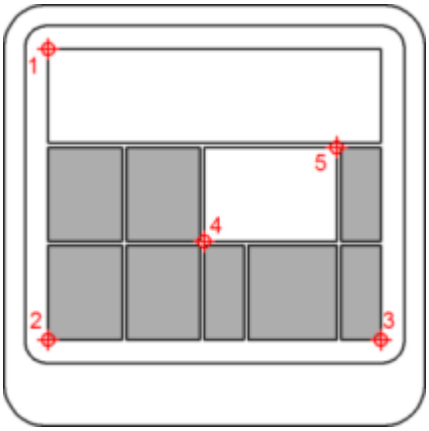
2 Choose the **Configure** option.

3 **You are prompted**: Do you want to align the tablet overlay? Yes/No/<No>.

Type **Y** in the Command line and press Enter or choose **Yes** in the prompt menu.

You are prompted: Digitize upper left corner of the overlay.

4 Click on the mark in the upper left corner (1) of the digitizing tablet overlay.



You are prompted: Digitize the lower left corner of the overlay.

5 Click on the mark in the lower left corner (2) of the digitizing tablet overlay.

6 **You are prompted**: Digitize the lower right corner of the overlay.

7 Click on the mark in the lower right corner (3) of the digitizing tablet overlay.

8 **You are prompted**: Digitize the lower left corner of the screen pointing area.

9 Click on the mark at position (4) on the of the digitizing tablet overlay.

10 **You are prompted**: Digitize the upper right corner of the screen pointing area.

11 Click on the mark at position (5) on the of the digitizing tablet overlay.

12 **You are prompted**: Tablet configured.

4.10.6 Calibrating 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 Launch the TABLET command.

2 Choose the **Calibrate** option.

3 **You are prompted**: Digitize point #1:

Click a point on the digitizing tablet.

You are prompted: Enter coordinates for point #1:

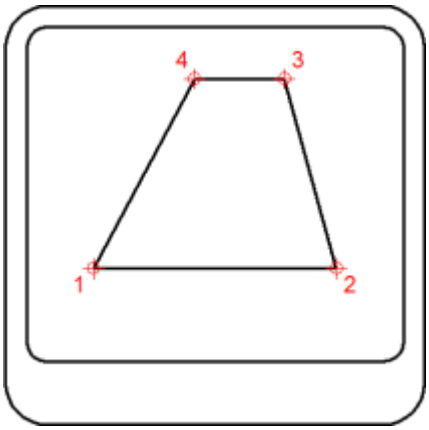
4 Do one of the following:
- Click a point in the BricsCAD drawing window.
- Enter the coordinates (X,Y) in the Command line.

5 Repeat step 3 to specify the following calibration point.

6 **You are prompted**: Digitize point #3 (or ENTER to end):

7 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.

8 BricsCAD responds:
- When 2 calibration points are specified: Tablet calibrated.
- When 3 or more than 4 calibration points are specified: Select transformation type Orthogonal/Affine:
- When 4 calibration points are specified: Select transformation type Orthogonal/Affine/Projective:



9 Select the transformation type.

10 Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.



- **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.

4.11 Panels

Panels are interactive UI elements that can remain open while you use other commands. They can be docked or floating and they can be stacked. And you can control the amount of data displayed on the panel. The location and size of the panel depends on the current workspace or its previous appearance.

You can dock panels at the top, bottom, left or right within the BricsCAD application window. They can be docked alongside other panels or toolbars but are always constrained by the size of the application window. The drawing area, within the BricsCAD application window, automatically reduces to accommodate the docked panels. You can move a docked panel using the grip bar, resize it with the stretch icon, and close it with the close icon.

Floating panels can be placed anywhere on any available monitor. They can be moved and resized beyond the boundaries of the BricsCAD application window. If a floating panel overlaps the drawing area, it floats above it, allowing the drawing area to maintain its size. You can move a floating panel using the title bar, resize it with the stretch icons, and close it with the close icon. Regardless of where a floating panel is

located, the stretch icon appears when you pass the cursor over any edge of the panel. The title bar and close icons are always on the top edge of a floating panel.

Multiple panels can be combined into a panel stack to conserve screen space. Each panel stack can be docked or floating, offering the same functionality as individually docked or floating panels. You can display panel stacks with tabs or icons.

By default, panels are docked at the left or right hand side of the screen as defined in the **Panels** node of a workspace in the**Workspaces** tab of the Customizedialog box:

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

The stack type is defined in the**Workspaces** tab of the Customizedialog box through the **Stack Type** property of a workspace.

The possible stack types are:

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

4.11.1 Opening or closing a panel

1 Move the cursor over a toolbar or ribbon panel, then right click.

A context menu displays.

2 Place the cursor over **Panels**.

A list of available panels displays.

Panels that are currently open are checked.

3 Click a panel in the menu to toggle its display.

- Some panels can be opened using a command as indicated in the table below.
- The -TOOLPANELcommand allows to show, hide or toggle the display of panels at the Command line.

4.11.2 Available panels

Icon	Title	Command
	Animation editor panel na página 169	
	Attachments panel na página 171	
	BIM BCF panel na página 173	
	BIM compositions panel na página 174	

Icon	Title	Command
	BIM profiles panel na página 176	
	BIM project browser panel na página 178	
	BOM manager panel na página 181	
	Bricsys 24/7 panel na página 188	
	Command line panel na página 190	
	Content browser panel na página 191	
	Details panel na página 193	
	Drawing customizations panel na página 195	
	Layers panel na página 202	
	Library panel na página 208	
	Mechanical browser panel na página 211	
$f(x)$	Parameters and constraints panel na página 215	
	Properties panel na página 217	
	Render materials panel na página 224	
	Report panel na página 226	
	Ribbon	

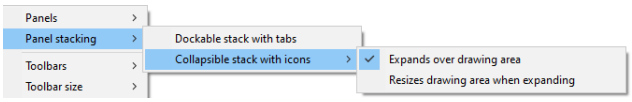
Icon	Title	Command
	Sheet sets panel na página 228	
	Structure panel na página 231	
	Tips panel na página 233	
	Tool palettes panel na página 234	Tool palettes na página 855

4.11.3 Setting the stack type

1 Move the cursor over a toolbar or panel, then right click.

A context menu displays.

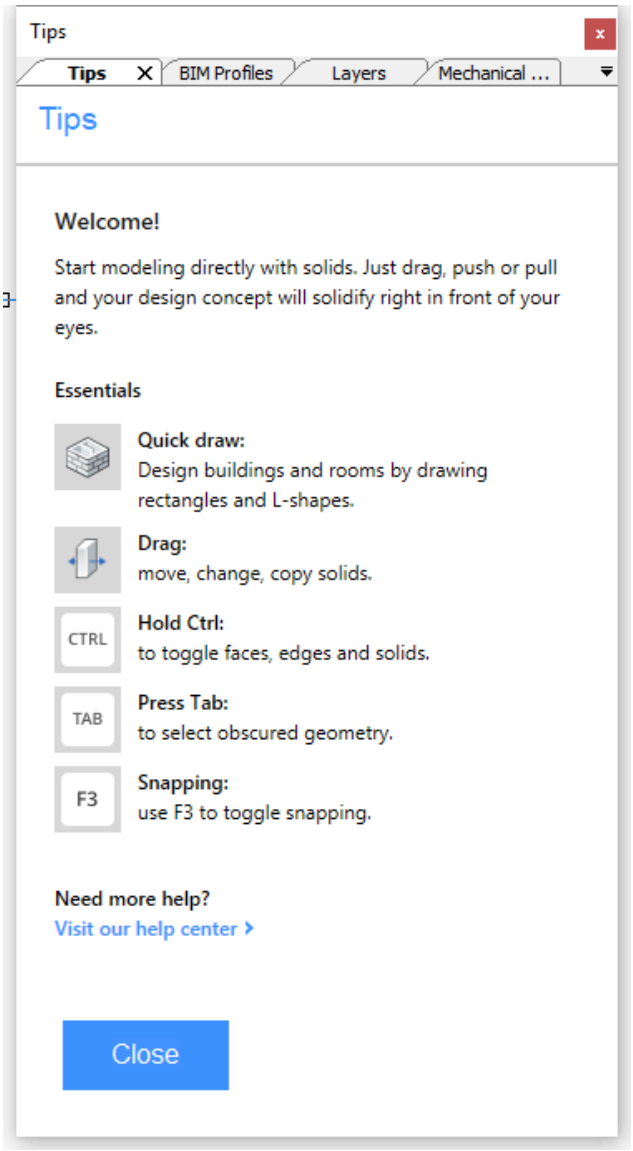
2 Choose **Panel stacking**.



3 Select the desired stack type.

Dockable stack with tabs

Panels are docked in a tabbed panel.

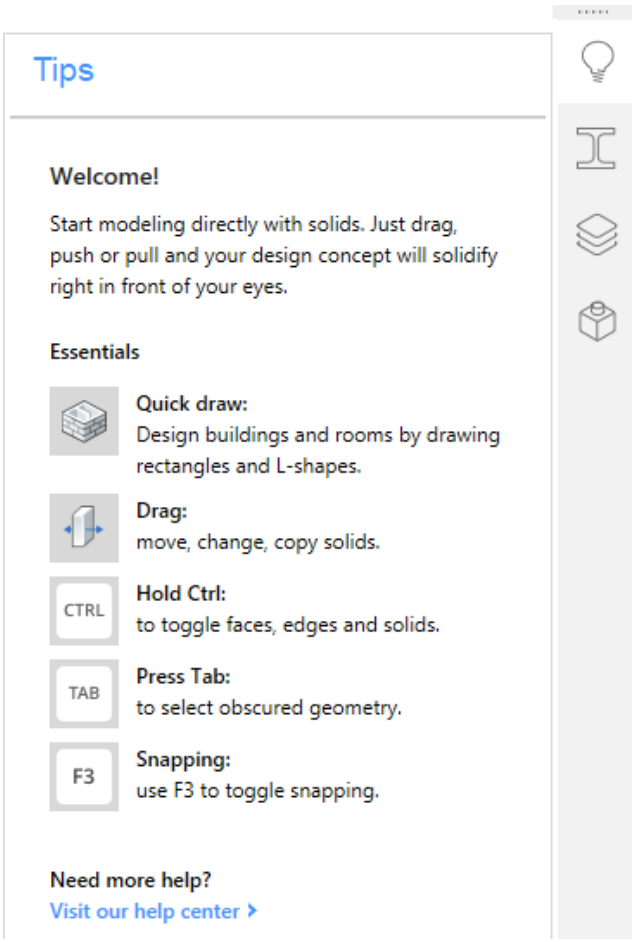


- Click a tab to display the panel.
- Press and hold the left mouse button to drag a tab to re-arrange the tab order.
- Double click a tab to make the panel floating.
- Double click the title bar of floating panel to dock it at its previous position.

Collapsible stack with flyouts

Panels are docked in an icon strip at the left and/or right hand side of the screen.

- **Expands over drawing area**: flyout panels overlap the drawing area, thus hiding part of the drawing.
- **Resizes drawing area**: flyout panels narrow the drawing area.

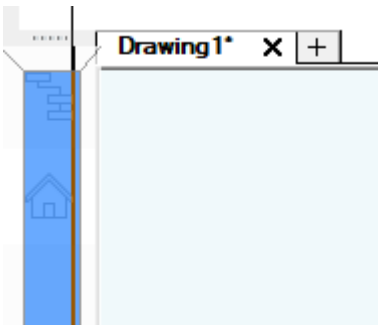


- Click an icon to expand a panel.
- To collapse an expanded panel click its icon again.
- Press and hold the left mouse button to drag an icon to re-arrange the icon order.
- Double click the icon to make the panel floating.
- Double click the title bar of a floating panel to dock it at its previous position.

4.11.4 Moving a panel to the other icon tab strip

1 Drag the icon to the other icon strip.

The docking location selector displays. The blue zone indicates the selected docking position.

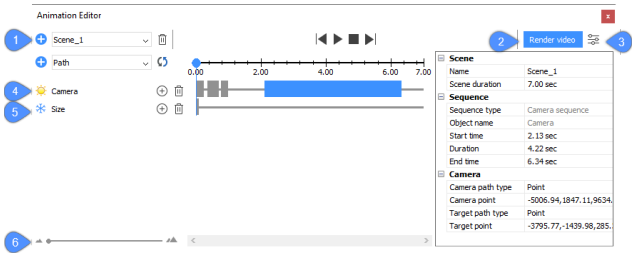


2 Move the cursor over the center docking location then release the left mouse button.

Nota: TheLOCKUIsystem variable controls whether the current position of the panels is locked or not. Press and hold the Ctrl-key to override. Click the**LOCKUI**field in theStatus Barto toggle the locking of the user interface.

4.11.5 Animation editor panel

The Animation Editor Panel allows you to create schedules for multi-parametric animations with complex camera movements and render them as 2D videos.



1 Add scene

2 Render video

3 Configurator

4 Disable timeline

5 Enable timeline

6 Slider

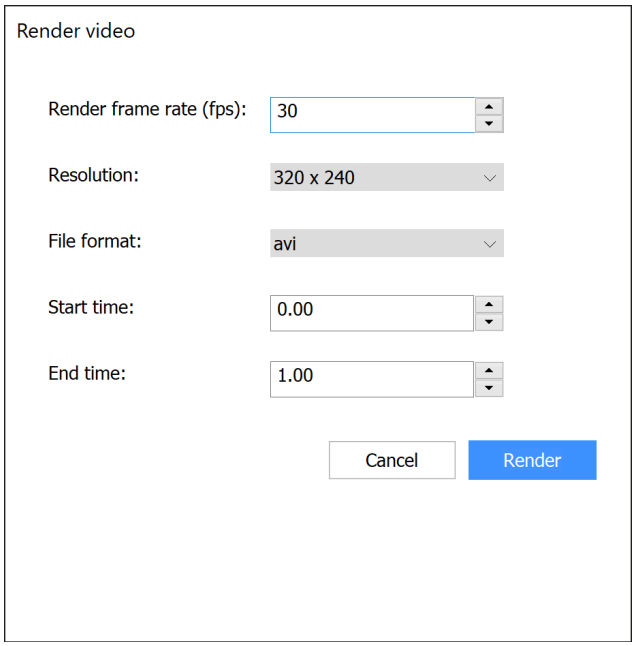
4.11.5.1 Add scene

Scene name.

Opens the Add Scene dialog box name to insert the scene name and its duration (s).

4.11.5.2 Render video

Opens the Render video dialog box to set the render options:



4.11.5.3 Render frame rate (fps)

Sets the framerate of the video.

4.11.5.4 Resolution

Sets the resolution of the video by selecting it from the drop-down list of supported resolutions.

4.11.5.5 File format

Sets the format of the output video. You can select a file format from the drop-down list:

- avi (AVI Animation)
- mpg (MPEG Animation)
- wmv (WMV Animation)

4.11.5.6 Start time

Sets the time position of the animation to start the render.

4.11.5.7 End time

Sets the time position of the animation to end the render.

After pressing the Render button, the Save Video as dialog box opens to allow you to save the rendered video.

4.11.5.8 Configurator

Toggles the configurator panel.

4.11.5.9 Disable timeline

Disables timeline effects.

4.11.5.10 Enable timeline

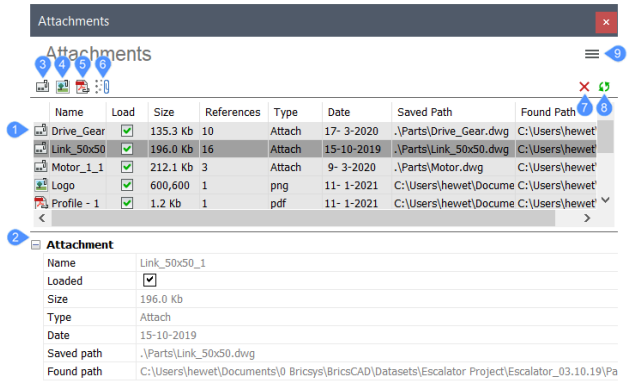
Enable timeline effects.

4.11.5.11 Slider

Move the slider to change the scale of the timeline.

4.11.6 Attachments panel

The Attachments panel offers a central location for viewing and managing attached drawings, images, PDF files, and point clouds.



1 Attachment list

2 Properties

3 Attach DWG

4 Attach Image

5 Attach PDF

6 Attach Point Cloud

7 Detach

8 Refresh

9 Menu

4.11.6.1 Attachment list

Displays a list of attached filesand their properties. Column headers describe theproperties of each attachment. You may sort theattachment listin order or reverse orderforanyof the properties byclickingonce or twice onthecolumn header.A right-click menuon the column headers enables you to customize them. A right-click menu on the attachments offers additional tools.

4.11.6.2 Name

Specifies the attachment name. By default, the attachment name is the same as the file name. To rename an attachment, double-click the name and enter the new name. Renaming an attachment does not change the file name.

4.11.6.3 Load

Specifies the load status of the attachments. Attachments with the Load property enabled are loaded and displayed in the drawing editor. Attachments with the Load property disabled maintain their attachment information, such as scale and location, but are not loaded or displayed in the drawing editor.

4.11.6.4 Size

Specifies the file size of the attachments.

4.11.6.5 References

Specifies how many times the attachments are referenced in the current drawing.

4.11.6.6 Type

Specifies the attachment type. Click the attachment type to toggle between Attach and Overlay.

4.11.6.7 Date

Specifies the date the attached files were last saved.

4.11.6.8 Saved Path

Specifies the path, including location and file name, of the attachments. To change the saved path, click it and enter the new path or select the Browse button to display the Select reference file dialog box. The Browse button is only displayed after you click on the saved path.

4.11.6.9 Found Path

Specifies the path, including location and file name, of the attachments. The Found path may or may not be the same as the Saved path.

4.11.6.10 Properties

Displays the properties of the selected attachment in a row format. These are the same properties displayed in the Attachments list with a column format.

4.11.6.11 Attach DWG

Displays the Open file to attach dialog box like theXREFcommand.

4.11.6.12 Attach Image

Displays the Select image file dialog box like theIMAGEATTACHcommand.

4.11.6.13 Attach PDF

Displays the Select PDF underlay file dialog box like thePDFATTACHcommand.

4.11.6.14 Attach Point Cloud

Displays the Preprocessed Point Cloud Data File dialog box like thePOINTCLOUDATTACHcommand.

4.11.6.15 Detach

Detaches the selected attachment from the drawing.

4.11.6.16 Refresh

Updates the display.

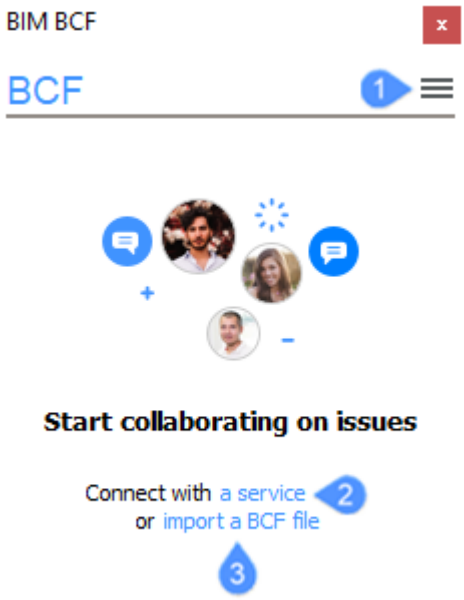
4.11.6.17 Menu

Grid View
Displays the attachments in a detailed grid view.

Tree View
Displays the attachments in a structured tree view.

4.11.7 BIM BCF panel

The BIM BCF panel allows you to link information such as a comment, a screenshot, a list of involved objects and a camera viewport to an issue.



1 Menu

2 Connect with a service

3 Import a BCF file

4.11.7.1 Menu

Opens the main menu of BIM BCF panel.

Import BCF file
Allows to import BIM Collaboration Format Files (*.bcf,*.bcfzip).

Log in with custom service
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

Log in with custom service
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

Refresh
Makes sure you see the latest updates in case you are working on the cloud.

Clear sources
Disconnects you with the cloud service, in case you are online, or closes the .bcf file, in case you are working on an imported .bcf file.

4.11.7.2 Connect with a service

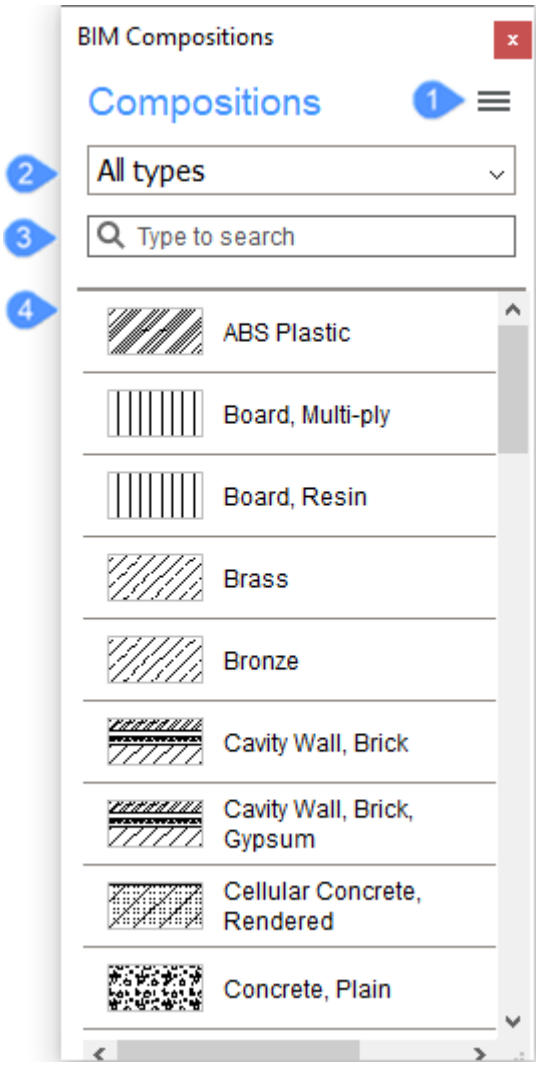
Opens the Custom log in dialog box.

4.11.7.3 Import a BCF file

Displays the Select a BIM Collaboration Format file to import dialog box.

4.11.8 BIM compositions panel

The BIM composition panel allows you to assign compositions to any solid in a BIM model.



1 Menu

2 Compositions types list

3 Search

4 Available compositions

4.11.8.1 Menu

Open material dialog...
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

Open composition dialog...
Opens the Compositions dialog box that allows you to create, modify and delete compositions.

Open project dialog...
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

Show only compositions in project
Displays only the compositions that are used in the project.

4.11.8.2 Compositions types list

Displays the compositions of the selected category (all types, generic, roof, slab, wall) in the Library database.

4.11.8.3 Search

Filters the list of compositions. Only compositions that contain the current search term are displayed.

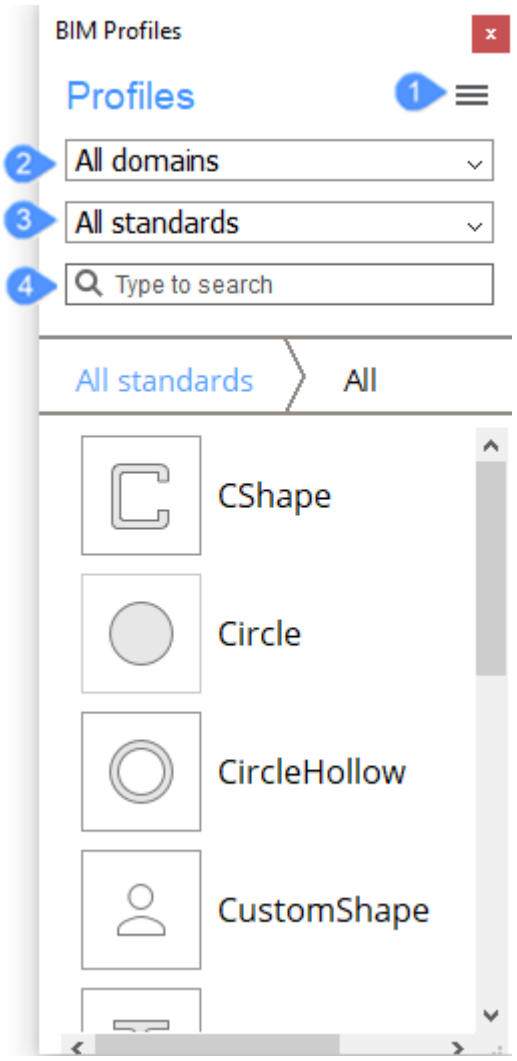
4.11.8.4 Available compositions

Displays all the available compositions.

When you double-click on a composition the Compositions dialog box will open.

4.11.9 BIM profiles panel

The BIM Profiles panel offers a central location to access standard and custom profiles.



1 Menu

2 Domains

3 Standards

4 Search bar

4.11.9.1 Menu

Open material dialog...
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

Open profile dialog...
Opens the Profiles dialog box that allows you to create, modify and delete profiles

Open project dialog...
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

Show only profiles in project
Displays only the profiles that are used in the project.

4.11.9.2 Domains

Filter profiles based on the domain.

4.11.9.3 Standards

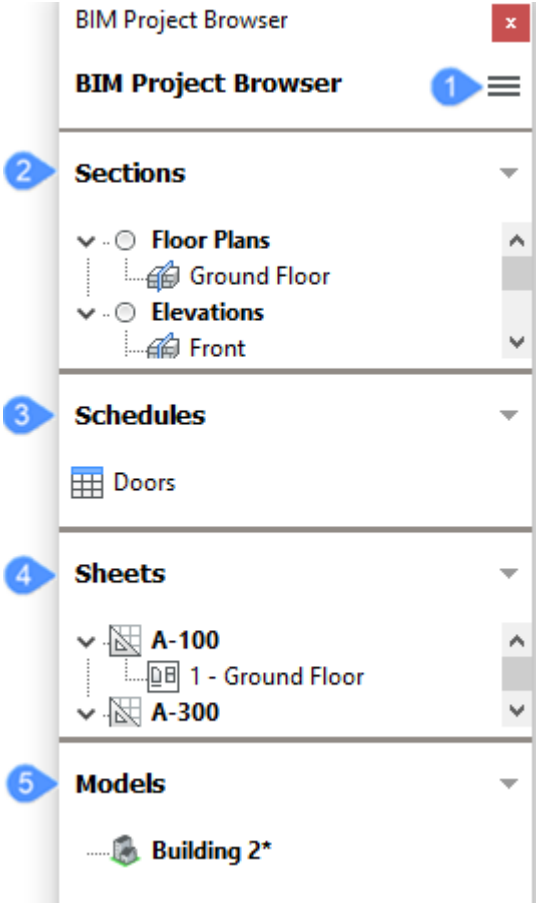
Filter profiles based on the country standard.

4.11.9.4 Search bar

Search for specific profiles using the search bar.

4.11.10 BIM project browser panel

The BIM project browser panel offers a central location to manage your BIM projects.



1 Menu

2 Sections

3 Schedules

4 Sheets

5 Models

4.11.10.1 Menu

Refresh
Refreshes the BIM Project Browser panel to match changes in the model.

Project Setup
Opens the Project Setup dialog box that gives a complete overview of the project's files, in different categories.

Sheetset Setup
Opens the Sheetset Setup dialog box that allows to set up the complete set of sheets required to fit all bimsections in the project.

Enable Background Update
Calculates section results automatically, without interrupting the user workflow.

4.11.10.2 Sections

Displays a list of project sections organized by type: floor plans and elevations. You can expand a section type to see its sections.

A context menu displays when you right-click on a section:

- **Section Properties...**: opens the Section Properties dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: selects the section entity in the project model drawing.
- **Rename**: renames the section entity.
- **Update**: recalculates the section result.
- **Display Section Result**: displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

4.11.10.3 Schedules

Displays a list of project schedules.

A context menu displays when you right-click on the Schedules section.

- **Add Schedule (Wizard)...**: opens the Wizard Page dialog box that allows you to create a schedule based on a new or an existing data extraction definition.
- **Add Schedule (From dxd)...**: opens the Create Schedule dialog box that allows you to quickly add a BIM schedule, without the creation of a new .dxd file.

A context menu displays when you right-click on a schedule.

- **Schedule Properties...**: opens the Schedule Properties dialog box that allows you to modify the properties of a schedule.
- **Create/Update Table**: creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: displays the schedule in the resulting drawing. If not already open, the drawing is opened.

- **Edit**: opens the Wizard Page dialog box that allows you to edit the schedule.
- **Remove...**: displays the Remove dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

4.11.10.4 Sheets

Displays a list of project sheets. You can expand a sheet name to see its sheet views.

A context menu appears when you right-click on the Sheets section:

- **Sheetset Setup...**: opens the Sheetset Setup dialog box that allows you to create a layout from scratch.
- **Add Subset**: adds a subset in the Sheets tab.
- **Add Sheet...**: adds a new sheet in a subset in the Sheets tab.
- **Sheetset Options...**: opens the Sheet sets tab in the Settings dialog box that allows you to modify the sheet sets options.

A context menu displays when you right-click on a sheet name:

- **Sheet Properties...**: opens the Sheet Properties dialog box that allows you to modify the properties of a sheet.
- **Open Sheet**: opens the corresponding sheet.
- **Remove...**: opens the Remove dialog box that allows you to choose between **Result Drawing** and **Sheet** and removes the sheet.

A context menu displays when you right-click on a sheet view:

- **View Properties...**: opens the Sheet View Properties dialog box that allows you to modify the properties of a sheet view.
- **Display View**: displays the corresponding result and zooms in on its viewport.
- **Update**: updates the sheet view. It executes the BIMSECTIONUPDATE command for the related BIM section.

A context menu displays when you right-click on a subset name:

- **Subset Properties...**: opens the Subset Properties dialog box that allows you to modify the properties of a subset.

4.11.10.5 Models

Displays a list of project models.

A context menu displays when you right-click on a model with the following options:

- **Model Properties...**: opens the Model Properties dialog box that allows you to see more information about the model.
- **Close Drawing**: for a currently open model,it closes the model.
- **Open Drawing**: for a currently closed model, it opens the model.
- **Remove from Model List**: removes the drawing from the project model drawings.

4.11.11 BOM manager panel

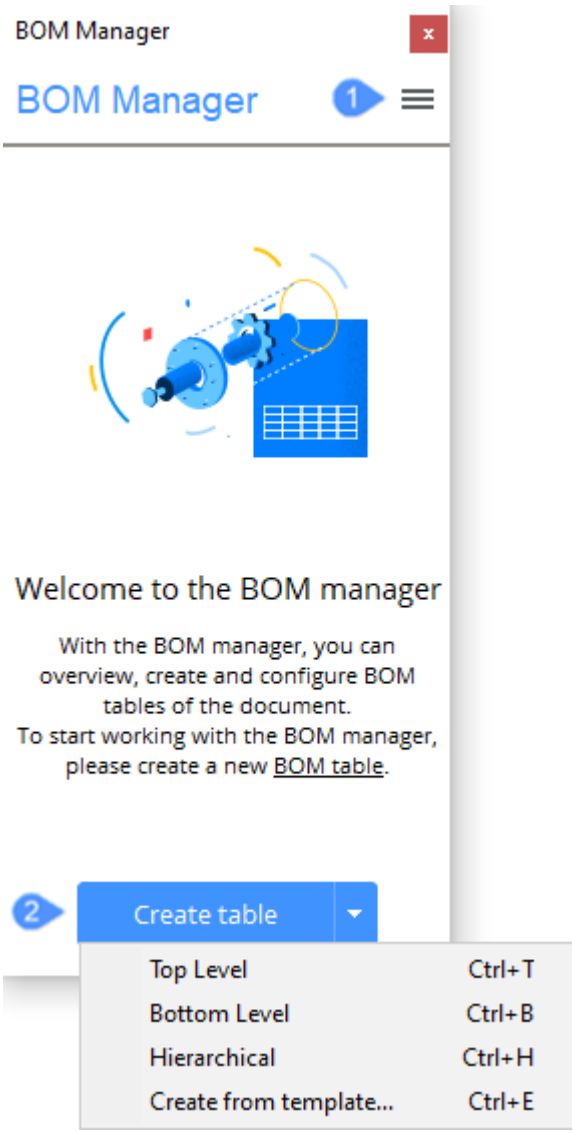
The BOM Manager panel allows you to browse, create and edit bills of materials (BOMs) in the current drawing.

The configuration of a bill of materials can be saved as a template file and reused later in other drawings.

With the Bill of Materials Manager, you can:

- Browse existing bills of materials in the drawing.
- Create a new bill of materials using either a standard type (such as top-level, bottom-level, hierarchical) or an existing BOM template.
- Edit bills of materials:
- Add or remove columns (includes formula columns).
- Move selected columns to new positions.
- Rename the columns.
- Configure how the BOM table is sorted.
- Configure the footer value for each individual column.
- Configure the units and how they are displayed for each column.
- Configure the width of each column with column properties.
- Configure the visibility of each individual column.
- Filter the components with a filter expression.
- Change the title of both the title row and the footer row.
- Search for components in the table.
- Highlight and select component instances.
- Save the BOM configuration as a template file.
- Load a BOM table configuration from a template file.
- Place the BOM table in a drawing.
- Delete BOM tables.

A Mechanical or Ultimate license is required to modify or create bills of materials. However, the manager is available with a Pro license.



1 Menu

2 Create a table

4.11.11.1 Menu

Opens the main menu.

New BOM
Opens a submenu, which includes:

- **Copy the current table**: adds a new BOM table using the template of the current table.

BOM tables in document
opens a submenu.

- **Go to first table**: switches to the first BOM table in the document.
- **Go to previous table**: Switches to the previous BOM table in the document.
- **Go to next table**: Switches to the next BOM table in the document.

- **Go to last table**: Switches to the last BOM table in the document.
- **<Name of the BOM>**: Switches to the corresponding BOM table in the document.

Place/Update BOM table
Starts the interactive placement of the table in the drawing. If the table is already in the drawing, this tool will update the table.

Export to CSV
Saves the content of the current table to a CSV file.

Replace from template…
Replaces the current BOM configuration with the one from the template.

Save as template…
Specify the location of the BOM template file.

Delete current BOM
Deletes the current BOM table from the document.

Set filter
Opens the Filter Configuration dialog box, which allows you to specify the filter configuration.

Selection set
Opens a submenu.

- **Entire model**: The BOM table will be created for the entire model.
- **Subassembly**: The BOM table will be created for a selected instance of a subassembly.
- **Drawing view**: The BOM table will be created for the parts selected for the corresponding drawing view, regardless of their visibility in the viewport.
- **Layout viewport**: The BOM table will be created for the selected layout viewport.
- **Custom selection**: The BOM table will be created only for the selected parts.
- **Show selection**: Shows the current selection for the BOM table.
- **Edit**: Allows you to edit the selection.

Refresh
Updates the current BOM and the list of all BOM tables in the drawing.

Insert column after
Inserts a column from the **Column selector** after the selected column.

Insert column before
Inserts a column from the **Column selector** before the selected column.

Nota: If no column is selected, a new column is added after the last column in the table.

Insert formula column
Opens a formula editor, which allows you to configure a formula column.

Insert template column
Opens a template editor, which allows you to configure a template column.

Edit evaluated column
Allows you to edit the selected formula or template column.

Delete column
Removes the selected column from the table.

Move column left
Moves the selected column(s) one column to the left.

Move column right
Moves the selected column(s) one column to the right.

Make column visible
Shows the selected column in the associated table in the drawing.

Make column invisible
Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.

Associated table
Opens a submenu.

- **Zoom to**: Zooms to the table. If necessary, sets the table layout as the current one.
- **Select**: Selects the table. If necessary, sets the table layout as the current one.
- **Delete**: Deletes the table from the document.
- **Detach**: Detaches the table. The table in the document is no longer updated when the BOM table is modified.

Footer
Opens a submenu, for each of the selected columns.

- **No footer**: Removes the footer.
- **Sum**: Displays the sum in the footer.
- **Average value**: Displays the average in the footer.
- **Minimal value:**: Displays the minimum value in the footer.
- **Maximal value:**: displays the maximum value in the footer.

Sorting
Opens a submenu.

- **Automatic order**:Sorts the table automatically**.**
- **Natural order**: Sort the table by the natural order of component instances in the document.
- **Ascending order**: Sorts the selected columns in ascending order.
- **Descending order**: Sorts the selected columns in descending order.

Aggregate function
Opens a submenu.

- **No function**: Removes the aggregate function.
- **Sum**: Displays the sum of all values in the group.
- **Average value**: Displays the average of all values in the group.
- **Minimal value**: Displays the minimum of all values in the group.
- **Maximal value**: Displays the maximum of all values in the group.

- **Concatenation**: displays a concatenation of all values in the group in ascending order, separated by a specified delimiter.
- **Concatenation with count**: Displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.

Disable/Enable warnings
Controls the display of warning messages during the creation or update of BOM tables.

4.11.11.2 Create a table

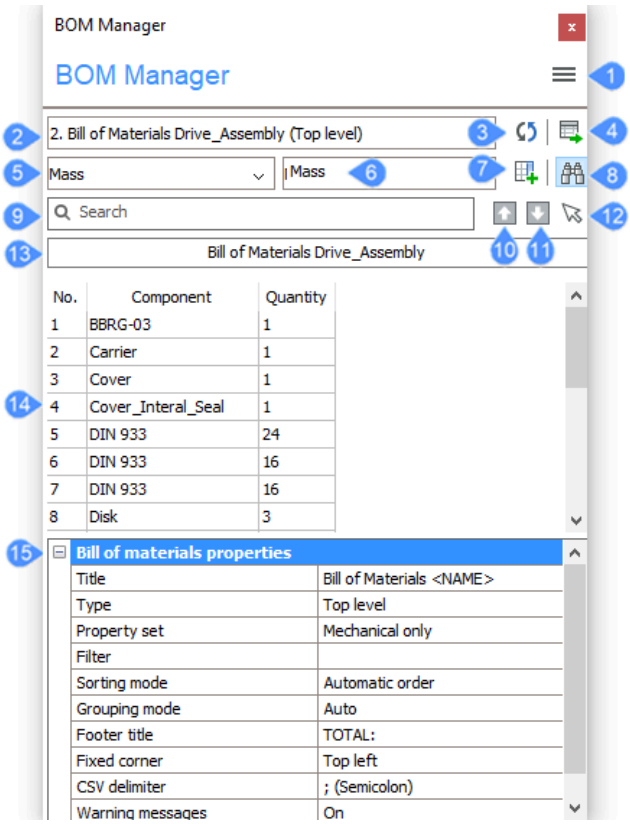
Top level
Creates a new Top level BOM.

Bottom level
Creates a new Bottom level BOM.

Hierarchical
Creates a new Hierarchical BOM.

Create from template
Opens a dialog box, which allows you to select a BOM Table template (*.BOM) file.

To start working with the BOM manager, you will have to create a new BOM table.



1 Menu

2 BOM Selector

3 Refresh

4 Place/update table

5 Category selector

6 Column selector

7 Insert column after

8 Find

9 Search

10 Find previous

11 Find next

12 Select all found

13 BOM title

14 BOM table

15 BOM and Column Properties

BOM Selector
Selects the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated in the current state of the drawing.

Refresh
Updates both the current BOM and the list of all BOM tables in the drawing.

Place/update table
Starts the interactive placement of a table in a drawing. If the table has already been placed in the drawing, it updates it. All edit operations automatically update the associated table in the drawing.

Category selector
Select the category of the properties that are displayed in the column selector. The category can be added to the current BOM pressing button no. 7.

Column selector
Select the column to add to the current BOM.
You can type here the formula for a formula field. To add a formula, type an expression starting with ”=”. To add a template expression type “$”, to refer columns, use <>. For example, to refer to the Mass column you will type: “=<Mass>”.
When you add a formula field and press button no. 7 the Formula column_1 dialog box will appear.

Insert column after
If a column is selected, it will insert the column from the Column selector after the selected column in the table.
If no columns are selected, the new column will be inserted after the last column in the table.

Find
Toggles the search field on and off. When you toggle on this button, buttons no. 9, 10, 11 and 12 will appear.
If you click on the BOM Manager panel, you can toggle the search field using **CTRL+F**.

Search
Type a string to search and press Enter; wildcards are supported.

Find previous
Selects the previous match on the BOM table.
You can also use **Shift+F3**.

Find next
Selects the next match in the BOM table.
You can also use **F3**.

Select all found
Selects all the items searched in the search field.

BOM title
Displays the title of BOM table. You can type the name you want instead of <NAME>. Press Enter or exit the field to apply the changes.

BOM table
Displays the content of the BOM table.

Right-click a column label offers additional tools:

- **Add template column from selected**: opens a template editor to configure a new template column. The selected columns will be automatically added to the template expression.
- **Column role**: opens a submenu.
  - **Number**: the column will be used as a source of numbers by the bmBalloon command.
  - **Name**: the column will be used as a source of names by the bmBalloon command.
  - **Quantity**: the column will be used as a source of quantities by the bmBalloon command.
  - **Regular**: the column will have no specific role.
- **Format**: opens the Format dialog box, which allows you to specify the format for the selected column(s).
- **Set column filter**: opens the Column Filter dialog box, which allows you to configure the column filter.
- **Fit column(s)**: fits the width of the selected column(s) to their content.
- **Fit all columns**: fits the width of all columns to their content.

  Nota: The column width automatically fits the content unless the width has been changed manually.

Right-click on cells offers additional tools:

- **Zoom to**: zooms to the select part(s) in the model.
- **Select**: selects the part(s). Replaces an existing selection.
- **Add to Selection**: adds part(s) to the selection.
- **Remove from Selection**: removes the parts from the selection.
- **Remove highlighting**: removes highlighting.

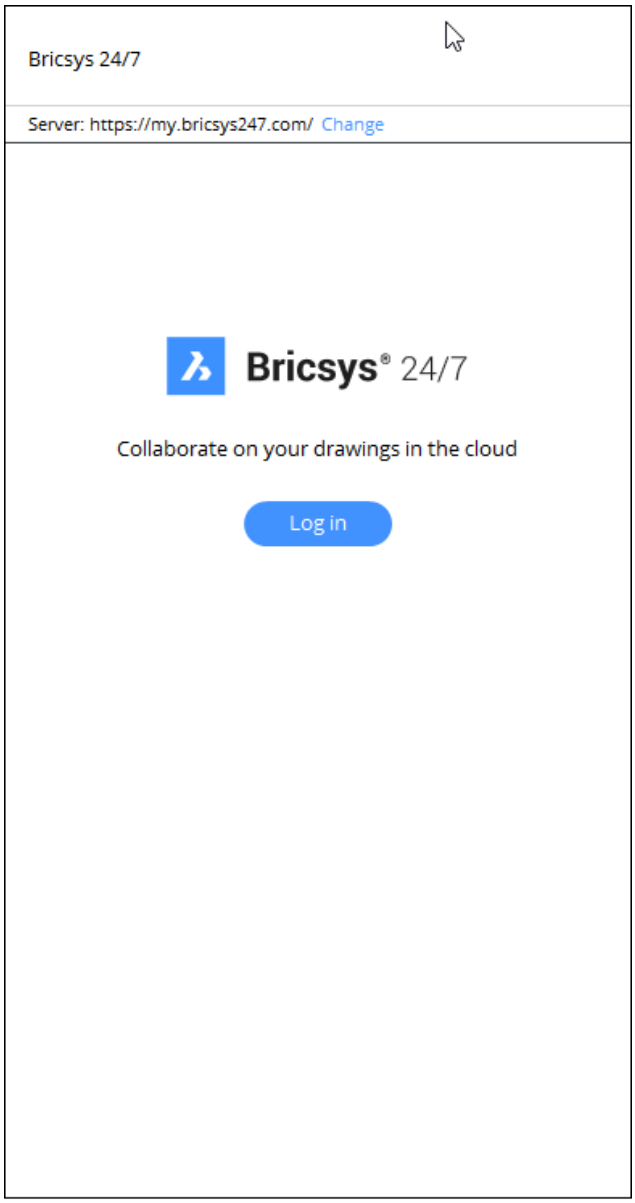
BOM Properties
Displays the BOM and columns Properties. The columns properties will appear after you insert a column on the BOM table.

4.11.12 Bricsys 24/7 panel

The Bricsys 24/7 Panel helps team members to find and share their drawings.

Bricsys 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 the Bricsys 24/7 Panel.

If you are logged out of your 24/7 account a screen will display:



1 Enter your Email address and Password and press the 'Log in' button.

2 (Option) Tick the**Remember Me**checkbox to store the username and password.

3 Once logged on, you will see an overview of your 'in the cloud' Projects, Folders, and Documents.

4.11.12.1 Project

The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.

4.11.12.2 Project Overview

At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.

4.11.12.3 Folder

When you open a project, a list of folders is shown.

4.11.12.4 Folder Overview

At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it.

4.11.12.5 Documents

In a folder you find documents. On the right-hand icons are displayed:

The profile picture or the initials from the author of the document.

The document is in the cloud.

Someone is currently working on the document.

The document is currently uploading.

The document is locked by someone, another person cannot change it.

Right-click on a folder to get a menu with 2 possible actions:

- Download files not locally available.
- Download out of date locally unmodified files.

Right-click on a document to get a menu with possible document actions

- Lock on server
- Unlock on server
- Upload local copy as a revision
- Download latest revision

4.11.13 Command line panel

The Command line panel allows you to launch commands or modify system variables by typing the command or variable name.



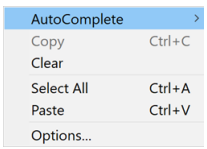
1 Command line

2 Command history

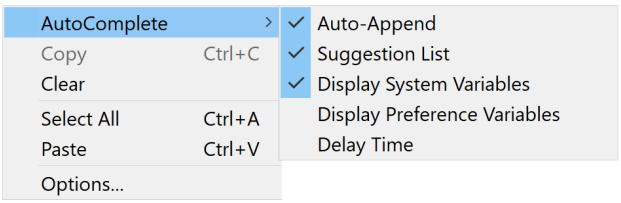
4.11.13.1 Command line

Displays the current command prompt. If no command is active, it prompts you to Enter Command. If a command is active, it displays prompts and options relevant to the current command. Prompt shortcuts are highlighted. You may enter the shortcut characters in the Command line or click on them with the left mouse button.

A right-click menu offers relevant tools.



AutoComplete
Hover over the AutoComplete option. Another context menu displays:



- If **Auto-Append** is checked, the entry is automatically completed if only one possible command is left.
- If **Suggestion List** is checked, the list of possible commands is displayed when typing in the command bar.
- If **Display System Variables** is checked, the system variables are included in the suggestion list.
- The **Delay Time** option sets the time in seconds before automated keyboard features display.

Copy
This option copies the selected text to the clipboard.

Clear
This option clears complete prompt history.

Select All
This option selects all the completed content of the prompt history.

Paste
This option pastes the text from the clipboard to the Command line.

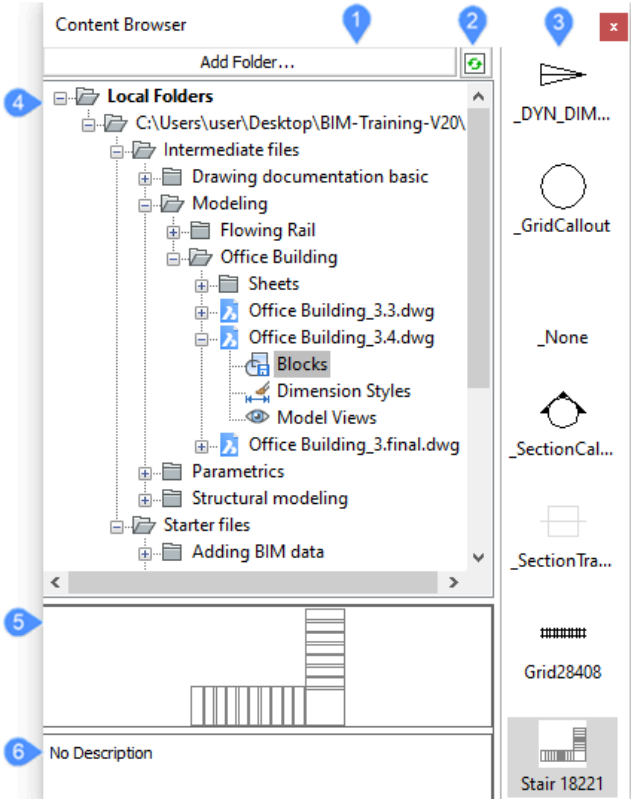
Options
This option displays the Settings dialog box at the Command line section.

4.11.13.2 Command History

Displays the history of the Command line input for the current BricsCAD session.

4.11.14 Content browser panel

The Content Browser panel offers you a quick and convenient way to open drawings, manage your block libraries, copy dimension styles, place views for your entire .dwg files stored on your computer.



1 Add Folder...

2 Refresh

3 Definitions in the selected drawing

4 Local Folders tree

5 Definitions' preview

6 Description

4.11.14.1 Add folder...

Displays the Choose a folder dialog box.

4.11.14.2 Refresh

Select a drawing and press the Refresh button to expand its content (Blocks, Dimension Styles, Model Views).

4.11.14.3 Definitions in the selected drawing

Select Blocks/Dimension styles/Model views to see the definitions used within the drawing.

You can drag & drop a block onto the current drawing and an Insert Block dialog box will appear.

If you drag & drop a dimension style to your drawing a Copy / Paste dialog box will appear.

4.11.14.4 Local Folders tree

Displays the content of a drawing/project. Right-click menus enable you to add or remove folders from within the Folders tree.

4.11.14.5 Definitions' preview

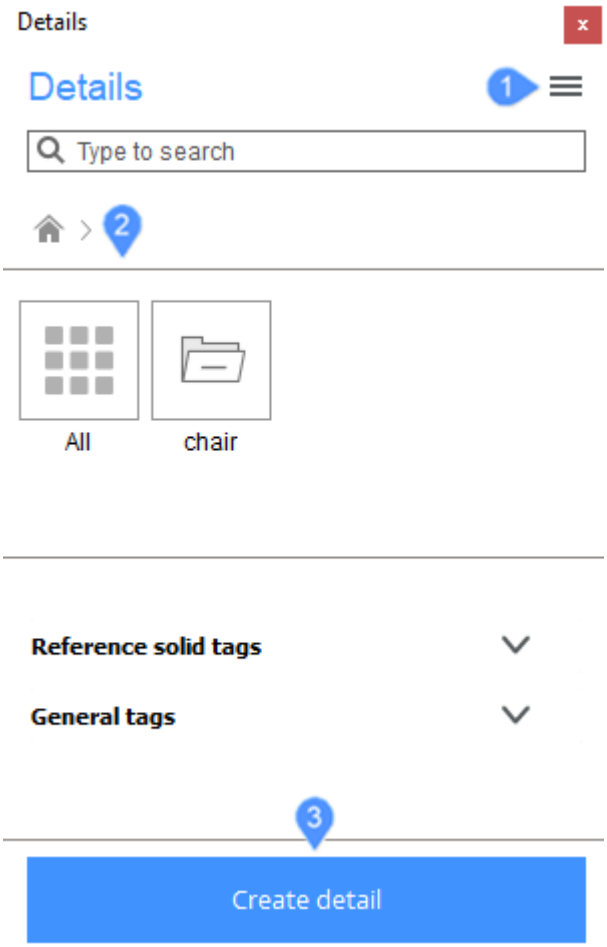
Shows a preview of your drawing/block/dimension style.

4.11.14.6 Description

Displays a description of your selected definition.

4.11.15 Details panel

The Details panel allows you to view details, add tags to details, propagate and adjust details.



1 Menu

2 Details library

3 Create detail

4.11.15.1 Menu

Grid View
Displays the details in a grid view.

List View
Displays the details in a list view.

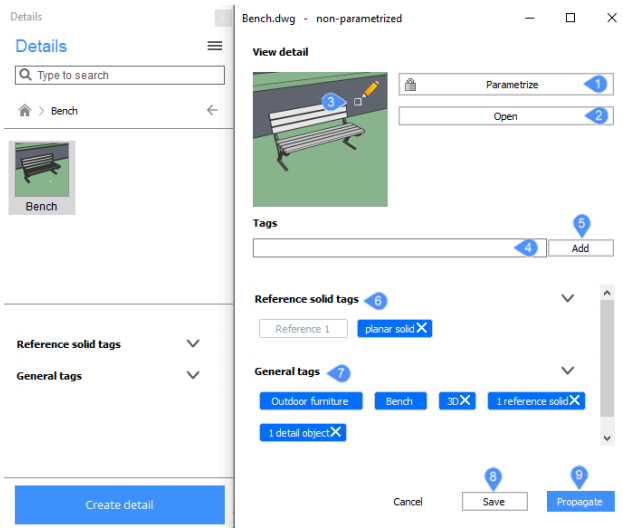
Generate thumbnails
Loads the new details in your library panel.

Manage libraries
Opens the Settings dialog box to change the details directory path.

4.11.15.2 Details library

You can click on the category of the detail you want to view. The details assigned to that category will be displayed.

When you click on a detail, the below dialog box will appear.



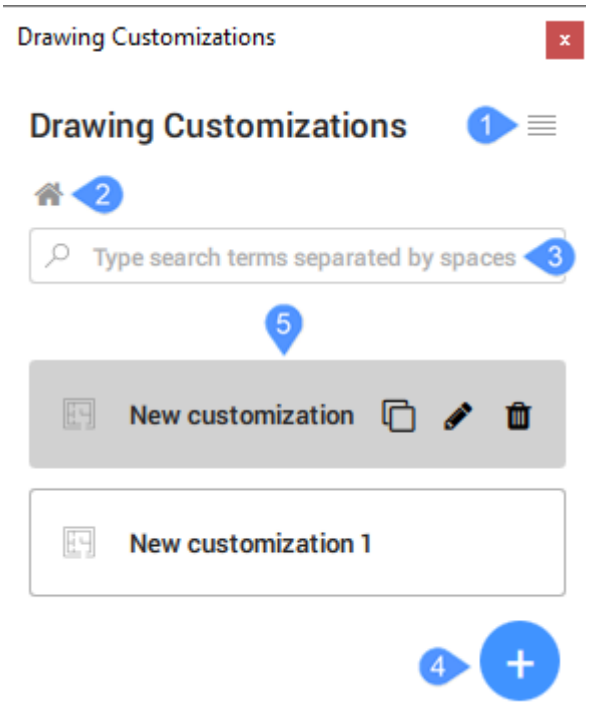
1 Parametrize
2 Open
3 Edit detail
4 Tags
5 Add tag
6 Reference solid tags
7 General tags
8 Save
9 Propagate

4.11.15.3 Create detail

Opens the Create detail dialog box.

4.11.16 Drawing customizations panel

The Drawing Customizations panel allows you to create custom templates for drawings.



1 Menu

2 Home

3 Search

4 Add customization

5 Customizations List

4.11.16.1 Menu

Refresh
Refreshes the Drawing Customizations panel.

Manage libraries
Opens the Settings dialog box to change the section settings search path.

Load customizations
Allows you to choose a folder that contains customizations.

4.11.16.2 Home

Moves the cursor to the main user interface page of the panel.

4.11.16.3 Search

Searches all the drawing customizations that matches the words you enter in the search box.

4.11.16.4 Add customization

Creates a new template.

You can create a template by clicking on the button. By default, the template is named **New Customization** with the text highlighted. After you have added a relevant name for the template, press Enter to save and apply the changes. The inserted customizations will be sorted alphabetically automatically. Use the scroll button to navigate through the list.

4.11.16.5 Customizations List

Displays all the available drawing customizations.

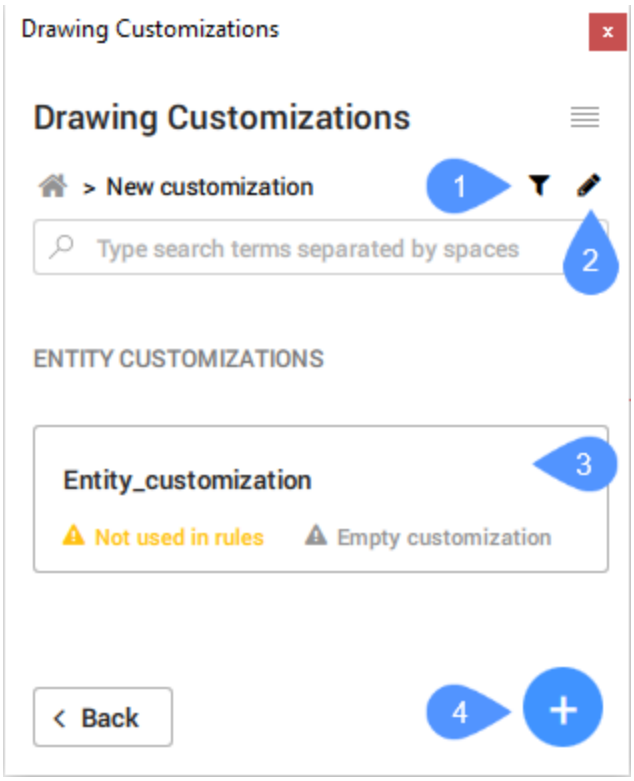
Duplicate
Creates a copy of your selected drawing customization.

Rename
Edits the name while the text is being highlighted. Press Enter to save and apply the changes.

Delete
Removes the drawing customization. The Removing a drawing customization dialog box appears.

Entity customizations
Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.
The **Entity customizations list** is empty for a newly created customization.

To create an entity customization left-click on the desired template, then click on the button. By default, the template is named **New Customization** with the text highlighted. Rename it while the text is being highlighted. Press Enter to save and apply the changes.
After creating your desired entity customization templates, click on the individual tabs to modify their respective settings.



1 To rule definitions

2 To style definitions

3 Entity customizations list

4 Add customization

To rule definitions

You can create a rule pressing the button. After creating the rule, the button offers you another 3 options:

- Entity customization
- Sub rule
- Filter

When creating an entity customization, you can add a sub rule for a rule already added.

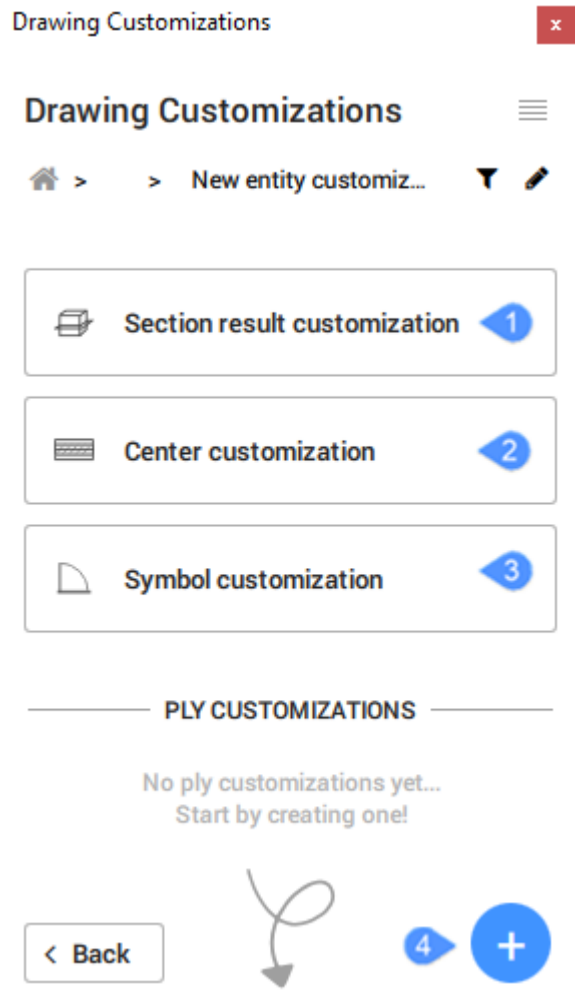
To style definitions

To add a style definition, press the button.
When selecting a color, the Select Color dialog box will appear.
When selecting a line type and pressing Load…, the Load Linetypes dialog box will appear.

Entity customizations list

Displays the list of entity customizations.

Add customization



1 Section result customization

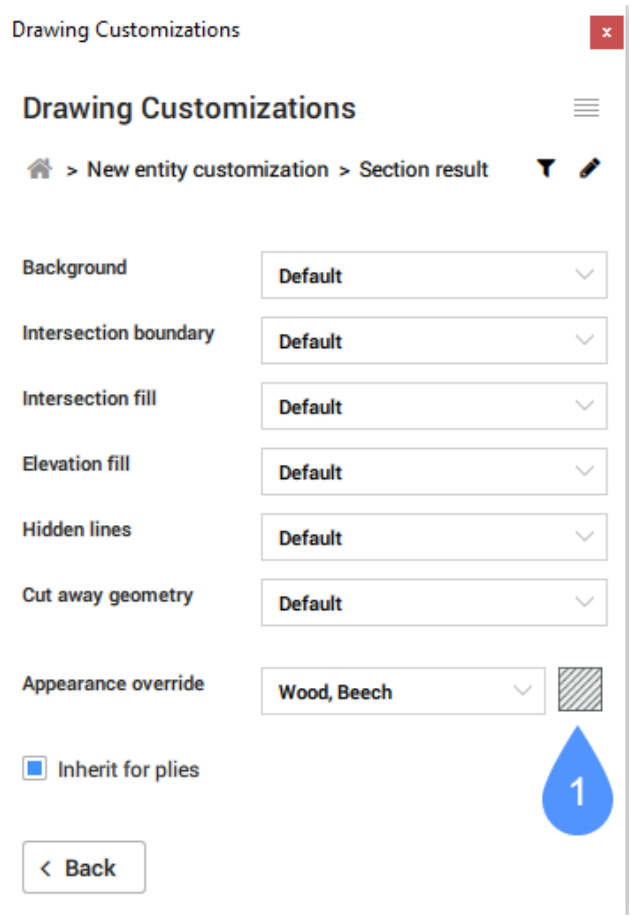
2 Center customization

3 Symbol customization

4 Add ply customization

Section result customization

Determines how the different pieces of geometry, generated by sectioning the entity with a section plane, are displayed.

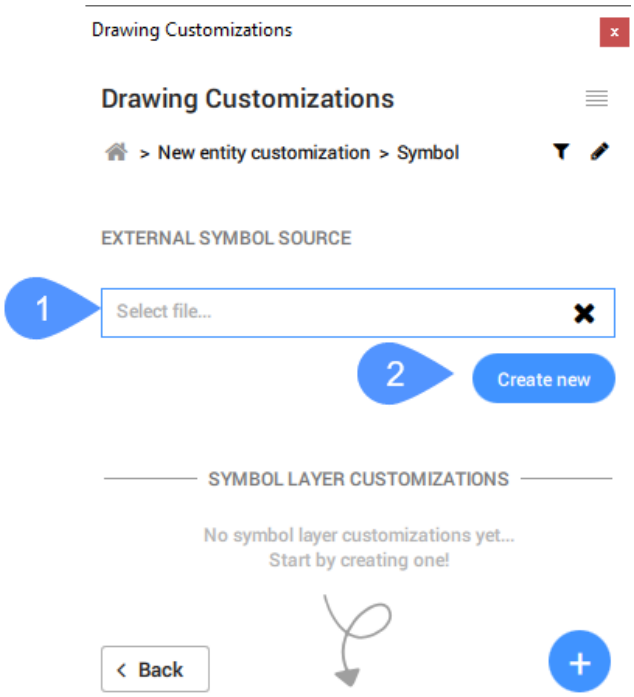


1 Physical Materials

Physical Materials
Opens the Physical Materials dialog box.

Center customization
Determines the appearance of the center of planar entities (center plane) or linear entities (center line).

Symbol customization
Determines the appearance of 2D symbols that are added for the entity and controls the source of those symbols.

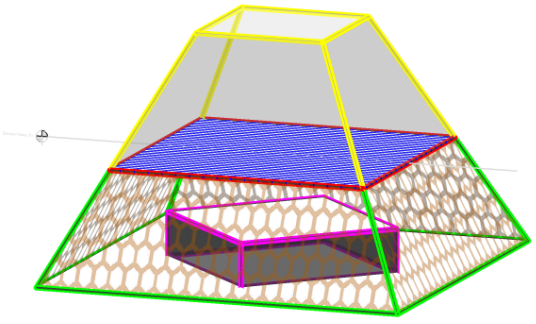


1 Select file...

2 Create new

Select file…
Opens the Set external symbol source dialog box.

Create new
Opens the New external symbol source dialog box.
The section result customization and center customization display a series of controls for which you can choose values from the drop-down list.



The diagram above illustrates how you could control the visibility and customization settings of an entity with the Drawing Customization tool. Notice that the section plane is aligned with the Blue square with Red outline.

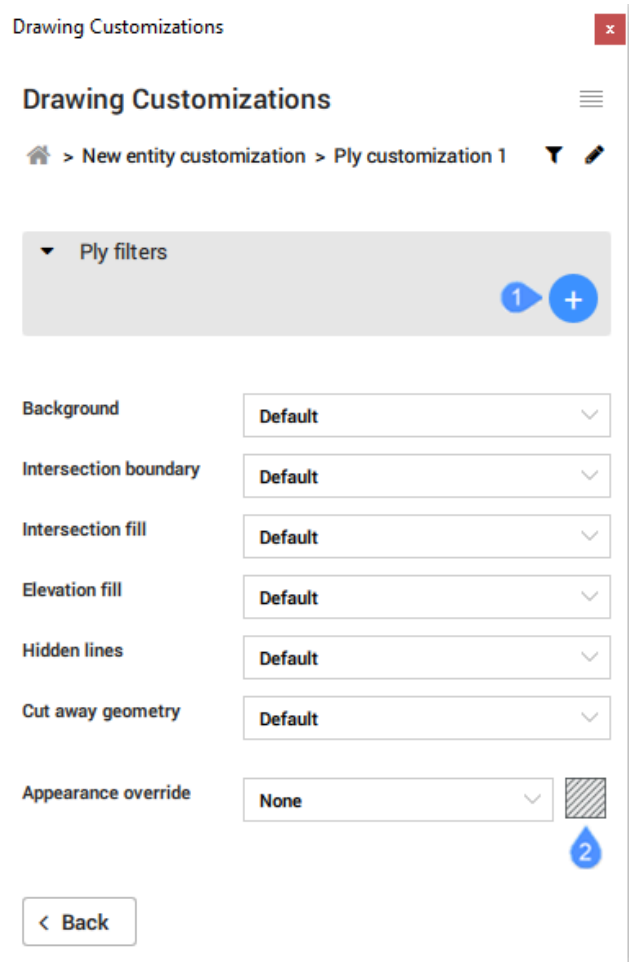
- Green outline represents**Background**, which is essentially the elevation outline of the cut object.
- Red outline represents**Intersection Boundary**, which traces over the cut area.

- Blue hatch represents **Intersection Fill**, as it shows the area of being cut by the section plane.
- Orange hexagons hatch represents **Elevation Fill**, which refers to the area bound by the elevation outline (see Background).
- Pink outline represents **Hidden Lines**, as it is below the blue hatch.
- Yellow outline represents **Cut away geometry**, as it is above the section plane.

Appearance Override allows you to specify how an entity's hatch appearances (section and elevation) would appear, in place of the default hatch patterns already defined within the Physical Materials library. When you create any new Physical Material entry in the "In Project" category as part of the Drawing Customization template, your desired configurations are independent of the drawing project and stored in the template instead. You can later apply this template to other project models as you wish.

Add ply customization

To add a ply customization, press the button.



1 Add default function

2 Physical Materials

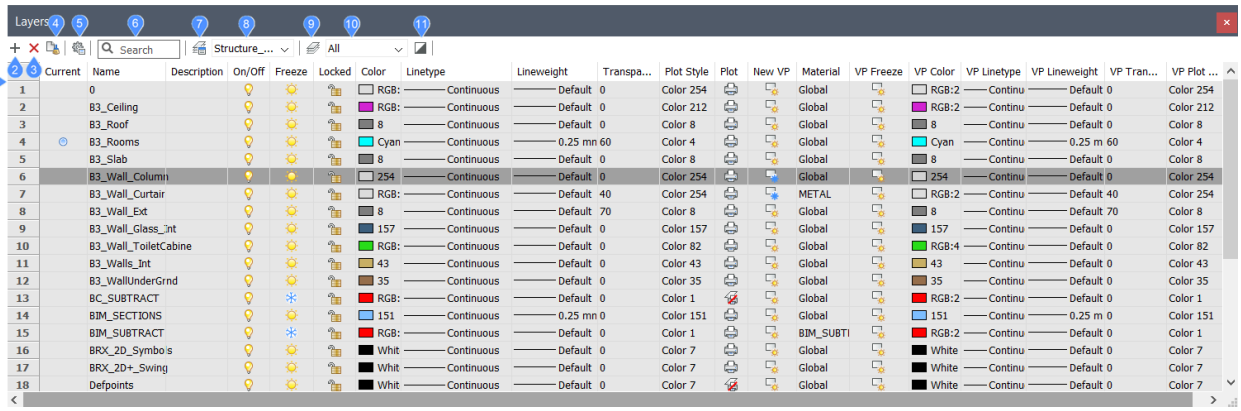
Add default function

Adds a default function for the ply customization.

Physical Materials
Opens the Physical Materials dialog box.

4.11.17 Layers panel

The Layers panel allows you to edit layer names and properties in the current drawing, as well as create, delete, edit, apply layers, layers states and layer filters.



1 Layer list

2 New layer

3 Delete

4 Purge

5 Settings

6 Search

7 Layer States Explorer

8 Layer State Control

9 Layers Explorer

10 Filter used

11 Invert filter

4.11.17.1 Layer list

Displays a list of layers and their properties. The list may include all layers in the current drawing or a subset of those layers based on selected layer states or filters. Column headers describe the properties of each layer. You may sort the layer list in order or reverse order for any of the properties by clicking once or twice on the column header.

A right-click menu on the column headers allows you to customize them. You can enable or disable the columns and restore the column positions.

A context menu displays when you right click menu on a layer name.

Merge to
Merges the selected layers to a target layer.

Nota: Merged layers will be purged from the drawing.

Remove From Group:
Removes the layer from group.

Select All:
Selects all available layers. You can also use the keyboard shortcut Ctrl+A.

Remove Viewport Overrides:
Removes overrides from properties that were set differently for viewports.

- **From Selected Layers**: Removes the overrides from the selected layers.
- **From All Layers**: Removes the overrides from all layers.
- **For Current Viewport**: Removes overrides from the current viewport.
- **For All Viewports**: Removes overrides from all viewports in the drawing

Isolate Selected Layers:
Isolates the selected layers. All other layers are either locked or turned off.

4.11.17.2 Current

Specifies the current working layer. Only one layer can be current at a time.

4.11.17.3 Name

Specifies the layer name. To rename a layer, double-click the layer name or select Rename from the right-click menu.

4.11.17.4 Description (optional)

Specifies the content or purpose of the layer. To create or edit a description, select one or more layers and then double-click the description field for one of the selected layers. The text you enter is applied to all selected layers.

4.11.17.5 On/Off

Specifies the On/Off status of the layers. Entities on layers are turned off are not visible in the drawing. However, they are processed during operations that require a regeneration of the drawing. Turning a layer on and off does not require the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for only a short time while you edit the drawing. To change the status, select one or more layers and then click a lightbulb icon for one of the selected layers. The status is applied to all selected layers.

To change the status, select one or more layers and then click a light bulb icon:

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

4.11.17.6 Freeze

Specifies the Thaw/Freeze status of the layers. Entities on frozen layers are not visible in the drawing and are not processed during operations that require a regeneration of the drawing. This can save processing time if the layer includes many entities. However, the process of freezing and thawing a layer requires the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for an extended time while you edit the drawing. To change the status, select one or more layers, and then click a sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer.
- **Freeze**: Hides entities that are on that layer.

4.11.17.7 Locked

Specifies the lock status of the layers. Entities on locked layers cannot be edited. They remain visible; however, they are dimmed based on the LAYLOCKFADECTL System Variable. To change the status, select one or more layers, and then click a padlock icon for one of the selected layers. The status is applied to all selected layers.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

4.11.17.8 Color

Specifies the color of the layers. To change the color, select one or more layers and then click the current color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

4.11.17.9 Linetype

Specifies the linetype of the layers. To change the linetype, select one or more layers and then double-click the linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

4.11.17.1 Lineweight 0

Specifies the Lineweight of the layers. To change the lineweight, select one or more layers and then double-click the lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

4.11.17.1 Transparency 1

Specifies the transparency of the layers. To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0

(fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY System Variable must be turned on to experience transparency.

4.11.17.1 Plot style

Specifies the plot style of the layers. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only. If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The Select Plot Style dialog box is displayed. The plot style you select is applied to all selected layers.

4.11.17.1 Plot

Specifies the print status of the layers. Entities on No Print layers remain visible in the drawing but are not printed. To change the status, select one or more layers, and then click the printer icon for one of the selected layers. The status is applied to all selected layers.

- **Print**: Prints the layer.
- **No Print**: Does not print the layer.

Some layers created by BricsCAD®, such as Defpoints, are set to No Print by default and cannot be changed.

4.11.17.1 New VP

Specifies the default Thaw/Freeze status of layers for new layout viewports. To change the status, select one or more layers and then click the sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer when a new viewport is created.
- **Freeze**: Hides entities that are on that layer when a new viewport is created.

4.11.17.1 Material

Specifies the material of the layers. To change the material, select one or more layers and then double-click the material for one of the selected layers. The material you select is applied to all selected layouts. If the material you want is not displayed in the drop-down list, you can add it to the current drawing using the MATERIALS command.

4.11.17.1 VP Freeze

6

Specifies the Thaw/Freeze status of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space Thaw/Freeze status. To change the status, select one or more layers and then click the sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer in the current layout or viewport.
- **Freeze**: entities that are on that layer in the current layout or viewport.

4.11.17.1 VP Color

7

Specifies the color of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space color.

To change the color, select one or more layers and then click the current vp color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

4.11.17.1 VP Linetype

8

Specifies the linetype of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space linetype.

To change the linetype, select one or more layers and then double-click the vp linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

4.11.17.1 VP Lineweight

9

Specifies the Lineweight of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space lineweight.

To change the lineweight, select one or more layers and then double-click the vp lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

4.11.17.2 VP Transparency

0

Specifies the transparency of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space transparency.

To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0 (fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

4.11.17.2 VP Plotstyle

1

Specifies the plot style of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space plot style. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only.

If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The Select Plot Style dialog box is displayed. The plot style you select is applied to all selected layers.

4.11.17.2 New layer

2

Creates new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

4.11.17.2 Delete

3

Deletes layers, except for the layer with the name **0** and any layer with entities, which cannot be deleted.

4.11.17.2 Purge

4

Purges layers from the current drawing, except for the following ones, which cannot be purged:

- The layer 0.
- The current layer.
- Any layer with entities.

4.11.17.2 Settings

5

Displays the Layer Settings dialog box.

4.11.17.2 Layer States Explorer

6

Displays the Drawing Explorer dialog box with the Layer States category selected.

4.11.17.2 Layer State Control

7

Displays the current layer state if there are any defined in the drawing. Click the down-arrow button to choose a different layer state.

4.11.17.2 Layers Explorer

8

Displays the Drawing Explorer dialog box with the Layers category selected.

4.11.17.2 Filter used

9

Displays the current layer filter if there are any defined in the drawing. Click the down-arrow button to choose a different layer filter. Some filters are automatically created by BricsCAD®: All, All Used Layers.

4.11.17.3 Invert filter

0

Inverts the content of the current filter so that the dialog box displays all layers not included in the filter.

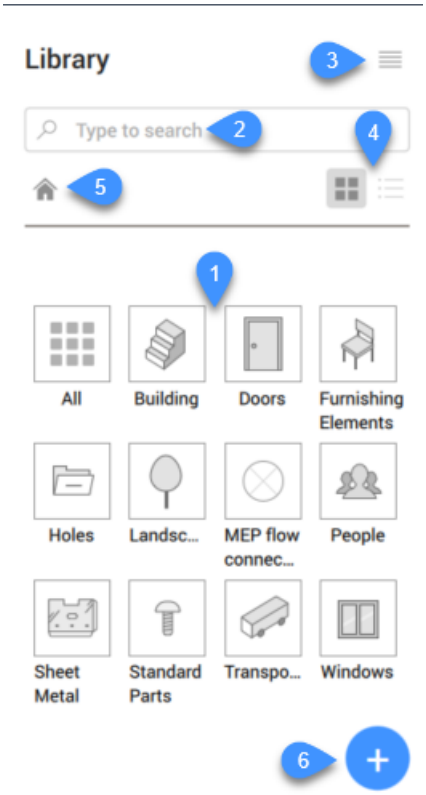
4.11.17.3 Search

1

Allows you to search for specific layer names. The list is shortened to display only the layers that contain the entered sequence of characters, regardless of where they are located in the layer name.

4.11.18 Library panel

The Library panel offers a central location to access 2D and 3D block libraries.



1 Block library

2 Search

3 Menu

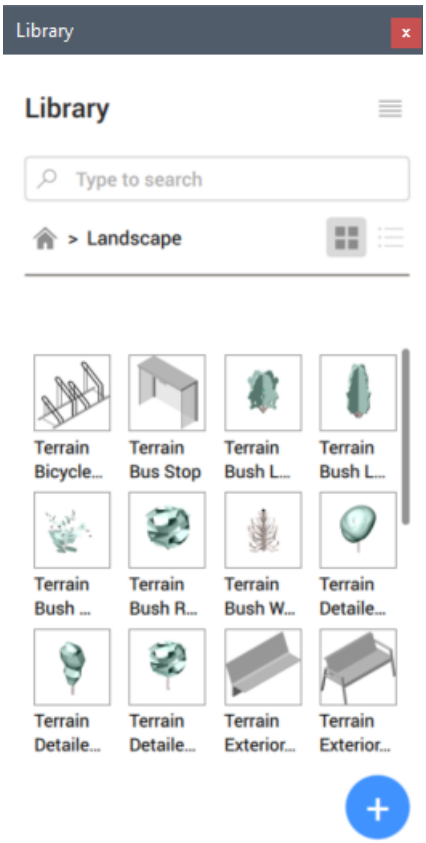
4 View

5 Home

6 Add

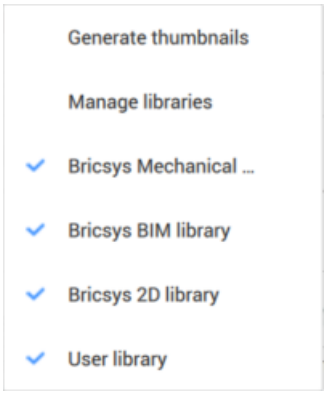
4.11.18.1 Block library

The block library is organized by categories. Select a category to view its blocks. Categories and blocks may be user-defined or pre-defined by BricsCAD®. You can scroll through the categories and blocks using the mouse wheel or the scroll bar. The availability of pre-defined blocks depends on your license level.



4.11.18.2 Menu

The Library panel menu enables you to control what is displayed in the block library.



Generate thumbnails
Generates or updates the thumbnail images of all blocks.

Manage libraries
Opens the Settings dialog box to change the library directory path.

Bricsys BIM library
When checked, the panel displays BIM components. The default path for BIM components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\Bim\Components*..

Bricsys Mechanical library
When checked, the panel displays Mechanical components. The default path for Mechanical components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\DesignLibrary*.

User library
When checked, the panel displays the user defined components. The path for user library is defined by the COMPONENTSPATH system variable.

4.11.18.3 Search

Searches the library for words you enter in the search box.

4.11.18.4 View

Toggles between grid and list views of the block content.

4.11.18.5 Home

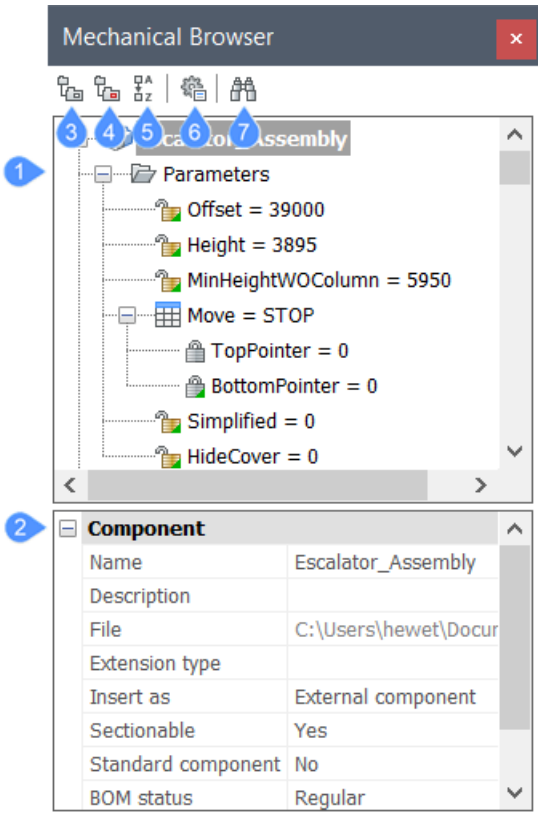
Returns the block library to the home screen.

4.11.18.6 Add

Opens the Add Block to Library dialog box.

4.11.19 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

4.11.19.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

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 to all components in the assembly.

Insert standard hardware
Inserts a standard hardware part as a mechanical component in the current drawing.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create exploded view
Creates a block with an exploded representation of the current.

Dissolve
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.

Delete
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

Disable
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.

Right-clickmenusoffer additional toolsfor components:

Open
Opens the referenced drawing.

Open a copy
Opens a copy of a component insert as a new drawing.

Update
Reloads all referenced components from external files and updates BOM tables.

Replace…
Replaces a component insert.

Nota: Replacing a local insert turns it into an external insert.

Set material to component
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

BOM status
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Link to parameter
Links subcomponent parameter to the main level parameter.

4.11.19.2 Properties

Specifies the properties of the selected item.

4.11.19.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

4.11.19.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

4.11.19.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

4.11.19.6 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

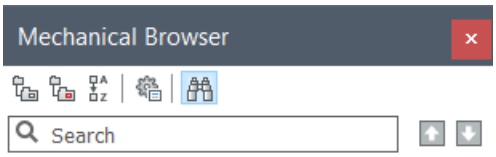
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

Sub-components of standard parts
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

4.11.19.7 Show search

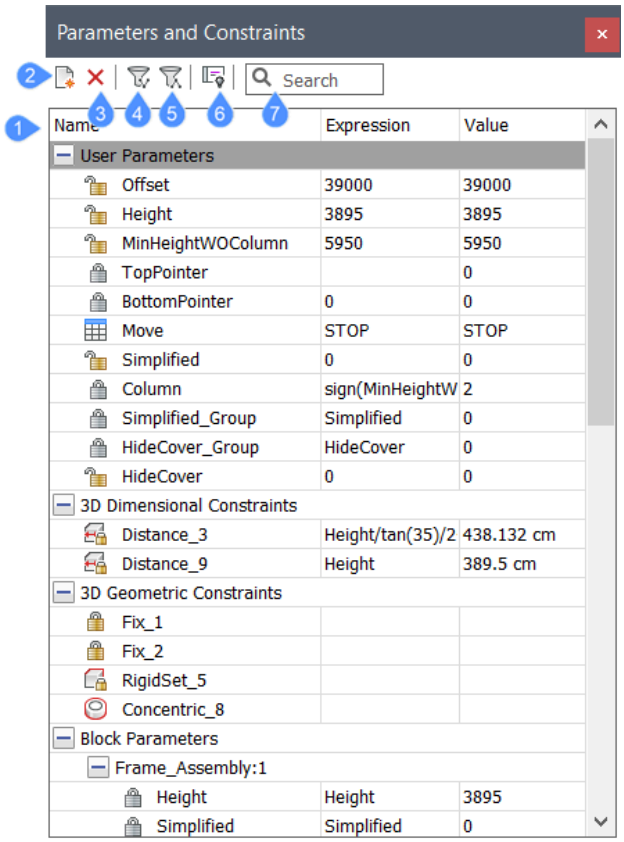
Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

Search field
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.



4.11.20 Parameters and constraints panel

The Parameters and Constraints panel offers a central location to view and modify parameters and constraints in the current drawing.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

4.11.20.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters, 2D dimensional constraints, 3D dimensional constraints,

3D geometric constraints, block parameters and array parameters. Right-click menus offer additional tools for some parameters and constraints:

Units
Sets the dimension of the parameter. You can choose between: none, linear, square and cubic.

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates parameter value within some range.

Exposed
Defines whether the parameter is visible and can be modified when the component is inserted in an assembly

Clean unused variables
Erases the unused variables.

Link to parameter
Links subcomponent parameter to the main level parameter.

4.11.20.2 Name

Specifies the name of the parameter.

4.11.20.3 Expression

Specifies the mathematical expression of the parameter.

4.11.20.4 Value

Specifies the value resulting from the parameter's expression.

4.11.20.5 New user parameter

Inserts a new user parameter with default values for the Expression and Value.

4.11.20.6 Delete a user parameter

Removes selected parameter and constraints from the drawing.

4.11.20.7 Show dependent parameters

Toggles the display of dependent parameters on or off.

4.11.20.8 Show defining parameters

Toggles the display of defining parameters on or off.

4.11.20.9 Hide geometrical constraints

Toggles the display of geometrical constraints on or off.

4.11.20.1 Search

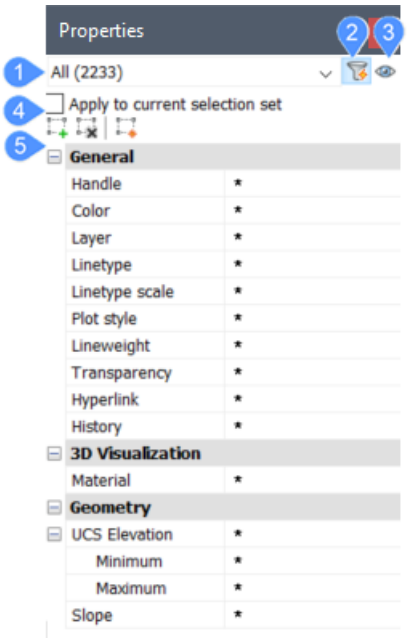
0

Filters the parameter list to display only the parameter names that match the character string you enter.

4.11.21 Properties panel

ThePropertiespanel offers a central location to access drawing, viewport, and entity properties.

It can be used in either Properties mode or Quick Select mode.



1 Entities list

2 Properties/Quick Select toggle

3 Property Preview toggle

4 Selection tools (Quick Select mode only)

5 Properties list

Entities list

Displays the type and number of selected entities.

Properties/Quick Select toggle

Switches between Properties and Quick Select mode.

Property Preview toggle

Turns the PROPERTYPREVIEW setting on or off.

Selection tools

Creates a selection set based on properties. These tools are only available in Quick Select mode.

Properties list

Displays relevant property categories based on the current mode and selection set. You can expand and collapse categories by clicking on the + and – icons.

4.11.21.1 Properties mode

In Properties mode, the Properties panel enables you to view and modify drawing, viewport, and entity properties. Drawing and viewport properties are accessible when the current selection set is empty. Entity properties are accessible when the current selection set has at least one entity.

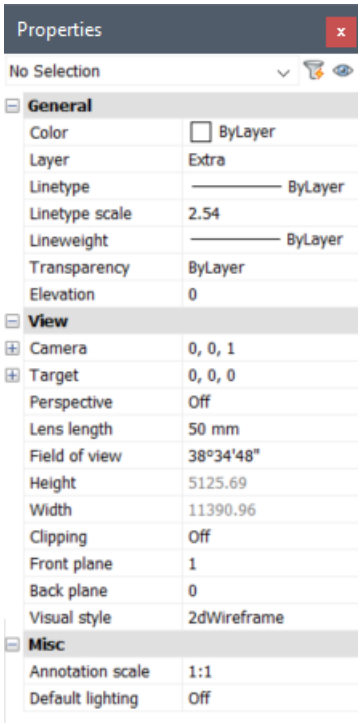
Entities list

When no entities are selected in Properties mode, the entities list shows No Selection and the properties list displays drawing and viewport properties.

When entities are selected in Properties mode, the entities list describes the type and number of entities in the selection set. You can filter the selection set by choosing a type of entity from the drop-down list. The types of entities in the selection set determine which properties are accessible.

Properties list (no selection set)

The drawing and viewport properties list, available when no entities are selected, displays General, View, and Misc properties.



General Properties

In Properties mode, when no entities are selected, General properties describe the current entity settings. View and modify current entity settings to specify the properties that apply to new entities as they are created. Changing the current entity settings in the Properties list has the same effect as changing the relevant system variables.

Color

Describes the current color (CECOLOR system variable). You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer

Describes the current layer (CLAYER system variable). You can specify a new layer from the drop-down list.

Linetype
Describes the current linetype (CELTYPE system variable). You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Describes the current linetype scale (CELTSCALE system variable). You can enter a new linetype scale.

Lineweight
Describes the current lineweight (CELWEIGHT system variable). You can specify a new lineweight from the drop-down list.

Transparency
Describes the current transparency level (CETRANSPARENCY system variable). You can specify a new transparency level from the drop-down list or enter a value.

Elevation
Describes the elevation relevant to the current UCS (ELEVATION system variable). You can enter a new elevation.

View Properties
When no entities are selected, View properties describe the current view and relevant view settings.

Camera
Describes the absolute coordinates of the camera in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Target
Describes the absolute coordinates of the camera target in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Perspective
Describes the current perspective setting (PERSPECTIVE system variable). You can turn Perspective on or off from the drop-down list.

Lens Length
Describes the current lens length (LENSLENGTH system variable). You can enter a new lens length.

Field of View
Describes the current angle for the field of view. You can enter a new angle for the field of view.

Height (read only)
Describes the height of the current view.

Width (read only)
Describes the width of the current view.

Clipping
Describes the clipping mode for the current view. You can specify a new clipping mode from the drop-down list. When clipping is turned on, parts of the view before the front clipping plane and/or behind the back clipping plane are hidden. This property is read only for layouts.

Front Plane
Describes the location of the front clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

Back Plane

Describes the location of the back clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

Visual Style

Describes the current visual style of the drawing. You can specify a new visual style from the drop-down list. This property is read only for layouts.

Misc Properties

When no entities are selected, Misc properties describe the current annotation scale and default lighting status.

Annotation Scale

Describes the current annotation scale (CANNOSCALE system variable). You can specify a new annotation scale from the drop-down list.

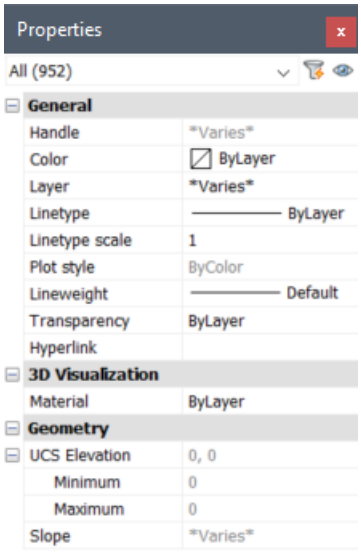
Default Lighting

Describes the default lighting status (DEFAULTLIGHTING system variable). You can turn Default Lighting on or off from the drop-down list.

Properties list (current selection set)

The Properties list only displays properties that are valid for all the selected or filtered entities. The more types of entities you select, the fewer properties are available on the Properties list. Filtering the entity list enables you to view and modify properties for multiple entities of the same type with minimal effort. When two or more entities are selected, the properties show their common values. Properties that differ between the selected entities are shown as *varies*.

All entities display General, 3D Visualization (Model space only), and Geometry properties. Additional properties may display for certain types of entities.



General Properties

In Properties mode, when entities are selected, General properties describe properties that are applied to the selected entities and are valid for almost any type of entity.

Handle (read only)

Describes the unique hexadecimal value by which the selected entity is identified in the DWG database.

Color
Describes the color of the selected entity. You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer
Describes the layer of the selected entity. You can specify a new layer from the drop-down list.

Linetype
Describes the linetype of the selected entity. You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Describes the linetype scale of the selected entity. You can enter a new linetype scale.

Plot Style
Describes the plot style of the selected entity. You can specify a new plotstle from the drop-down list. Note that this property is only available when plot styles are enabled. Also see the PlotStyle command

Lineweight
Describes the lineweight of the selected entity. You can specify a new lineweight from the drop-down list.

Transparency
Describes the transparency level of the selected entity. You can specify a new transparency level from the drop-down list or enter a value.

Hyperlink
Describes the URL of the selected entity. You can enter a new URL or choose the Browse button to access the Edit Hyperlink dialog box.

History
Enables you to undo editing operations for a selected entity. You can specify a previous step from the drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since. The History property only displays when the selection set includes only one entity.

Thickness
Describes the thickness of the selected entity. You can specify a new thickness by entering a value or selecting the picker icon and picking two points in the drawing.

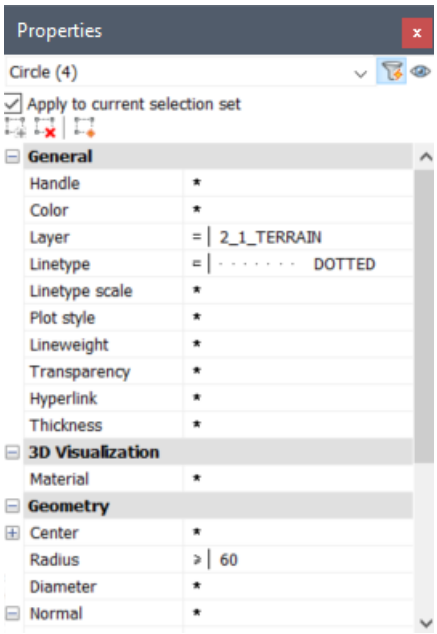
3D Visualization Properties
3D Visualization properties describe the materials assigned to the selected entities.

Material
Describes the current material of the selected entities. You can specify a new material from the drop-list.

Geometry Properties
Geometry properties describe the geometric characteristics of the selected entities. These properties can vary for different types of entities. See Drawing Entities for more information.

4.11.21.2 Quick Select mode

In Quick Select mode, the Properties panel enables you to create a selection set based on entity properties. It includes selection tools and a operators.



Selection tools

Create a selection set based on entity properties.

Apply to current selection set

This option is only valid if there's an existing selection set. When enabled, the Entities list is limited to the entities in the selection set. When disabled, the Entities list includes all entities in the current viewport or layout.

Add to current selection set

Adds entities to the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is enabled since all possible entities are already included in the current selection set.

Remove from current selection set

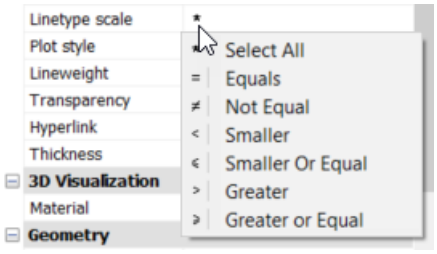
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is disabled since entities can only be removed from an existing selection set.

Add to new selection set

Creates a new selection set based on the entity types and property values you specify in the Entities and Properties lists.

Operators

A list of operators enables you to filter properties based on values. You can click on the current operator to display the list and change it to a different operator.



Select All
Does not filter the entity list based on the property.

Equals
Filters the entity list to include only those entities whose property equals the specified value.

Not Equal
Filters the entity list to include only those entities whose property does not equal the specified value.

Smaller
Filters the entity list to include only those entities whose property is less than the specified value.

Smaller Or Equal
Filters the entity list to include only those entities whose property is less than or equal to the specified value.

Greater
Filters the entity list to include only those entities whose property is greater than the specified value.

Greater or Equal
Filters the entity list to include only those entities whose property is greater than or equal to the specified value.

Properties list
The properties list displays entity properties on which you can filter. The available properties depend on the types of entities in the drawing and whether the option to Apply to current selection set is enabled. General, 3D Visualization (Model space only), and Geometry properties are available for all entities. Additional properties may display for certain types of entities.

General Properties
In Quick Select mode, General properties enable you to filter drawing entities based on their general properties.

Handle
Enables you to filter entities based on the unique hexadecimal value by which the entity is identified in the DWG database.

Color
Enables you to filter entities based on the color of the entity. You can specify a color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer
Enables you to filter entities based on the layer of the entity. You can specify a layer from the drop-down list.

Linetype
Enables you to filter entities based on the linetype of the entity. You can specify a linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Enables you to filter entities based on the linetype scale of the entity. You can enter a linetype scale.

Plot Style
Enables you to filter entities based on the plot style of the entity. You can specify a plotstle from the drop-down list.

Nota: This property is only available when plot styles are enabled.

Lineweight
Enables you to filter entities based on the lineweight of the entity. You can specify a lineweight from the drop-down list.

Transparency
Enables you to filter entities based on the transparency level of the entity. You can specify a transparency level from the drop-down list or enter a value.

Hyperlink
Enables you to filter entities based on the URL of the entity. You can enter a URL or choose the Browse button to access the Edit Hyperlink dialog box.

Thickness
Enables you to filter entities based on the thickness of the entity. You can specify a thickness by entering a value or selecting the picker icon and picking two points in the drawing.

3D Visualization Properties
In Quick Select mode, 3D Visualization properties enable you to filter drawing entities based on their 3D visualization properties.

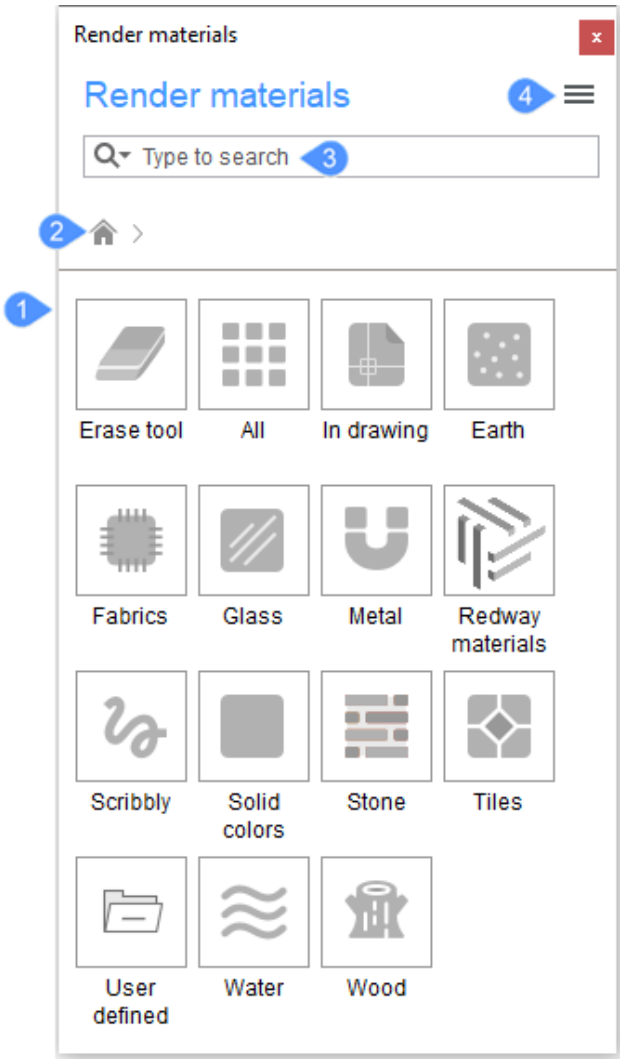
Material
Enables you to filter entities based on the material of the entity. You can specify a material from the drop-list.

Geometry Properties
In Quick Select mode, Geometry properties enable you to filter drawing entities based on their geometric properties. These properties can vary for different types of entities.

4.11.22 Render materials panel

The Render Materials panel offers access to materials used for renderings.

Material definitions are saved in the drawing. Each drawing contains the Global material. It is not possible to delete or rename the Global material, however you can edit the properties of the Global material.



1 Material library

2 Home

3 Search

4 Menu

4.11.22.1 Material library

The material library is organized by categories. You can select a material from a category and then apply it to an entity using the brush that appears. You can scroll through the materials using the mouse wheel or the scroll bar.

Nota: Right-click on a material to add it to library, to remove it from drawing or to add it to drawing.

4.11.22.2 Home

Returns the material library to the home screen.

4.11.22.3 Search

Searches the library for words you enter in the search box.

4.11.22.4 Menu

Grid View
Displays the materials in a grid view.

List View
Displays the materials in a list view.

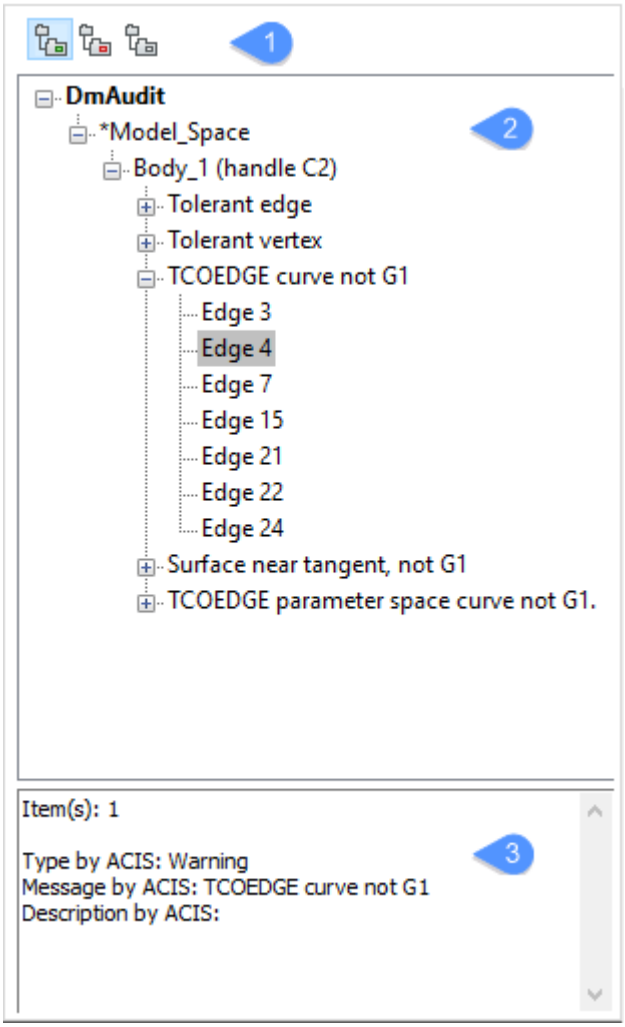
Manage libraries
Opens the Settings dialog box to change the render materials directory path.

Get more materials
Opens the Render Materials documentation in a browser, where you can download the complete Render Material Library.

4.11.23 Report panel

The Report panel offers feedback for relevant commands including DMAUDIT, DMSTITCH, and all sheet metal (SM*) commands.

The appearance of the Report panel is determined by the REPORTPANELMODE system variable.



1 Display grouping

2 Report output

3 Detailed items output

4.11.23.1 Display grouping

Specifies how information is grouped for the report.

- Sort by Structure/Message/Subentity
- Group by Message/Structure/Subentity
- Group by Structure/Subentity/Message

4.11.23.2 Report output

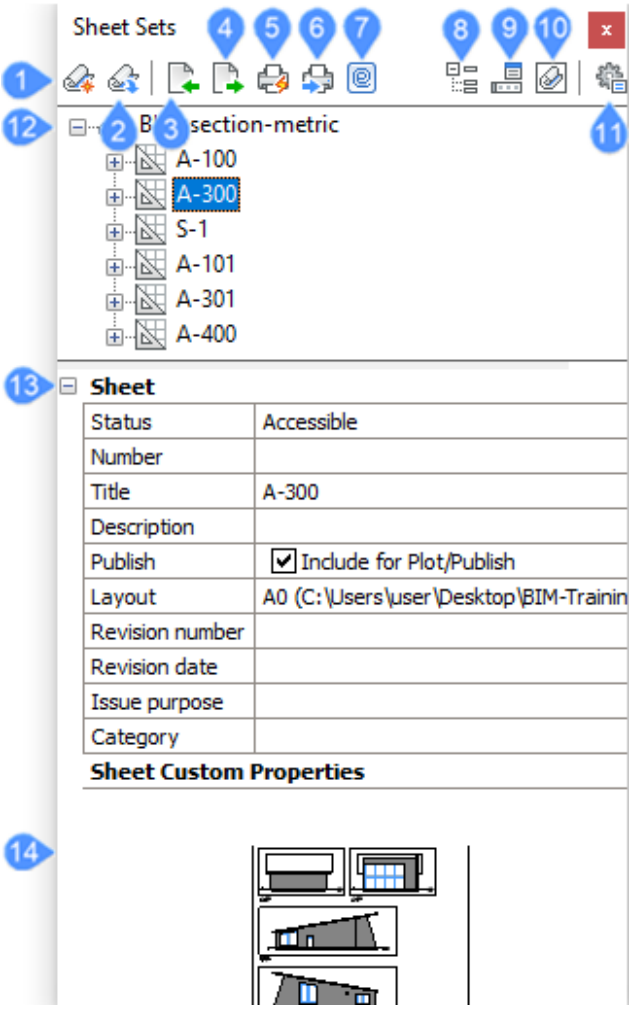
Displays command output.

4.11.23.3 Detailed items output

Shows detailed command output for selected item in the report output.

4.11.24 Sheet sets panel

The Sheet Sets panel offers a central location to create and manage sheet sets.



1 New sheet set...

2 Open sheet set...

3 Import from XML...

4 Export to XML...

5 Plot

6 Publish...

7 eTransmit...

8 Sheet selections sets...

9 Custom properties...

10 View categories

11 Sheet set options...

12 Sheets

13 Sheet properties

14 Sheet preview

4.11.24.1 New sheet set...

Opens the Create New Sheet Set dialog box.

4.11.24.2 Open sheet set...

Opens the Select a Sheet Set File dialog box.

4.11.24.3 Import from XML...

Allows you to import an XML file that was created in BricsCAD® with **Export to XML...**. Opens the Import Sheet Set dialog box.

4.11.24.4 Export to XML...

Exports a sheet set in XML format. Opens the Export sheet set as dialog box.

4.11.24.5 Plot

Plots the currently selected sheet set(s), subset(s) or sheet(s).

4.11.24.6 Publish...

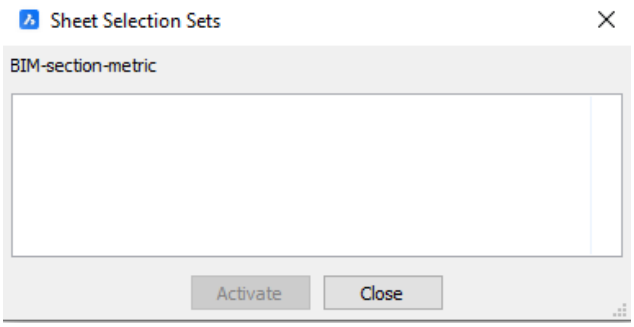
Publishes the currently selected sheet set(s), subset(s) or sheet(s). Opens the Publish dialog box.

4.11.24.7 eTransmit...

Launches the eTransmit command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure. Opens the eTransmit dialog box.

4.11.24.8 Sheet selections sets...

Opens the Sheet selection sets dialog box.



4.11.24.9 Custom properties...

Allows you to add custom properties for sheet sets and sheets. Opens the Custom properties dialog box.

4.11.24.1 View categories

0

Expands and collapses the currently loaded sheet sets.

4.11.24.1 Sheet set options...

1

Opens the Sheet Sets options in the Settings dialog box.

4.11.24.1 Sheets

2

Displays the opened and created sheet sets.

Right-clickmenusoffer additional tools:

Renumber
Opens the Renumber Sheet Set dialog box, which allows you to renumber all sheets in the sheet set.

New Sheet
Displays the New Sheet(s) dialog box, which allows to add new sheets to the current sheet set.

New Subset
Adds a new subset to the current sheet set.

Insert Sheet List Table
Inserts the sheet list table into the current drawing.

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

Nota: You can double-click on a sheet set or a subset to expand/collapse. Double-click on a sheet to open it.

4.11.24.1 Sheet properties

3

Displays the properties and variables of the selected sheet set, subset or sheet.

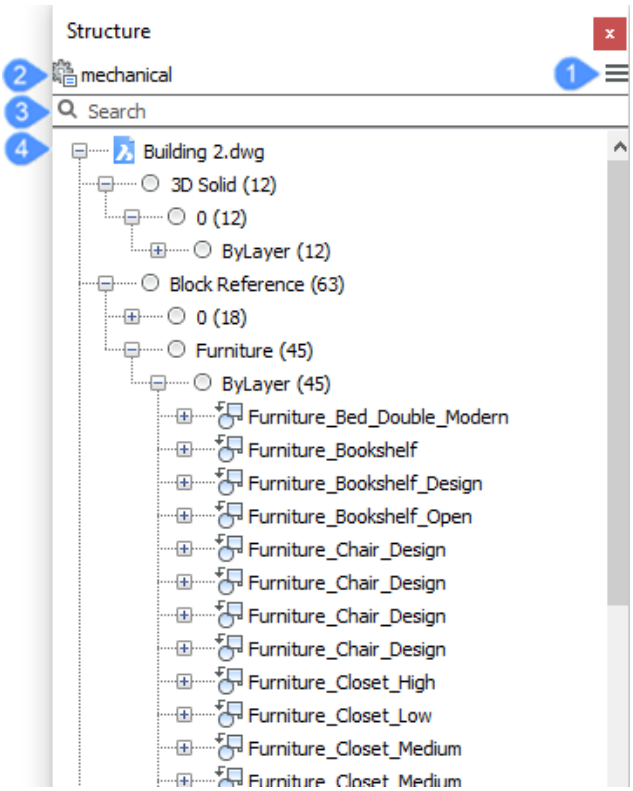
4.11.24.1 Sheet preview

4

Displays a thumbnail of the currently selected sheet set without opening it.

4.11.25 Structure panel

The Structure panel enables you to view and select drawing content from a structured tree.



1 Menu

2 Configure Structure Tree

3 Search

4 Content tree

4.11.25.1 Menu

Displays the Structure panel main menu.

bim
Displays a BIM entity-based structure.

default
Displays a layer-based structure.

mechanical
Displays an entity-based structure.

Configure structure tree
Opens the Configure Structure Tree dialog box, which allows you to configure the structure tree.

Select Configuration…
Allows you to load Structure Tree Configuration Files (*.cst).

4.11.25.2 Configure Structure Tree

Displays the Configure structure tree dialog box.

4.11.25.3 Search

Searches the structure tree for character strings that match what you enter.

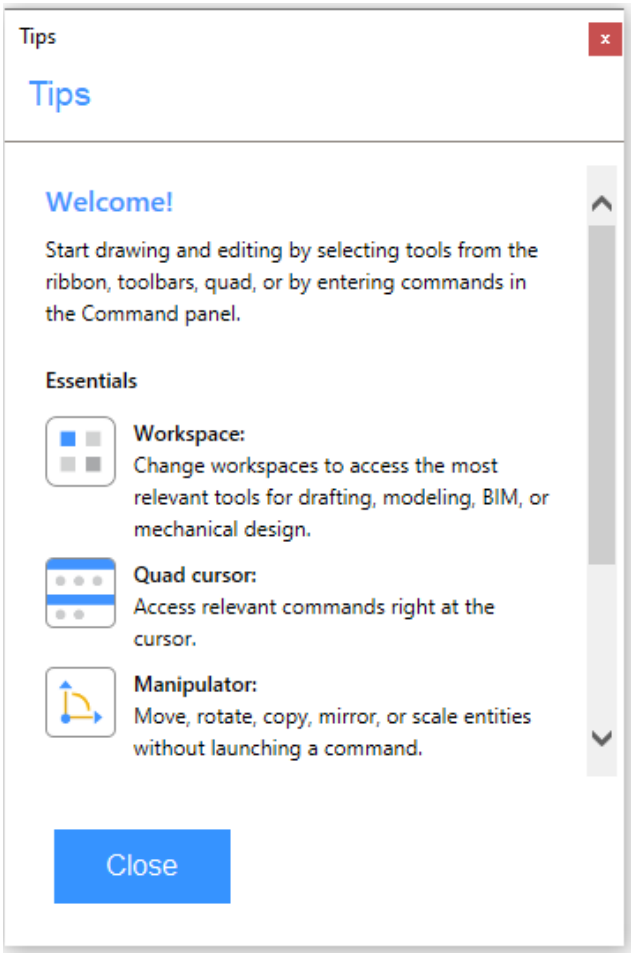
4.11.25.4 Content tree

Displays a structured tree view of the drawing content. The configuration of the structure tree is defined in a .CST file and specified by the STRUCTURETREECONFIG system variable. The location of CST files is specified by the SRCHPATH system variable. You can create custom CST files to meet your needs.

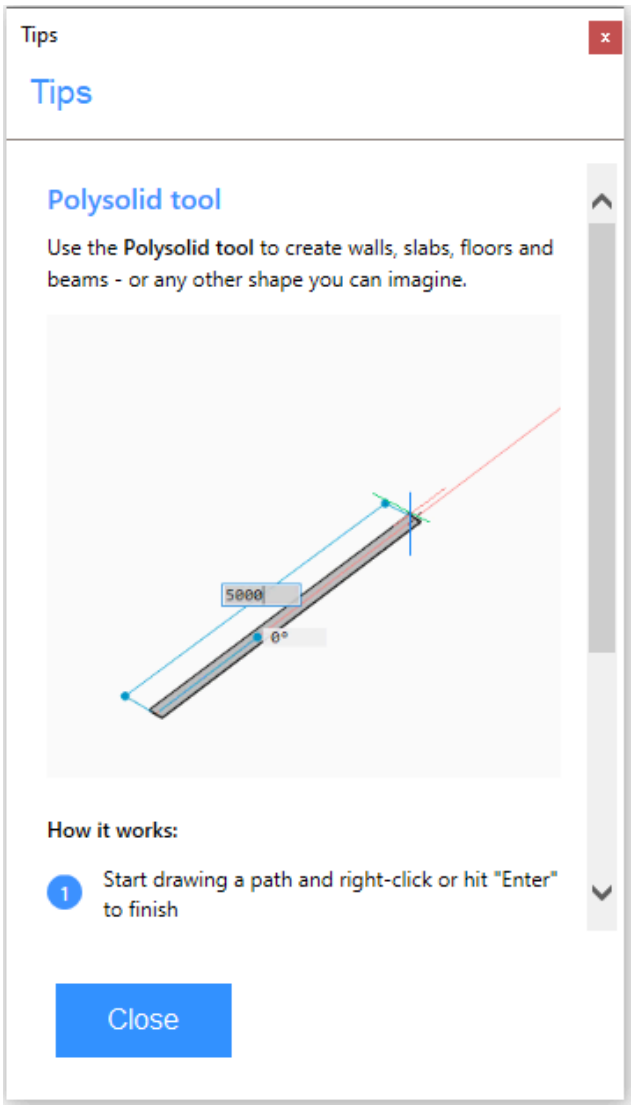
A right-click menu on a tree node or on an entity offers relevant tools.

4.11.26 Tips panel

The Tips panel offers relevant advice based on the context in which you are working.



When you start certain commands, the panel displays a short animation and additional information about the command.



4.11.27 Tool palettes panel

The Tool palettes panel offers a central location to access blocks, hatches, and command tools.

It includes default tabs with sample tools. You can easily create and customize your own tabs and tools using the right-click menu.

Right-clickmenusoffer additional tools:

Update tool image:
Updates tool image for hatches and blocks.

Specify Image…:
Allows you to specify another image for the tool.

> ***Nota:*** To restore the original tool image, right-click the tool and choose **Remove Image** in the context menu.

Properties…:
Opens the Tool Properties dialog box, which allows you to edit a tool.

View Options…:
Opens the View options dialog box, which allows you to edit the display of the tool icons.

Add Text:
Adds a text separator.

Add Separator:
Adds a line separator.

Customize Palettes…:
Opens the Customize dialog box, which allows you to customize the palettes. You can import and export palettes or create, import and export palettes groups.

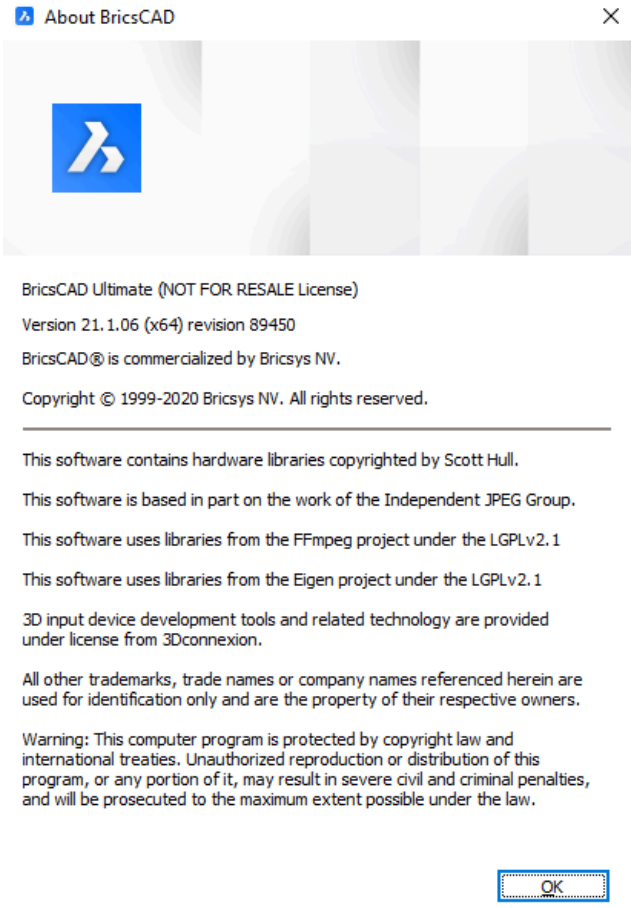
Add Tool…:
Opens the Customize dialog box, which allows you to add a new tool.

4.12 Dialog boxes

4.12.1 About BricsCAD dialog box

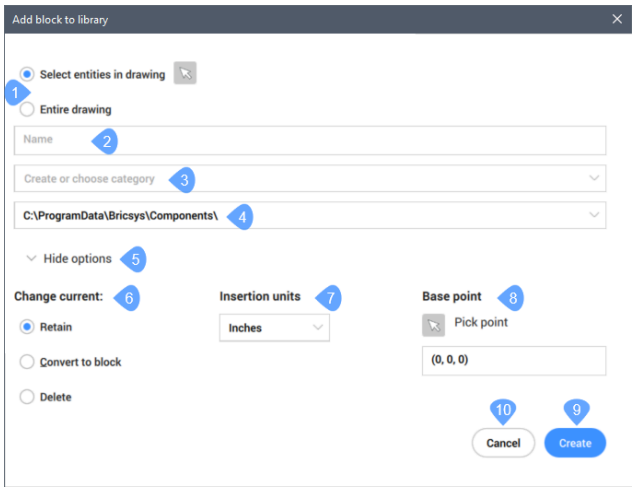
The About BricsCAD dialog box displays copyright and product information.

The product information includes the license type, version and revision number.



4.12.2 Add block to library dialog box

The Add block to library dialog box creates a block definition and adds it to the Library.



1 Selection set

2 Name

3 Category

4 Location

5 Options

6 Change current

7 Insertion units

8 Base point

9 Create

10 Cancel

4.12.2.1 Selection set

Specifies which entities are included in the block definition. You can select entities in the drawing or use the entire drawing.

4.12.2.2 Name

Specifies the name of the block.

4.12.2.3 Category

Specifies a category for the block. You can choose from a list of existing categories or enter a new category name.

4.12.2.4 Location

Specifies the location where the block dwg will be saved. Available locations are controlled by the Library directory path setting (COMPONENTSPATH system variable).

4.12.2.5 Show/Hide Options

Shows or hides additional options. The available options depend on the selection set.

4.12.2.6 Change current

Species what happens to selected entities after the block is created. You can choose to retain the individual entities, convert them to a block insertion using the new definition, or delete them. This option is only available if you define the selection set by choosing to Select entities in drawing.

4.12.2.7 Insertion units

Specifies the units for the block insertion. You can choose from the following list of standard units:

- Inches
- Feet
- Miles
- Millimeters
- Centimeters
- Meters
- Kilometers
- Microinches
- Mils
- Yards
- Angstroms
- Microns
- Decimeters
- Dekameters
- Hectometers
- Gigameters
- Astronomical Units
- Light-years
- Parsecs
- US Survey Feet
- Us Survey Inch
- US Survey Yard
- US Survey Mile
- Base point

Specifies the base point for block definition. You can pick a point in the drawing or enter X,Y,Z coordinates. This option is only available if you define the selection set by choosing to Select entities in drawing.

4.12.2.8 Create

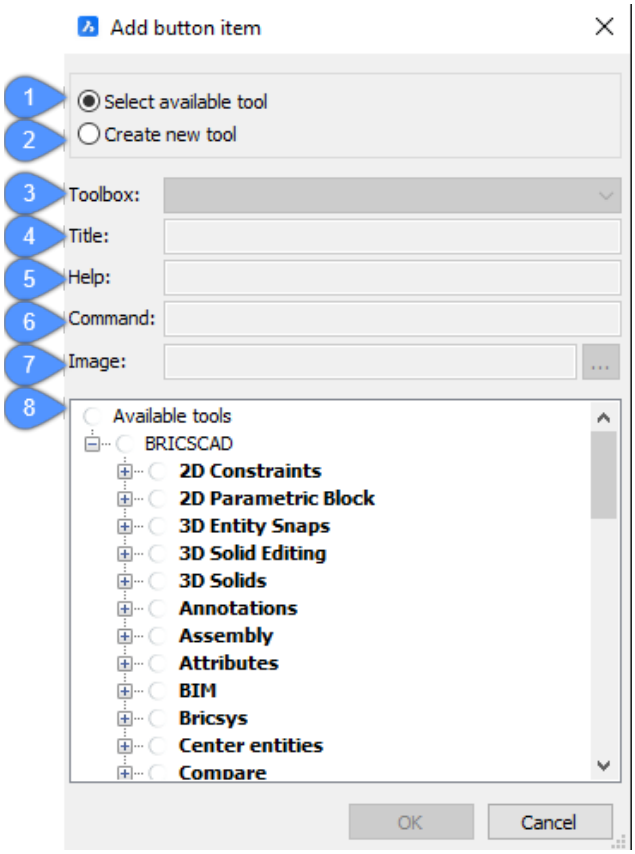
Creates a DWG file for the block definition in the specified location, adds the block to the library panel under the specified category, and closes the dialog box.

4.12.2.9 Cancel

Closes the dialog box without creating the block definition.

4.12.3 Add button item dialog box

The Add button item dialog box allows you to add a button item by selecting an available tool or by creating a new tool.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

4.12.3.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

4.12.3.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

4.12.3.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

4.12.3.4 Title

Specifies the name of the new tool.

4.12.3.5 Help

Specifies the help string displayed on the status bar.

4.12.3.6 Command

Specifies the command(s) or macros.

4.12.3.7 Image

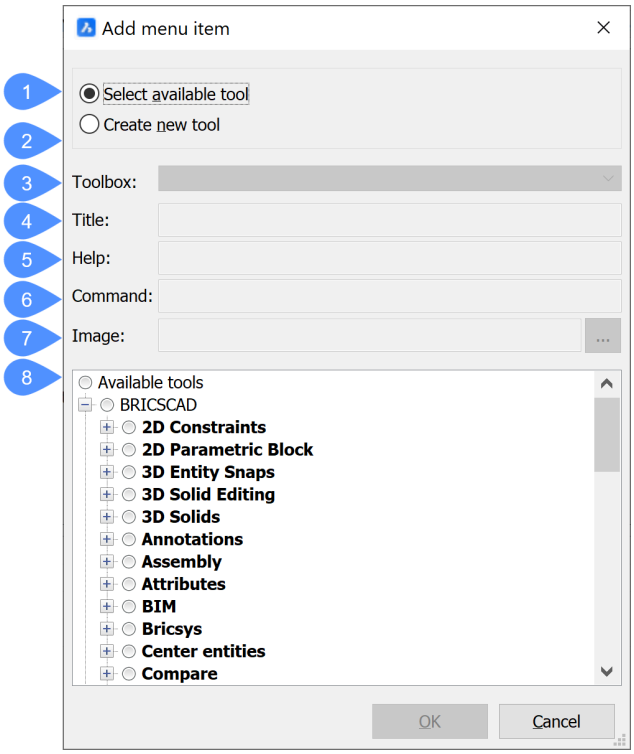
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

4.12.3.8 Available tools

Selects an existing tool.

4.12.4 Add menu item dialog box

The Add button item dialog box allows you to add a menu item by selecting an available tool or by creating a new tool.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

4.12.4.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

4.12.4.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

4.12.4.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

4.12.4.4 Title

Specifies the name of the new tool.

4.12.4.5 Help

Specifies the help string displayed on the status bar.

4.12.4.6 Command

Specifies the command(s) or macros.

4.12.4.7 Image

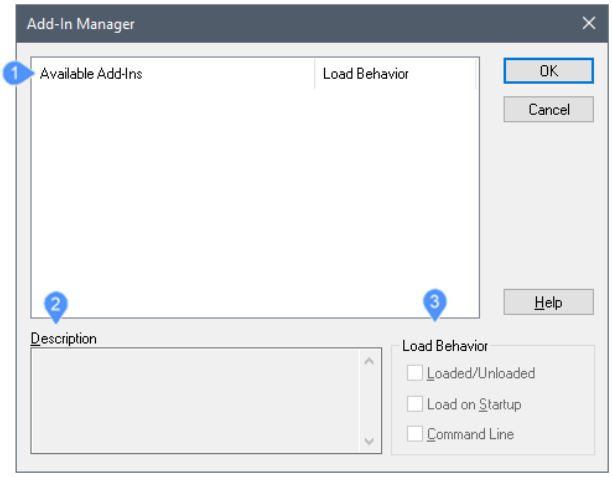
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

4.12.4.8 Available tools

Selects an existing tool.

4.12.5 Add-in manager dialog box

The Add-In Manager dialog box controls how Visual Basic for Applications COM (Common Object Model) modules are loaded. You can either toggle the modules between loaded and unloaded or start modules when BricsCAD starts up.



1 Available Add-ins

2 Description

3 Load Behavior

4.12.5.1 Available Add-ins

Lists the names and status of VBA COM modules available to BricsCAD. "VBA COM" is short for Visual Basic for Applications Common Object Model.

4.12.5.2 Description

Describes the selected VBA COM module, if any.

4.12.5.3 Load Behavior

Determines how modules are loaded.

Loaded/Unloaded
Toggles the module between loaded and unloaded.

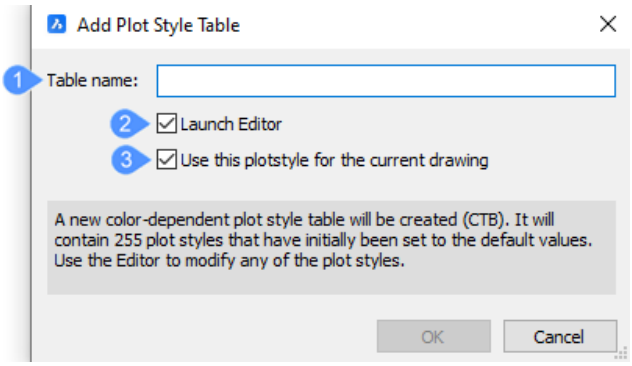
Load on Startup
Loads modules when BricsCAD starts up.

Command line
Loads modules through the Command line.

4.12.6 Add plot style table dialog box

The Add plot style table dialog box allows you to create new plot styles.

If the drawing uses named plot styles, the wizard creates a new named plot style table (STB file). If the current drawing does not use named plot styles, the wizard creates a new color-dependent table (CTB file).



1 Table name

2 Launch Editor

3 Use this Plotstyle for the current drawing

4.12.6.1 Table name

Enter the name you want to give to the plot style table.

4.12.6.2 Launch Editor

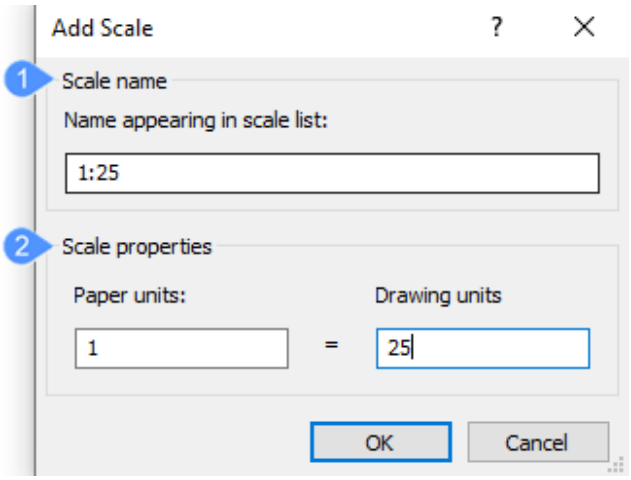
Opens the Plot style table editor dialog box after you enter the name and press the OK button.

4.12.6.3 Use this Plotstyle for the current drawing

If you do not check the Launch Editor option, it saves and sets the table name you have entered as current plot style table in the current layout.

4.12.7 Add scale dialog box

The Add Scale dialog box allows you to add a new scale factor to the list.



1 Scale name

2 Scale properties

4.12.7.1 Scale name

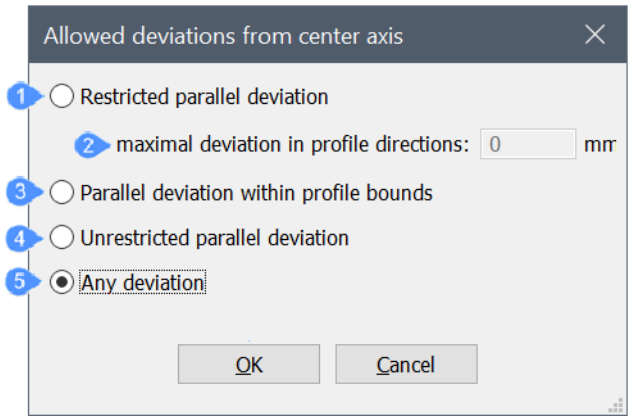
Specifies the scale name as it will appear in the scale list.

4.12.7.2 Scale properties

Specifies the scale factor in paper (size when plotted) and drawing units.

4.12.8 Allowed deviations from center axis dialog box

The Allowed Deviation from Center Axis dialog box allows you to set the level of freedom used when generating an analytical model.



1 Restricted parallel deviation

2 Maximal deviation in profile directions

3 Parallel deviation within bounds

4 Unrestricted parallel deviation

5 Any deviation

4.12.8.1 Restricted parallel deviation

Axes remain parallel to center axis and within the specified distance from the center axis in both profile directions.

4.12.8.2 Maximal deviation in profile directions

Set the maximal deviation in profile directions.

4.12.8.3 Parallel deviation within bounds

Axis remain parallel to center axis and do not exceed profile bounds.

4.12.8.4 Unrestricted parallel deviation

Axes remain parallel to center axis, but can be placed outside the profile bounds if is useful to avoid node splits or rigid links.

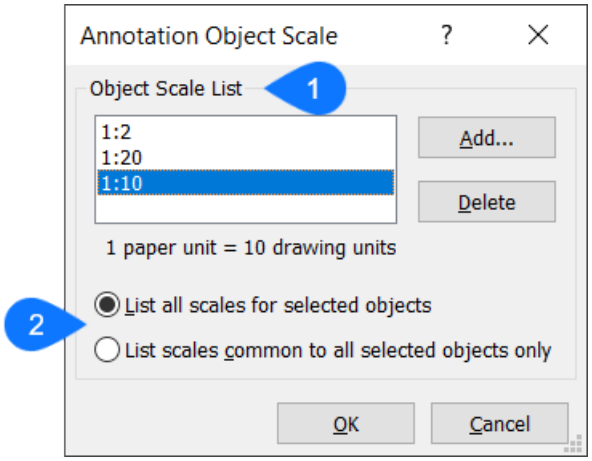
4.12.8.5 Any deviation

The main axes remain parallel to the center axis, but tilted axis (with respect to vertical and main horizontal directions) can be allowed to deviate from parallel if useful to avoid node splits and rigid links. Other axes can move outside of profile bounds.

Click **OK** to open the Visual settings dialog box.

4.12.9 Annotation Object Scale dialog box

The Annotation Object Scale dialog box allows you to add or remove scale factors used with annotative entities.



1 Object Scale List

2 Options

4.12.9.1 Object Scale List

Shows all the scales supported by the selected annotative objects.

Add

Adds annotation scale factors to the selected entities.

Delete
Removes annotation scales from the selected entities. Choose **No** in the **Annotative** field on the Properties Bar to remove the annotative property.

Nota: This option does not work if there is just one scale factor attached to the entities.

4.12.9.2 Options

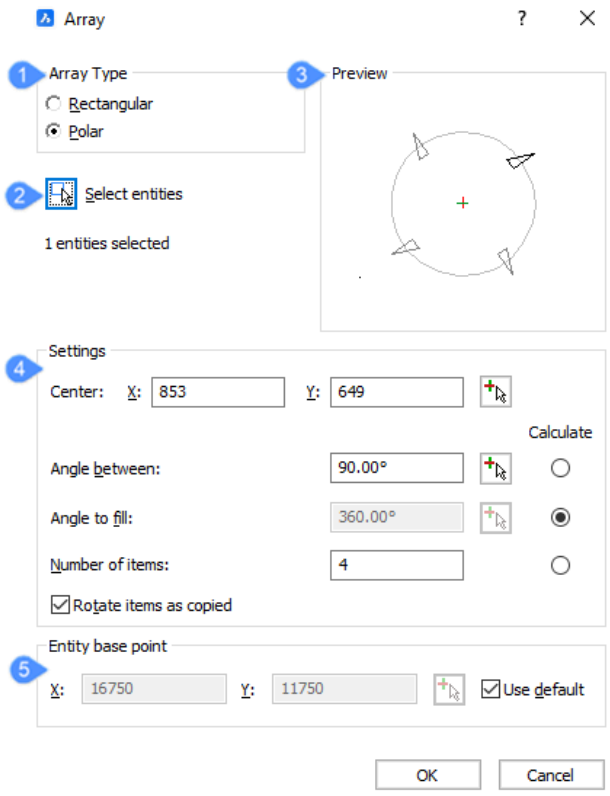
List all scales for selected objects
Lists all the scale factors.

List scales common to all selected objects only
Lists only the scale factor shared by the selected entities.

Nota: This option is meaningless when only one entity is selected.

4.12.10 Array dialog box

The Array dialog box allows you to create a 2D rectangular or polar array of entities.



1 Array Type
2 Select entities
3 Preview
4 Settings
5 Entity base point (Polar only)

4.12.10.1 Array Type

Specifies whether a rectangular or polar array will be created.

4.12.10.2 Select entities

Temporarily closes the Array dialog box and prompts you to select entities. Press Enter to complete your select and return to the Array dialog box.

4.12.10.3 Preview

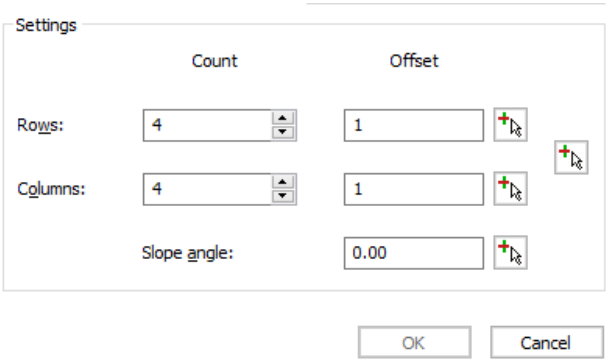
Displays a preview of the array based on the specified settings.

4.12.10.4 Settings

Specifies the number and location of items in the array. The available settings differ based on the array type.

Rectangular

Copies the selected entities in a rectangular pattern.



Count

Specifies the number of rows and columns in the array.

Offset

Specifies the distance between rows and columns. You can type a distance in the Offset fields or click the buttons to pick the offset distance in the drawing.

Offset field

Enables you to type distances for the offsets. Positive distances add rows along the positive Y axis and add columns along the positive X axis. Negative distances add rows along the negative Y axis and add columns along the negative X axis.

Offset buttons

Enables you to specify the offset distance by picking two points in the drawing. Clicking one of these buttons temporarily closes the Array dialog box and prompts you to pick points.

Pick Row Offset

Specifies the distance between rows. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if rows are added along the positive or negative Y axis.

Pick Column Offset

Specifies the distance between columns. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if columns are added along the positive or negative X axis.

Pick Both Offsets

Specifies the distance between rows and the distance between columns. A temporary rectangle displays between the two points you pick. The length of the rectangle determines the distance between columns and the width of rectangle determines the distance between rows.

Slope Angle

Specifies the angle of the array from the X axis. You can type an angle in the Angle field or click the button to pick an angle in the drawing.

Angle field

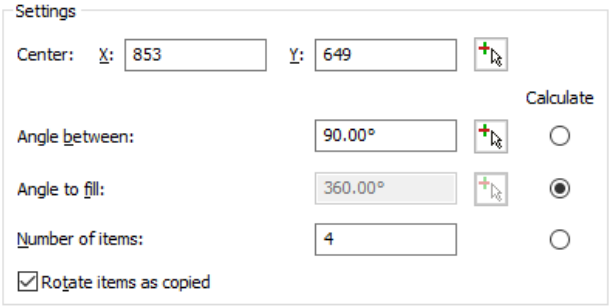
Enables you to type an angle for the slope.

Pick Angle of Array button

Enables you to specify the angle of the array by picking two points in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick points.

Polar

Copies the selected entities in a circular pattern.



Center

Specifies the center point of the polar array. You can type coordinates in the X and Y fields or click the button to pick the center in the drawing.

X field

Enables you to type a coordinate to place the center of the array along the X axis.

Y field

Enables you to type a coordinate to place the center of the array along the Y axis.

Pick Center Point button

Enables you to specify the center of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

Angle between

Specifies the angle between each item in the polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field

Enables you to type a value to specify the angle between each item of the array.

Pick Angle Between Items button
Enables you to specify the angle between each time by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the angle between items.

Calculate
Enables you to specify the angle between each item based on the Angle to fill and number of items. Choosing the Calculate radio button disables the Angle between options and enables the Angle to fill options.

Angle to fill
Specifies the fill angle and direction for a polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field
Enables you to type a value to specify the fill angle of the array. Enter 360 for a full circle of copies. Enter a smaller number to draw a partial polar array. Positive values draw the array counterclockwise and negative values draw the array clockwise.

Pick Angle to Fill button
Enables you to specify the fill angle of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the fill angle.

Calculate
Enables you to specify the fill angle based on the angle between items and the number of items. Choosing the Calculate radio button disables the Angle to fill options and enables the Angle between options.

Number of items
Specifies the number of items in the polar array. You can type a number in the field or choose the Calculate option.

Number field
Enables you to enter the number of items to include in the array.

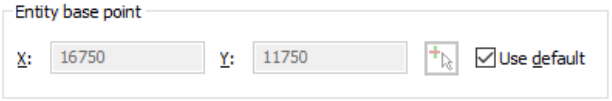
Calculate
Enables you to specify the number of items based on the angle between items and the fill angle. Choosing the Calculate radio button disables the Number of items options and enables the Angle Between and Angle to Fill options.

Rotate items as copied
Specifies whether array items are rotated when they are copied to form the polar array.

- No: does not rotate items; all copies have the same orientation as the original item.
- Yes: rotates copies.

4.12.10.5 Entity base point (Polar only)

Specifies the base point of items in a polar array.



X field
Enables you to type a coordinate to specify the basepoint of array items along the X axis.

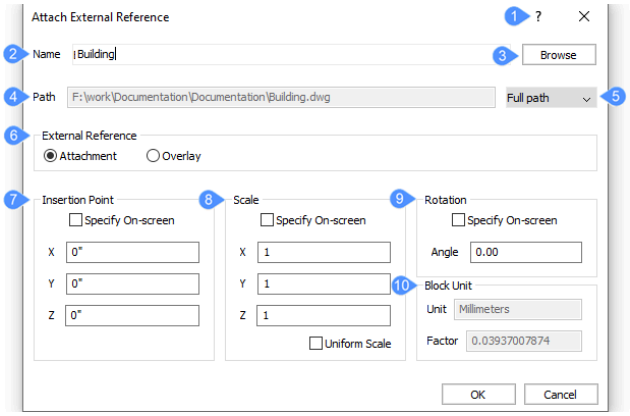
Y field
Enables you to type a coordinate to specify the basepoint of array items along the Y axis.

Pick base point button
Enables you to specify the base point of array items by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

Use Default
Uses the default basepoint for array entities. Choosing the Use default option disables the other entity base point options.

4.12.11 Attach external reference

The Attach External Reference dialog box allows you to attach externally - referenced drawings to the current drawing.



1 Help
2 Name
3 Browse
4 Path
5 Path type
6 External reference
7 Insertion Point
8 Scale
9 Rotation
10 Block Unit

4.12.11.1 Help

Opens the Bricsys Help article about the XATTACH command.

4.12.11.2 Name

Specifies the name of the DWG file attached to the current drawing.

4.12.11.3 Browse

Opens the Select Reference File dialog box to allow you to choose a different *.dwg file

4.12.11.4 Path

Displays the path of the drawing file.

4.12.11.5 Path type

Determines how much of the path is stored:

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 path from the parent drawing's location to the referenced drawing's location, such as *.. \filename.dwg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the DWG file name, such as *filename.dwg*.

4.12.11.6 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.

4.12.11.7 Insertion Point

Specifies the location of the xref's lower-left corner:

Specify On-screen
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y, and Z fields.

X, Y, or Z
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.

4.12.11.8 Scale

Specifies the size of the xref.

Specify On-screen
Determines how you specify the scale factors.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X, Y, and Z fields.

X, Y, or Z
Specifies the x, y, and/or z scale 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 x.

4.12.11.9 Rotation

Specifies the rotation angle of the xref.

Specify On-screen
Determines how you specify the rotation angle.

- On: specify the angle in the drawing after the dialog box is closed.
- 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. Negative angles rotate the xref clockwise. Use 0 to keep the xref at its original orientation.

4.12.11.1 Block Unit

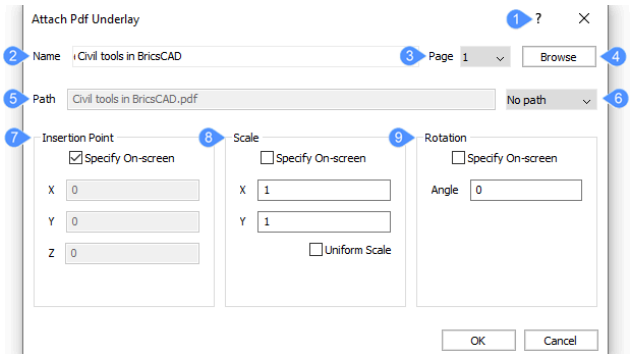
0
Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm:

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.

4.12.12 Attach PDF underlay dialog box

The Attach PDF underlay dialog box allows you to attach a PDF file as an underlay into the current drawing.



1 Help

2 Name

3 Page

4 Browse

5 Path

6 Path type

7 Insertion Point

8 Scale

9 Rotation

4.12.12.1 Help

Opens the Bricsys Help article about the -PDFATTACH command.

4.12.12.2 Name

Displays the file name of the PDF file to attach. When more than one PDF file has been attached to the drawing, their names are listed in the drop-down list.

4.12.12.3 Page

Allows you to display the page you want from the whole PDF file. You can change the number of the page from the drop-down arrow list.

4.12.12.4 Browse

Opens the Select PDF underlay file dialog box from where you can select another PDF file than the one already selected.

4.12.12.5 Path

Displays the path of the file according to the path type you choose.

4.12.12.6 Path type

Choose the way the program remembers the path to the PDF file. You can select from the Path drop-down list one of the following options:

Full path
Stores the drive and the folder(s) name to the PDF file as an absolute reference such as: C:\Users\user \PDFATTACH\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.

4.12.12.7 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.
- 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.

4.12.12.8 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.
- Off: specify the scale factor in the dialog box using the X and Y fields.

X,Y
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

4.12.12.9 Rotation

Specifies the rotation angle of the PDF underlay:

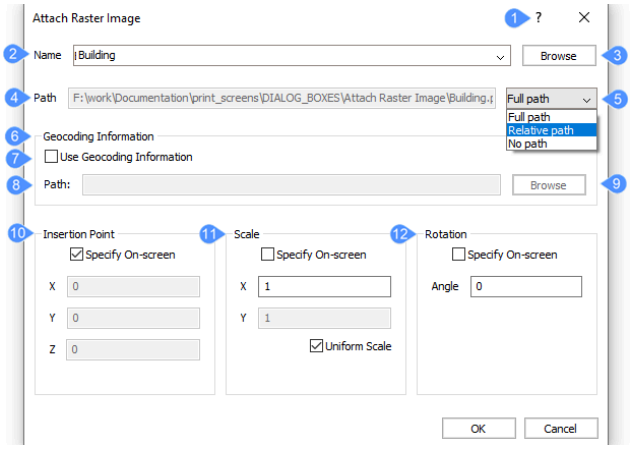
Specify On-screen
Determines how the rotation angle is specified:

- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the rotation angle in the dialog in Angle field.

Angle
Specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the underlay at its original orientation.

4.12.13 Attach raster image dialog box

The Attach image raster dialog box allows you to attach raster images to the drawing.



1 Help

2 Name

3 Browse

4 Path

5 Path type

6 Geocoding Information

7 Use Geocoding Information

8 Path

9 Browse (image position file)

10 Insertion Point

11 Scale

12 Rotation

4.12.13.1 Help

Opens the Bricsys Help article about the IMAGEATTACH command.

4.12.13.2 Name

Specifies the name of the file to attach.

4.12.13.3 Browse

Opens the Select Image File dialog box to allow you to choose a different raster file.

4.12.13.4 Path

Displays the path of the image file.

4.12.13.5 Path type

Specifies how much of the path is stored.

Full path
Stores the full path to the image file as an absolute reference such as: *C:\foldername\imagename.jpg*

Relative path
Stores the path from the drawing's location to the image's location. For example: *..\foldername \imagename.jpg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the image file name such as *imagename.jpg*.

4.12.13.6 Geocoding Information

Position files or world files have an extension that depends on what kind of image file they are associated with.

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
Displays the path of the position file.

Browse (image position file)
Opens the Select Image Position File dialog box.

4.12.13.7 Insertion Point

Specifies the location of the image's lower left corner.

Specify On-screen
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y and Z fields.

X,Y,Z
Specifies the x, y, and z coordinates for the image's insertion point.

4.12.13.8 Scale

Specifies the size of the image.

Specify On-screen
Determines how you specify the scale factor.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X and Y fields.

XY
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

4.12.13.9 Rotation

Specifies the rotation angle of the image.

Specify On-screen
Determines how you specify the rotation angle.

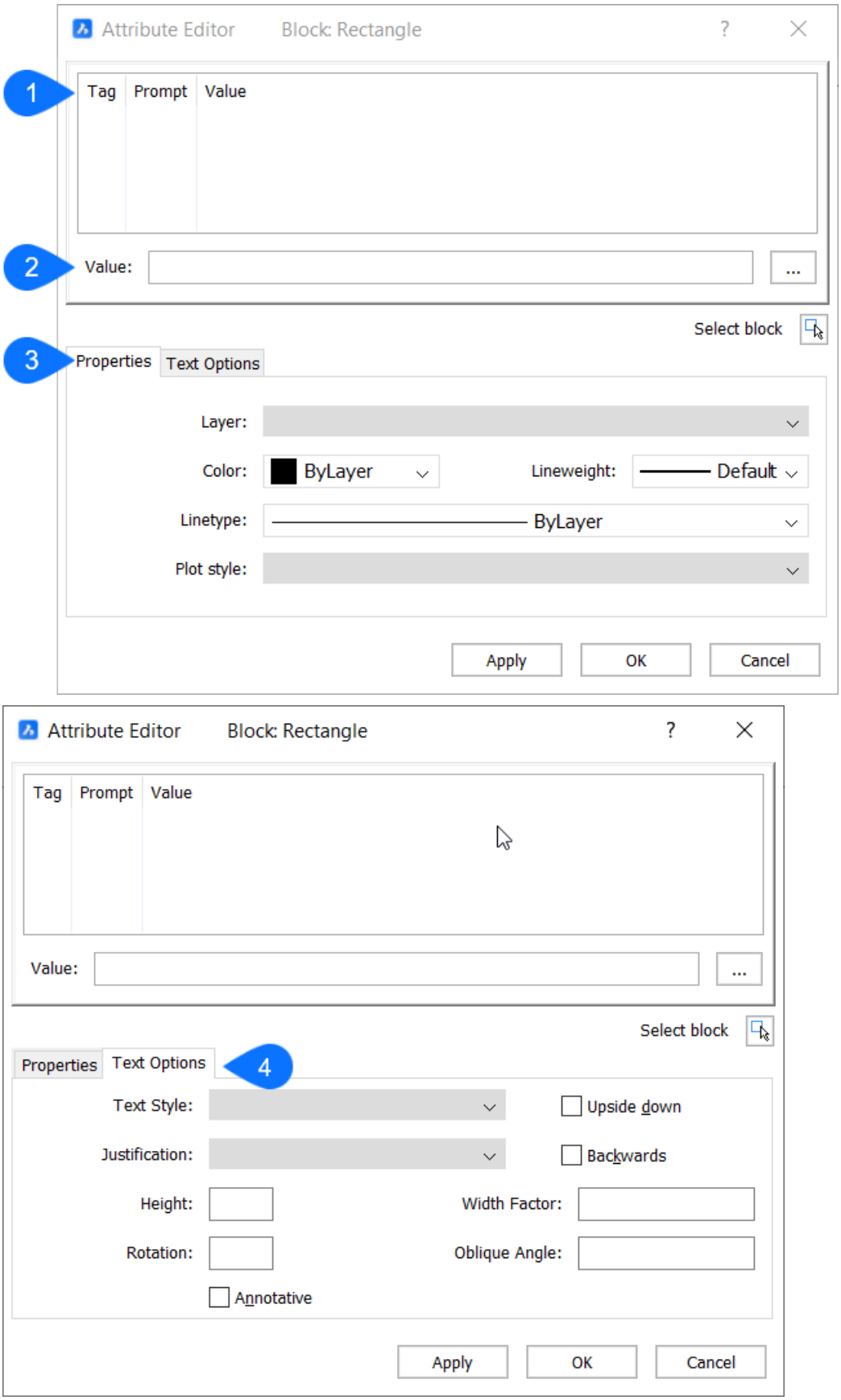
- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

4.12.13.1 Angle

0
Specifies the rotation angle of the image about the insertion point. Positive angles rotate the image counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the image at its original orientation.

4.12.14 Attribute editor dialog box

The attribute Editor dialog box allows you to edit attributes within blocks.



1 Overview list

2 Value

3 Properties

4 Text Options

4.12.14.1 Overview list

Displays an overview of all attributes used in the block.

4.12.14.2 Value

Displays the currently selected attribute from the list. A new value can be typed.

4.12.14.3 Properties

Specifies properties of the selected attribute.

Layer
Specifies the layer of the selected attribute.

Color
Specifies the color of the selected attribute.

Linetype
Specifies the linetype of the selected attribute.

Plot style
Specifies the plotstyle of the selected attribute.

4.12.14.4 Text Options

Text style
Specifies the font of the text in the selected attribute.

Justification
Specifies the alignment of the text in the selected attribute.

Upside down
When checked, text is displayed upside down.

Backwards
When checked, text is displayed backwards.

Height
Specifies the height of the text in the selected attribute.

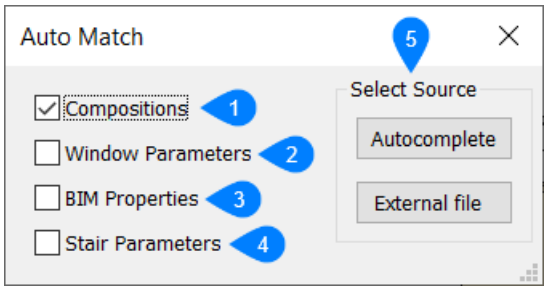
Rotation
Specifies the rotation angle of the text in the selected attribute.

Width factor
Specifies the width of the text in the selected attribute.

Oblique angle
Specifies the oblique angle of the text in the selected attribute.

4.12.15 Auto Match dialog box

The Auto Match dialog box defines which characteristics of example entities are matched and lets you select a source.



1 Compositions

2 Window Parameters

3 BIM Properties

4 Stair Parameters

5 Select Source

4.12.15.1 Compositions

When checked on, the BIM properties of entities will be copied.

Entities without a composition will be matched with the other compositions in your drawing, based on type, spatial location and orientation of the entity, as well as on the 'Is external'-property.

If all entities have compositions, you can overwrite some by choosing a source entity. AutoMatch will then copy this source's composition to the rest of the valid entities.

4.12.15.2 Window Parameters

When checked on, the parameters of the windows will be copied.

If there are unchanged windows in your drawing, AutoMatch will match the properties based on width, height, type, spatial location and orientation of the windows and type and composition of the entities they are in.

If all windows have changed parameters, you can overwrite some by choosing a source window. AutoMatch will then copy this source's parameters to the rest of the valid windows.

Nota: Some window parameters will never be copied along.

- W (= width of the window)
- H (= height of the window)

4.12.15.3 BIM Properties

When checked on, the BIM properties of entities will be copied.

If there are entities with unchanged properties in your drawing, AutoMatch will match the properties based on BIM type, composition and spatial location of the other entities in your drawing.

If all entities have a changed property, you can overwrite it by choosing a source entity. AutoMatch will then copy this source's property to the rest of the valid entities.

Some general BIM properties will never be copied along:

- GUID

- Composition
- Profile
- SpatialLocation
- IsExternal

Some beam and column related BIM properties will never be copied along:

- eomHash
- athString
- XAxis
- YAxis
- ZAxis
- ProfileName
- EndPoint
- StartPoint
- Valid
- InstancePropertySets

Some window and door related BIM properties will never be copied along:

- Name
- SubtractAllSolids
- Translucency
- OverallWidth
- OverallHeight

Some slab related BIM properties will never be copied along:

- RoomBounding

Some room related BIM properties will never be copied along:

- Number
- RoomArea
- RoomRepresentation

Some stair related BIM properties will never be copied along:

- StairType
- NumberOfRisers
- NumberOfTreads

4.12.15.4 Stair Parameters

When checked on, the parameters of a stair will be copied.

If there are unchanged stairs in your drawing, AutoMatch will match the properties based on number of treads, number of risers, height, width, length and type of the stair.

If all stairs have changed parameters, you can overwrite some by choosing a source stair. AutoMatch will then copy this source's parameters to the rest of the valid stairs.

Nota: Some stair parameters will never be copied along:

- Height_Staircase
- Stair_Width
- Length_Staircase

4.12.15.5 Select Source

Autocomplete
Matches all chosen options for entities in the open drawing with similar entities in the open drawing.

External file
Opens the Select file to insert dialog box. Here you can choose a file to match all chosen options for entities in the chosen drawing with similar entities in the open drawing.

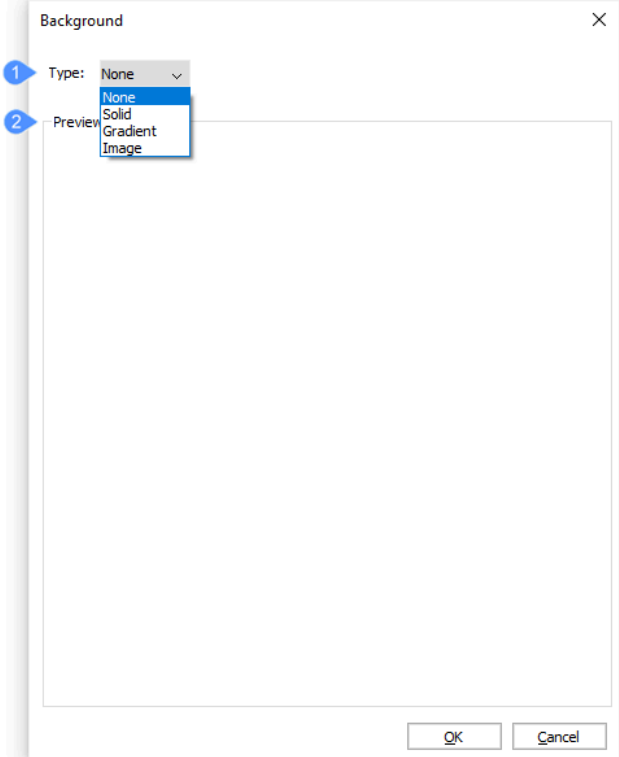
4.12.16 Background dialog box

The Background dialog box enables you to apply a background to the current viewport. You can choose from 4 types: None, Solid, Gradient, and Image.

Nota: Set Backgrounds = On in Visual Styles in Drawing Explorer dialog box to apply the changes you may make in Background dialog box. This option is available in all visual styles, except 2DWIREFRAME.

4.12.16.1 None

When background type is set to None, the Background dialog box includes 2 areas:



1 Type

2 Preview

Type
Specifies the background type for the current viewport. When the background type is None, default colors are applied to the current viewport.

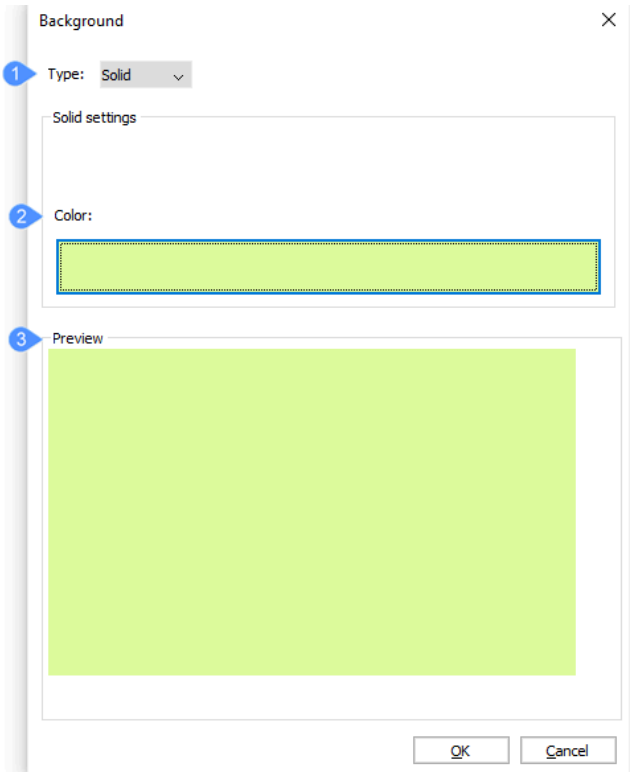
Preview
Displays a preview of the background. When set to None, the background depends on the PERSPECTIVE variable:

- **Off**: background is the color specified by the BKGCOLOR variable, most often black or white.
- **On**: two-color gradient that simulates the ground with the sky; the colors are specified by variables that begin with Gradient-, such as GRADIENTCOLORTOP, GRADIENTCOLORMIDDLE and GRADIENTCOLORBOTTOM.

Nota: After changing the variables' values, rerun the GRADIENTBKGON command.

4.12.16.2 Solid

When the background type is set to Solid, the Background dialog box includes 3 areas:



1 Type

2 Solid settings

3 Preview

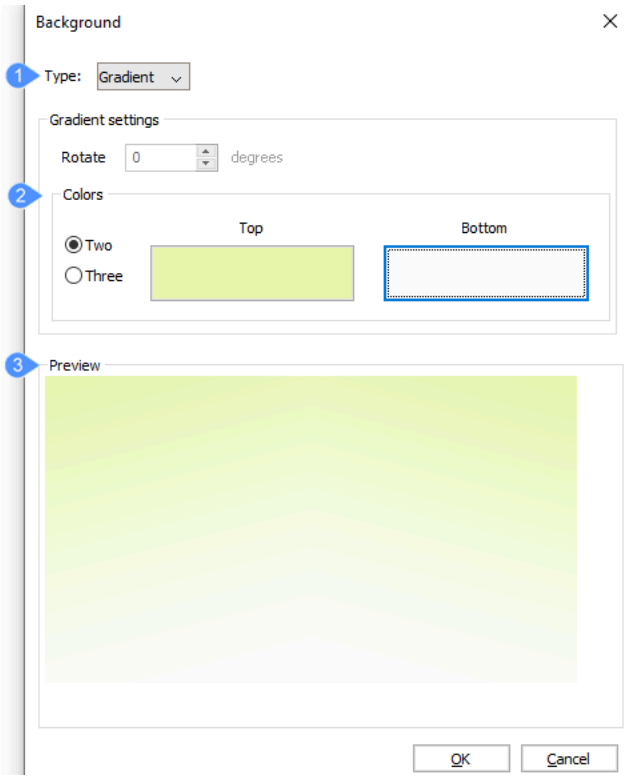
Type
Specifies the background type for the current viewport. When the background type is Solid, a single color is applied to the current viewport.

Solid settings
Specifies the current color for the solid background. You can click on the color swatch to display the Select Color dialog box and specify a different color.

Preview
Displays a preview of the background using the color you selected.

4.12.16.3 Gradient

When the background type is set to Gradient, the Background dialog box includes 3 areas:



1 Type

2 Gradient settings

3 Preview

Type
Specifies the background type for the current viewport. When the background type is Gradient, a gradient between 2 or 3 colors is applied to the current viewport.

Gradient settings
Specifies the rotation and colors for the gradient background.

Rotate field
Specifies the rotation angle of the gradient background. You can type an angle or use the arrows to increase or decrease the angle.

Colors
Specifies the colors for the gradient background.

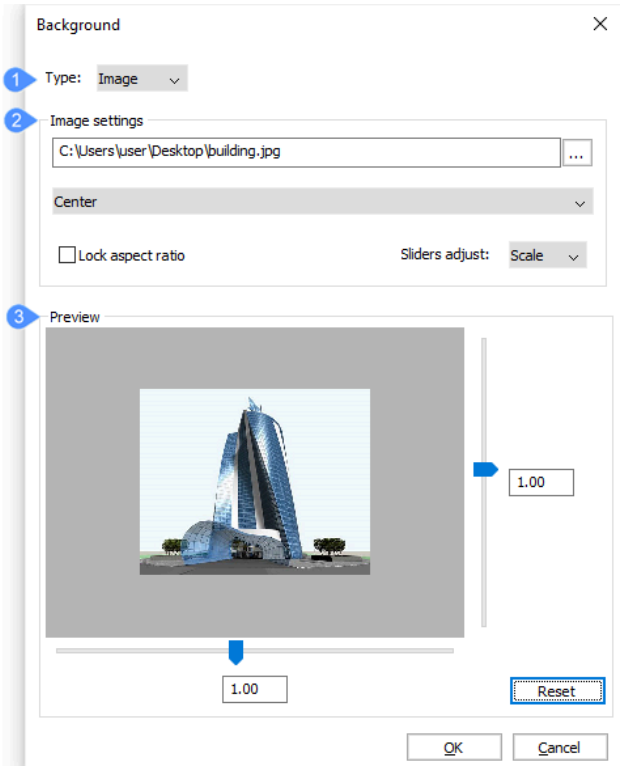
Two
Applies a two-color gradient to the background. When you select this option, the dialog box displays two color swatches. One is for the top color and one is for the bottom color. You can click on the color swatch to display the Select Color dialog box and specify a different color.

Three
Applies a three-color gradient to the background. When you select this option, the dialog box displays three color swatches. One is for the top color, one is for the middle color, and one is for the bottom color. You can click on the color swatch to display the Select Color dialog box and specify a different color.

Preview
Displays a preview of the background using the colors you have selected.

4.12.16.4 Image

When the background type is set to Image, the Background dialog box includes 3 areas:



1 Type

2 Image settings

3 Preview

Type
Specifies the background type for the current viewport. When the background type is Image, a raster image is applied to the current viewport.

Image settings
Specifies the file and position for the image background.

Path field
Specifies the name and location of the image file. You can enter the path by typing or choose the browse button to open the Select Image File dialog box.

Position
Specifies the position of the image within the current viewport. Options include Center, Stretch and Tile.

Center
Centers the image in the viewport.



Stretch
Stretches small images to fit the size of the viewport.



Tile
Tiles the image as needed to fill the viewport.



Lock aspect ratio
Specifies whether the aspect ratio of the image is maintained when the image is scaled. This option is only available 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 distorted; the x and y sliders move independently of each other.

Sliders adjust
Specifies the function of the 2 slides that display in the Preview pane.

- **Offset**: moves the image in the viewport.
- **Scale**: resizes the image; allows you to make the image the same size as the viewport.

Preview
Displays a preview of the background using the image file and options you specify.

X and Y sliders
Adjusts the offset and/or the scale of the image in the viewport. Which one is adjusted, offset or scale, is determined by the Slides adjust setting.

- **X**: moves the slider left and right. The X field updates accordingly. You can also type a value into the X field.
- **Y**: moves the slider up and down. The Y field updates accordingly. You can also type a value into the Y field.

Reset
Resets the offset to a default value of 0 and resets the scale to a default value of 1.

4.12.17 BIM BCF panel - Custom login dialog box

The BIM BCF Custom Log In dialog box allows you to connect to a cloud service to start collaborating on issues in the drawings.

Source URL

It is generally easier to use cloud services to manage BCF issues. If you wish to connect to these services from within BricsCAD BIM, you will need an account of these services first. This allows you to create issues and manage them in real-time from anywhere.

> ***Nota:*** It is not possible tocreateissues within BricsCAD BIM. The BCF panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

To connect to a cloud service, insert in the Source URL field one of the following Cloud services:

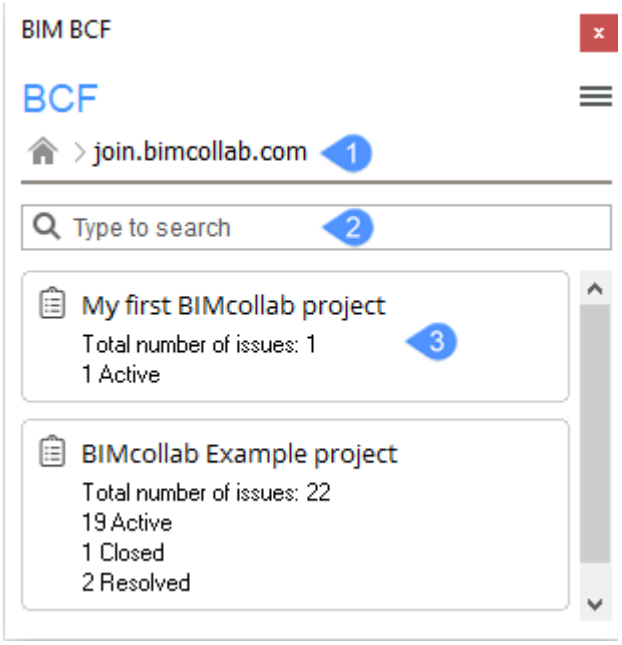
Cloud Service	Source URL
BIMcollab	<space name>.bimcollab.com *
BimSync	bcf.bimsync.com
BIMtrack	Bcfrestapi.bimtrackapp.co

Nota: * The Source URL for BIMcollab includes the name of the space you bought. If using a free account, use join.bimcollab.com.

4.12.17.1 Log in

Connects you to the cloud service. You are asked to insert the e-mail address and the password to connect.

After you successfully log in, you can close the browser window and start collaboration on issues in BricsCAD.



1 Source URL

2 Search

3 Project list

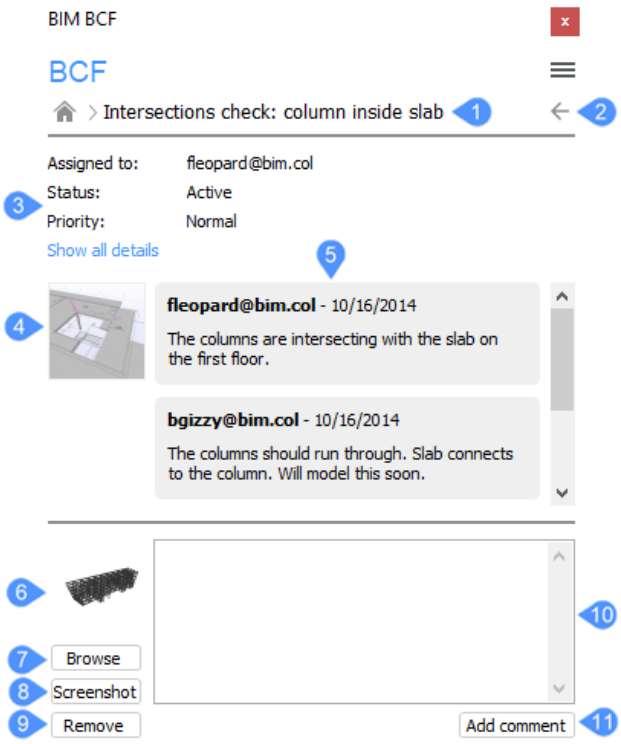
Source URL
Displays the source URL.

Search
Displays only the projects filtered by the word you type in the search bar.

Project list
Displays the list of projects that you are involved in.

4.12.17.2 Project list

After you left-click the project will open it and show a list of the issues. When you left-click on an issue, this dialog box will open.



1 Issue name

2 Issue overview

3 Details

4 Thumbnail image

5 Comments

6 Preview

7 Browse

8 Screenshot

9 Remove

10 Comment

11 Add comment

Issue name
Displays the name of the opened issue.

Issue overview
Returns you to the issue overview.

Details
Displays detailed information about the issue, such as status, author and creation date.

Thumbnail image
If the comment has an associated camera position, then clicking the thumbnail image will take the camera in the current drawing to this position.

This is useful if you want to know where exactly the issue is, and you do not want to manually search through the model.

Comments
Displays the comments on the issue.

Preview
Displays a preview of the issue after you press the button no. 8.

Browse
Opens the Select a viewpoint for this comment dialog box.

Screenshot
Insert a screenshot from model space in the preview field (no. 6).

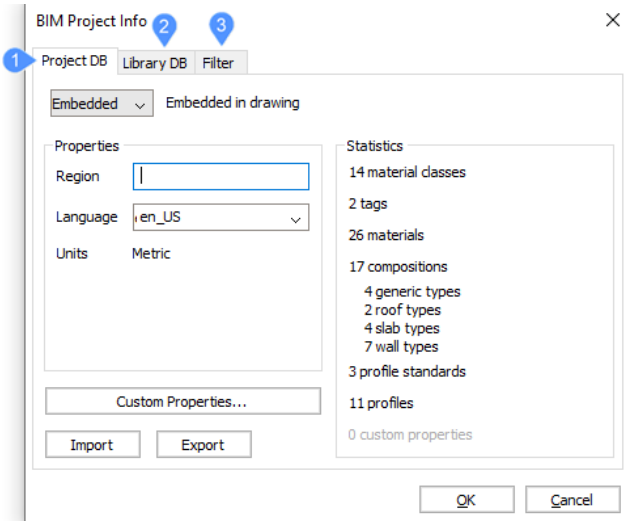
Remove
Removes the screenshot preview.

Comment
Allows you to insert a comment for an issue.

Add comment
Adds the comment you have inserted in field no. 10 to an existing issue.

4.12.18 BIM project info dialog box

The BIM Project Info dialog box allows you to set up your project and library database of your BIM models, to control the location, properties and content of the Project and Library databases.



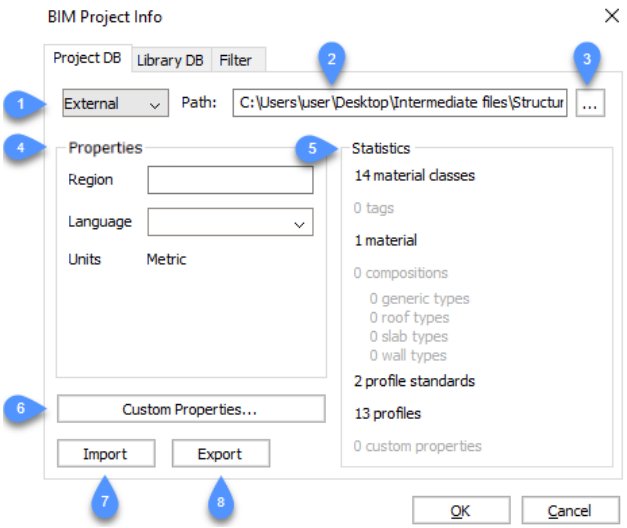
1 Project DB

2 Library DB

3 Filter

4.12.18.1 Project DB

Shows the BIM project info.



1 Project database drop-down list

2 Path

3 Browse

4 Properties

5 Statistics

6 Custom properties...

7 Import

8 Export

Project database drop-down list
Choose between Embedded or External.

Embedded
Saves the project library in the drawing file.

External
Saves the project library in a BIM database file (.bsyslib).

Nota: 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 External to Embedded the content of the external library is copied to the embedded project library.

Path
Displays the path where the project library can be found.

Browse
Opens the Select or create a Bricsys Library dialog box.

Properties
Region, Language and Units are the properties of the BIM project.

Region
Fill out the region field.

Language
You can choose one of the language options from the drop-down list.

Units
Displays the system of measurement used in the BIM project (metric, imperial).

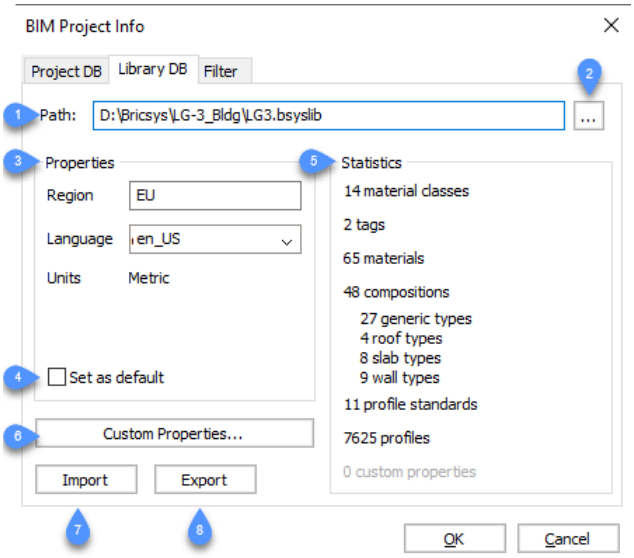
Statistics
Shows the location, properties, and the content of the current project library of the BIM model.

Custom properties…
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

Import
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

Export
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

4.12.18.2 Library DB



1 Path

2 Browse

3 Properties

4 Set as default

5 Statistics

6 Custom Properties…

7 Import

8 Export

Path
Displays the path where you the project library can be found.

Browse
Opens the Select or create a Bricsys Library dialog box.

Properties
Region, Language and Units are the properties of the project.

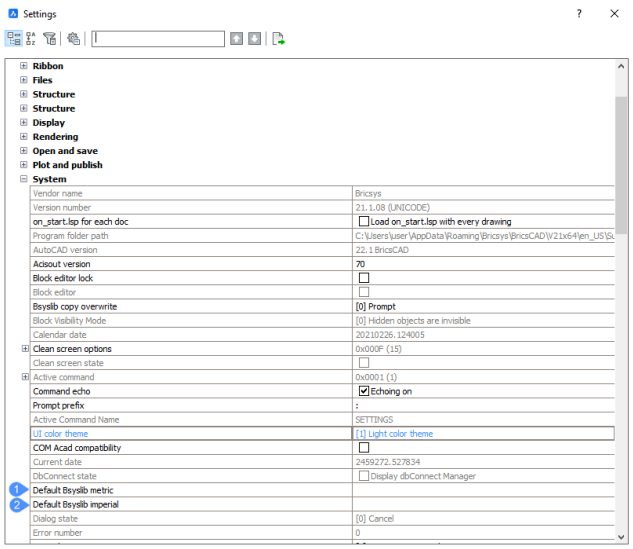
Region
Fill out the region field.

Language
You can choose one of the language options from the drop-down list.

Units
Displays the system of measurement used in the BIM project (metric, imperial).

Set as default
This option allows you to set the selected database as default library database for next projects.

Nota: You can edit the default library settings in the Settings dialog box under Program options/ System.



Statistics
Shows the location, properties, and the content of the current project library of the BIM model.

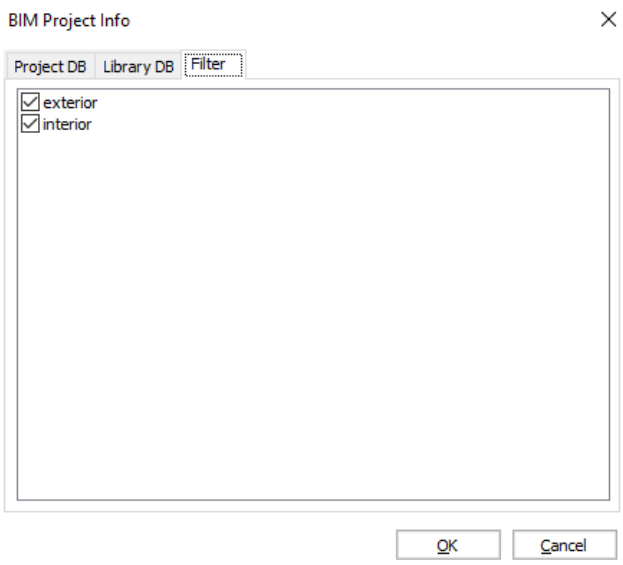
Custom Properties…
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

Import
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

Export
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

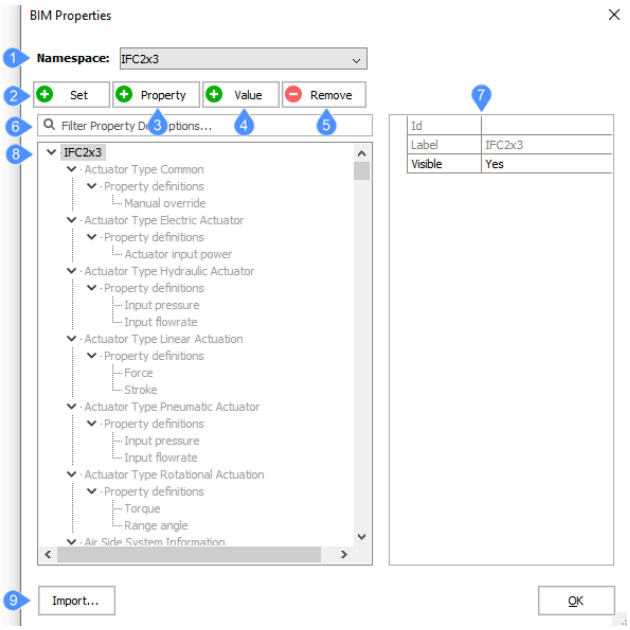
4.12.18.3 Filter

Check the tags you want to use in the filter.



4.12.19 BIM properties dialog box

The BIM Properties dialog box allows you to create, edit and delete property definitions and to organize properties in property sets.



1 Namespace

2 Add Set

3 Add Property

4 Add Value

5 Remove

6 Filter

7 Properties

8 Properties Tree

9 Import

4.12.19.1 Namespace

By default, 3 namespaces are available:

- **IFC2X3**: shows the IFC2x3 properties; is read-only. This namespace contains properties that are defined by the IFC2X3 schema.
- **Quantity**: shows the quantities properties; is read-only.
- **User**: allows you to create user defined properties.

It is also possible to import namespaces, using the **Import** button.

- **Classification System**: allows you to organize your BIM models with standard classification codes that are used in the construction industry.

4.12.19.2 Add Set

Creates a new property set.

To create a custom property set, first select the user option in the namespace drop-down list, then click the button.

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 is displayed. If you type in the **Label** field, the **Id** field will not be displayed any more. The Id must be unique.
- Per Instance: Allows to enable/disable user-defined properties, on a per-entity basis. When set, the BIM/Entity property sets property is available in the Properties panel when a BIM entity is selected.

4.12.19.3 Add Property

Creates a new property set.

4.12.19.4 Add Value

Creates a new value definition.

4.12.19.5 Remove

Removes the selected property set, property definition or value definition.

4.12.19.6 Filter

Displays only the properties of which the Id and Label contain the search string typed. The properties tree is filtered as you type.

4.12.19.7 Properties

- **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, then the Id is used instead.
- **Visible**: defines whether the namespace properties are visible in the Properties panel.

4.12.19.8 Properties Tree

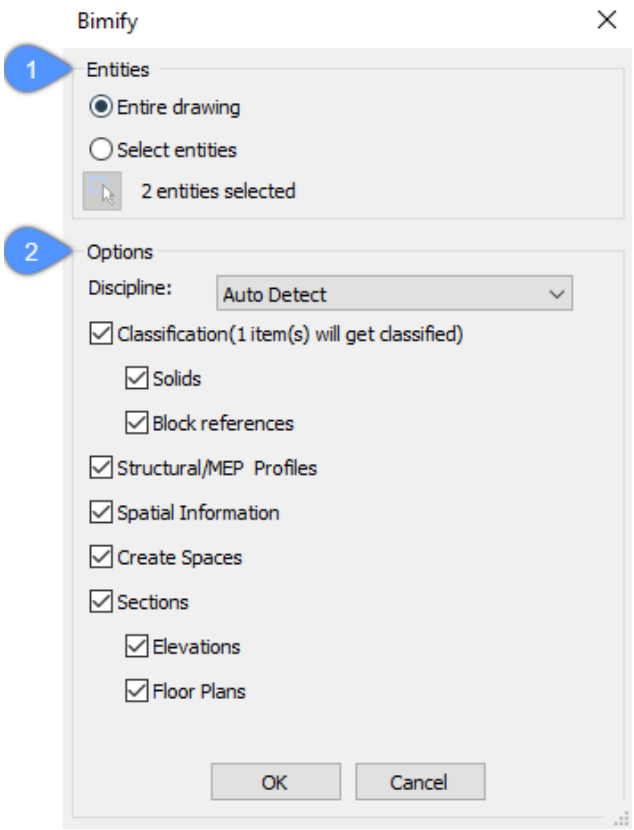
Displays the property sets, property definitions and value definitions.

4.12.19.9 Import

Opens the Select XML file to import dialog box.

4.12.20 Bimify dialog box

The Bimify dialog box allows you to analyze the model and to run an automatic classification and a spatial location assignment for the entire model.



1 Entities

4.12.20.1 Entities

Specifies if the BIMIFY command needs to be run for all entities in the drawing or just several selected entities.

4.12.20.2 Options

Discipline
Determines to what discipline the model belongs to in order to improve the automatic classification.

- **Auto Detect**: an AI algorithm determines the discipline automatically.
- **Architectural**: the model is classified as an architectural model.
- **MEP**: the model is classified as a MEP model.
- **Mixed Model**: the model cannot be assigned to 1 discipline and is perceived as a mixed model.

Classification
Defines the classification of solids and blocks.

- **Solids**: makes sure that 3D solids are classified as walls, slabs, etc.
- **Blocks**: makes sure that blocks are classified as windows, doors, etc.

Structural/MEP profiles
Assigns columns, beams, members or flow segments profiles that are available in BricsCAD BIM profile panel.

> ***Nota:*** If no match can be found for a profile element in the standard library, the *BIMIFY* command creates a new custom profile in the project's library.

Spatial information
Assigns spatial locations (buildings and floors).

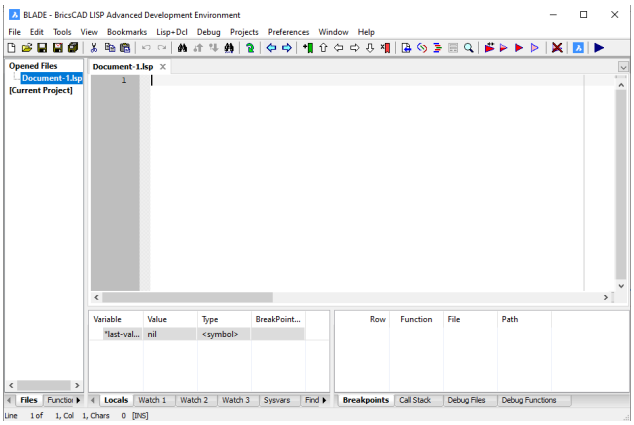
Create Spaces
Detects external walls and sets the property Wall Common/is External = ON.

Sections
Creates BIM Section entities.

- **Elevations**: creates 4 elevations (Front, Back, Left and Right).
- **Floor Plans**: creates a plan section for each floor.

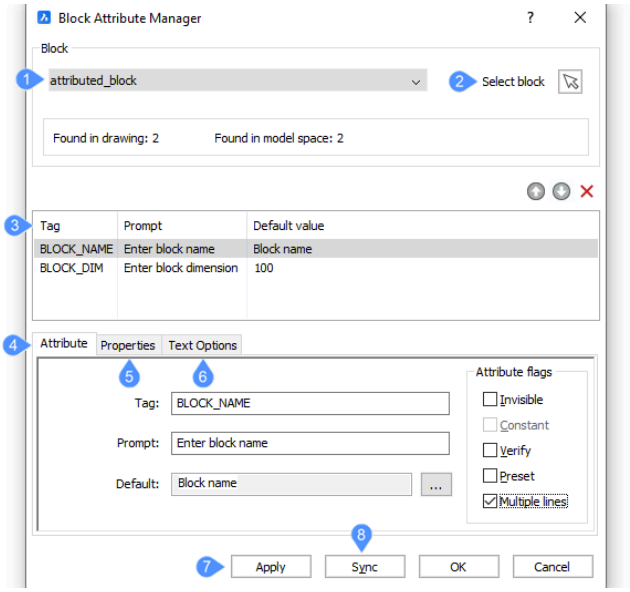
4.12.21 Blade - BricsCAD LISP advanced development environment dialog box

The Blade - BricsCAD LISP Advanced Development Environment dialog box allows you to edit and debug LISP applications.



4.12.22 Block attribute manager dialog box

The Block attribute manager dialog box allows you to edit all aspects of attributes in a block definition and then optionally apply the changes to all blocks of the same name in the drawing.



1 Block name

2 Select block

3 Attribute list

4 Attribute tab

5 Properties tab

6 Text Options Tab

7 Apply

8 Sync

4.12.22.1 Block name

Select a block from the drawing whose attributes you want to edit.

You can select a block from the drop-down list.

4.12.22.2 Select block

Allows you to select a block in the drawing.

4.12.22.3 Attribute list

Select an attribute from the list. It includes the tag, the prompt and the default value.

: moves the attribute up the list.

: moves the attribute down the list.

: deletes the attribute up the block.

4.12.22.4 Attribute tab

Allows you to edit the definition of the selected attribute.

There are 5 attribute flags options:

- **Invisible**: hides the attributes from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the *ATTDISP* 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 command.
- **Multiple lines**: when checked, allows the use of multiline text.

4.12.22.5 Properties tab

The**Properties**tab allows you to edit the properties of the selected attribute.

4.12.22.6 Text Options Tab

The**Text Options**tab allows you to edit the text properties of the selected attribute.

4.12.22.7 Apply

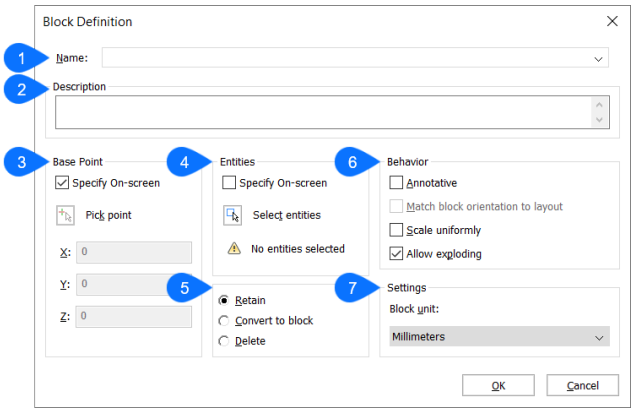
Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.

4.12.22.8 Sync

Applies the changes to all instances of the same block in the drawing.

4.12.23 Block definition dialog box

The block definition dialog box allows you to create a new block definition.



1 Name

2 Description

3 Base Point

4 Entities

5 Block Options

6 Behavior

7 Settings

4.12.23.1 Name

Specifies the name of the block.

4.12.23.2 Description

Provides a description of the block (Optional).

4.12.23.3 Base Point

Modifies the properties of the base point of the block definition. This is the point at which the block will be inserted with the INSERT command.

Specify On-screen
Allows you to define the base point in the model space after you click the **OK** button, if the box is checked.

Pick point
Allows you to pick a point in the drawing for the block's base point. You can also enter the X,Y,Z coordinates in the drawing.

X/Y/Z
Defines the coordinates where the block should be inserted. The Z coordinate is usually optional.

4.12.23.4 Entities

Selects the entities that make up the block.

Specify On-screen
Allows you to select the entities in the model space after you click the **OK** button, if the box is checked.

Nota: Optionally, include Reference Curves in the selection, which allows to automatically align a block during insertion.

Select entities
Selects one or more entities in the drawing.

4.12.23.5 Block Options

Defines how the entities are turned into a block.

Nota: The Convert to block option is the most efficient one.

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.

4.12.23.6 Behavior

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 the block to follow annotative scaling:

- **On** – the block will scale itself to whatever annotative scale is in effect. The Scale Uniformly option is grayed out (unavailable).
- **Off** - the block follows the scale factor you give the block during the *Insert* 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 scaled non-uniformly. This option is not available to annotatively-scaled blocks.

- **On** - X, Y, and Z scale factors of blocks are the same. This prevents blocks from being distorted.
- **Off** - blocks can be inserted with different X, Y, and Z scale factors. This is useful for objects that can have different dimensions, 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.

Dica: To edit the entities of an unexploded block, use the *BEDIT* command.

- **On** - block can be exploded after being inserted, with the Explode command

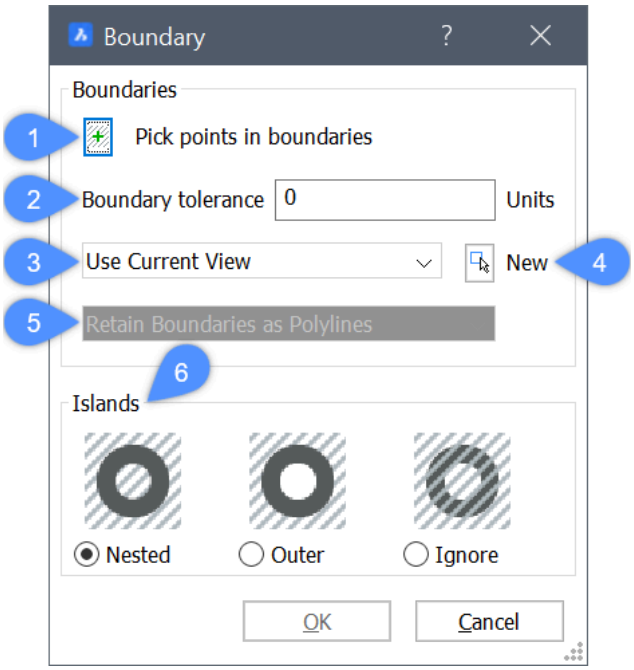
- **Off** - blocks cannot be exploded; you can change this property with the *EXPBLOCKS* command, Blocks section.

4.12.23.7 Settings

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.

4.12.24 Boundary dialog box

The Boundary dialog box allows you to create closed polylines, defined by surrounding entities.



1 Pick points in boundary
2 Boundary tolerance
3 Boundary set
4 New
5 Retain Boundaries as Polylines
6 Islands

4.12.24.1 Pick points in boundary

It allows you to specify a point inside an enclosed area for which you want to create a boundary. It temporarily dismisses the Boundary dialog box.

4.12.24.2 Boundary tolerance

Specify the largest gap that can exist in the boundary for BricsCAD to consider the area closed.

Nota: When boundary tolerance is 0, no gaps can exist.

4.12.24.3 Boundary set

Specify where BricsCAD 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.

4.12.24.4 New

It allows you to create a new selection set of entities that make up the boundary. It temporarily dismisses the Boundary dialog box.

4.12.24.5 Retain Boundaries as Polylines [Read only]

Indicates how boundaries are retained.

4.12.24.6 Islands

Specify island detection. An island is an enclosed area inside a boundary.

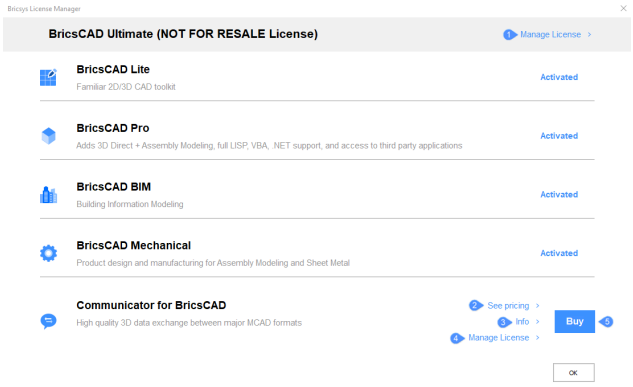
Nested
Treats every island as a boundary.

Outer
Creates a boundary only of the outermost entities.

Ignore
Ignores the innermost areas. A boundary is created between the outermost area and islands.

4.12.25 Bricsys license manager dialog box

The Bricsys License Manager dialog box allows you to activate/deactivate the software licenses for BricsCAD and Communicator.



1 Manage License

2 See pricing

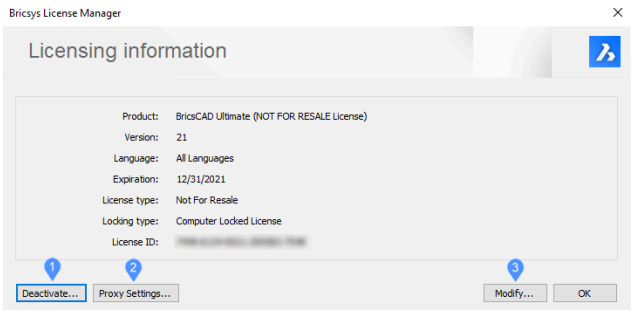
3 Info

4 Manage License for Communicator

5 Buy

4.12.25.1 Manage License

Opens the Bricsys License Manager dialog box.



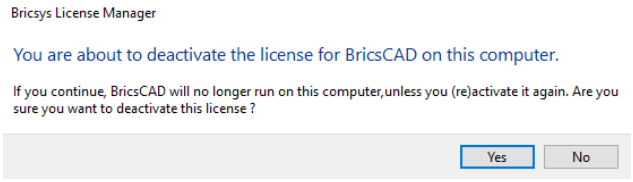
1 Deactivate

2 Proxy Settings...

3 Modify...

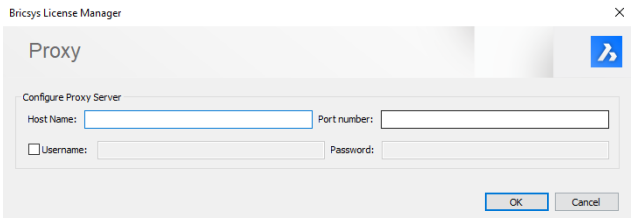
Deactivate

Opens the Bricsys License Manager dialog box for confirming deactivation.



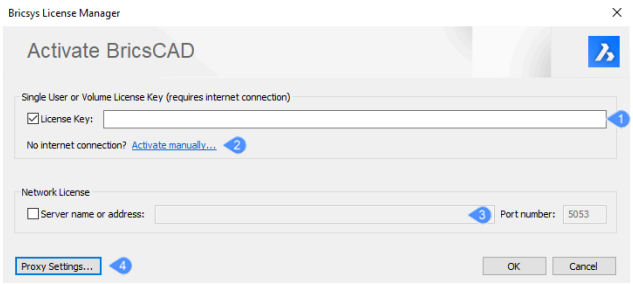
Proxy Settings...

Opens the Bricsys License Manager dialog box where you can configure the proxy server.



Modify...

Opens the Bricsys License Manager dialog box for activating BricsCAD where you can insert your license key, your network license or activate it manually.



1 License Key

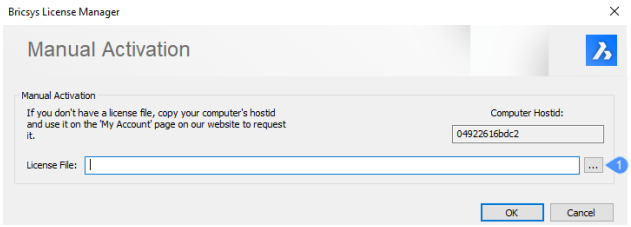
2 Activate manually...

3 Network License

4 Proxy Settings...

License Key
If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually...
Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic



1 Select license file

Select license file
Opens the Select license file dialog box.

Network License
If you are using a network license, enter the host name or the IP address of the network license server.

Proxy Settings...
Opens the Bricsys License Manager dialog box to configure the proxy server.

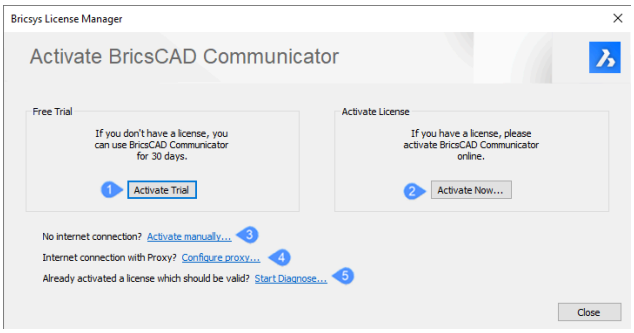
4.12.25.2 See pricing

Opens the Bricsys official website (Bricsys), where you can see the prices for Communicator and 1 year/3 years/lifetime BricsCAD subscription.

4.12.25.3 Info

Opens the Communicator Info dialog box.

4.12.25.4 Manage License for Communicator



1 Activate Trial

2 Activate Now

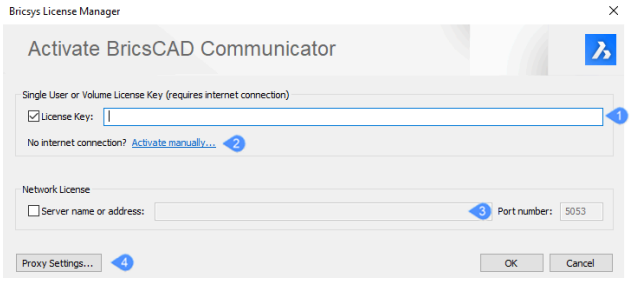
3 Activate manually...

4 Configure proxy...

5 Start Diagnose...

Activate Trial
Activates the 30 days free trial of BricsCAD Communicator.

Activate Now
Opens the Activate BricsCAD Communicator dialog box.



1 License Key

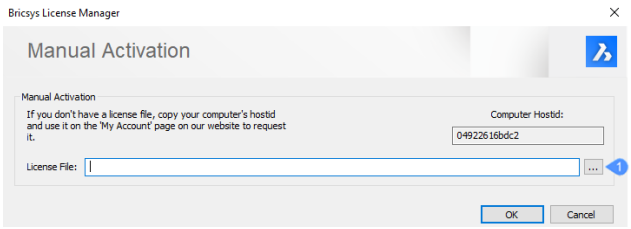
2 Activate manually

3 Network License

4 Proxy settings

License Key
If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually…
Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic



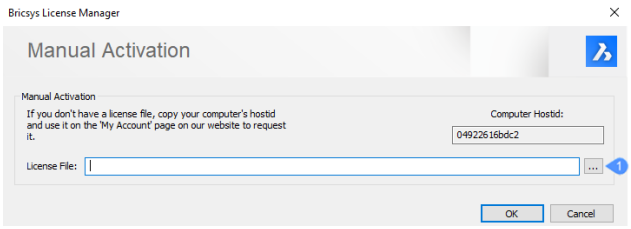
1 Select license file

Select license file
Opens the Select license file dialog box.

Network License
If you are using a network license, enter the host name or the IP address of the network license server.

Proxy Settings…
Opens the Bricsys License Manager dialog box to configure the proxy server.

Activate manually…
Opens the Bricsys License Manager dialog box for activating BricsCAD Communicator.



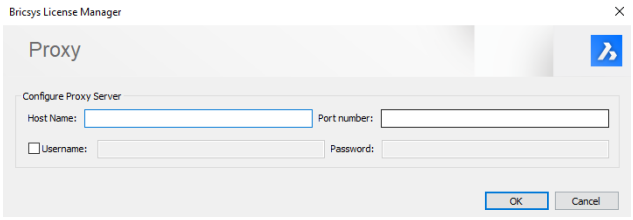
1 Select license file

Select license file

Opens the Select license file dialog box.

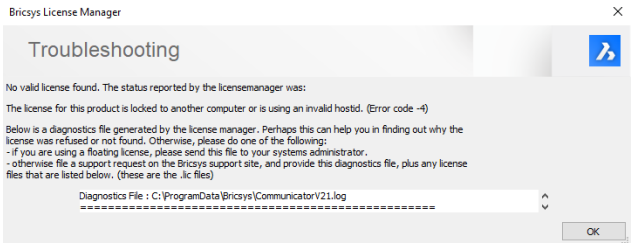
Configure proxy…

Opens the Bricsys License Manager dialog box where you can configure the proxy server.



Start Diagnose…

Opens the Bricsys License Manager dialog box for Troubleshooting.



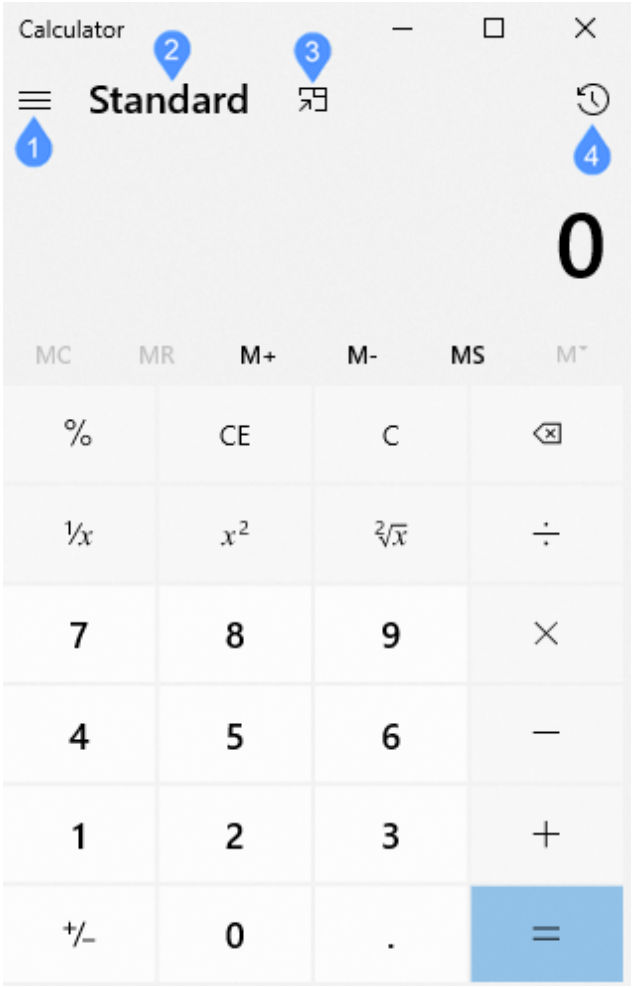
4.12.25.5 Buy

Opens the Bricsys official website (Bricsys) where you can buy or subscribe to BricsCAD.

4.12.26 Calculator dialog box

The Calculator dialog box displays the operating system's software calculator.

When you are using the Standard view, it includes 4 areas.



1 View

2 View's name

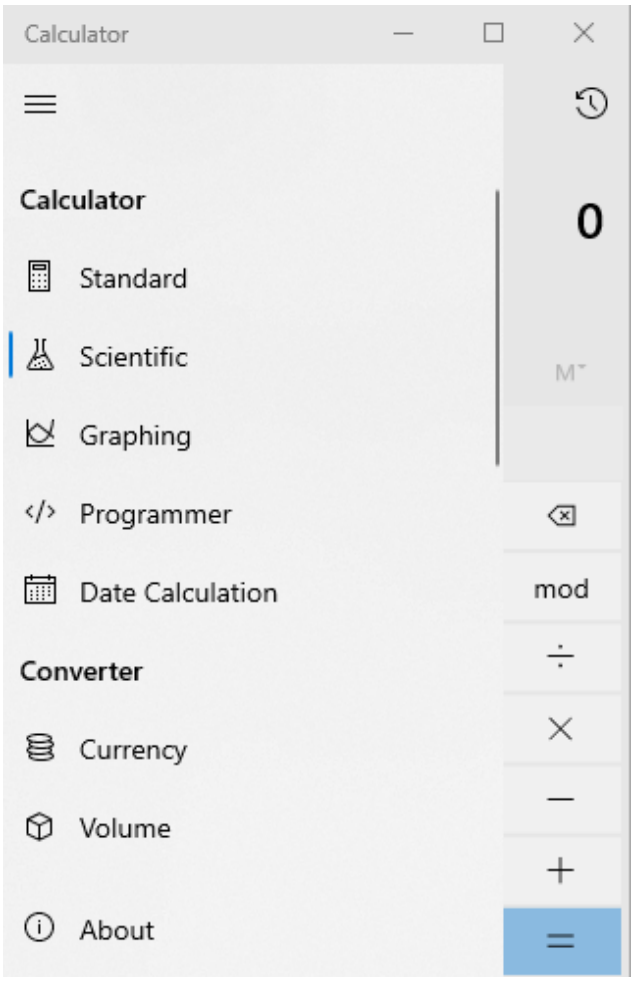
3 Keep on top

4 History

4.12.26.1 View

Displays the possible views you can choose from.

It has 2 categories to choose from: **Calculator** (Standard, Scientific, Graphic, Programmer, Date Calculation) and **Converter** (Currency, Volume, Length, Weight and Mass, Temperature, Energy, Area, Speed, Time, Power, Data, Pressure, Angle).



4.12.26.2 View's name

Displays the current view name.

4.12.26.3 Keep on top

You can also use the keyboard shortcut ALT + Up to access this function.

Note that this function is available only for the Standard view.

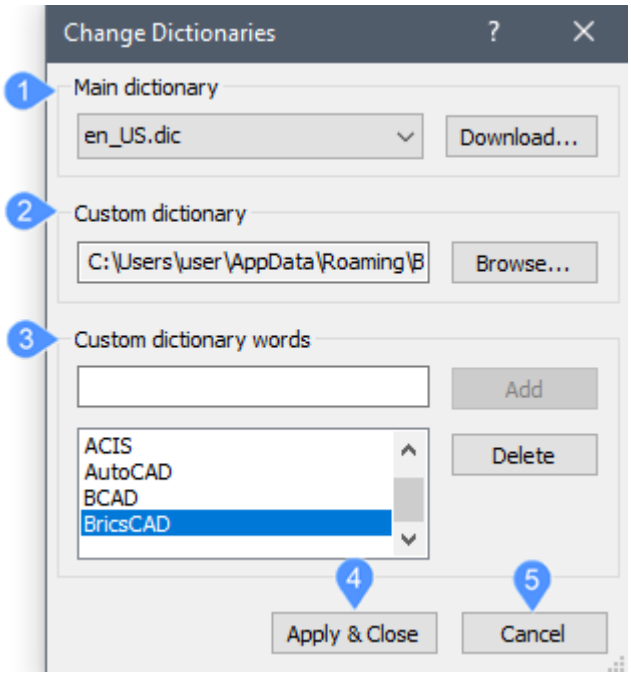
4.12.26.4 History

Displays the history of your calculations done in the current session.

Nota: This function is available only for the Standard and Scientific calculator.

4.12.27 Change dictionaries dialog box

The Change dictionaries dialog box allows you to change dictionaries.



1 Main dictionary

2 Custom dictionary

3 Custom dictionary words

4 Apply & Close

5 Cancel

4.12.27.1 Help

Opens the Bricsys Help article for the SPELL command.

4.12.27.2 Main dictionary

Specifies the main dictionary to use for spell checking.

File name
Specify the dictionary file to use as the main dictionary. Available files are displayed in the drop-down list.

Download
Download additional dictionaries from OpenOffice.org. Opens the Web browser at: https://extensions.openoffice.org/dictionaries

4.12.27.3 Custom dictionary

Specifies a custom dictionary to use for spell checking. Use custom dictionaries for discipline-specific words, such as medical or mechanical terms.

File location
Specifies the file name and location of a custom dictionary to use in addition to the main dictionary.

Browse
Open the Select Custom Dictionary dialog box to select a custom dictionary file.

4.12.27.4 Custom dictionary words

Specifies words to be included in the custom dictionary.

Word field
Allows you to enter a word to be included in the custom dictionary.

Add
Add the word you entered in the word field to the custom dictionary.

Words list
Displays all the words included in the custom dictionary.

Delete
Delete a selected word from the custom dictionary.

4.12.27.5 Apply & Close

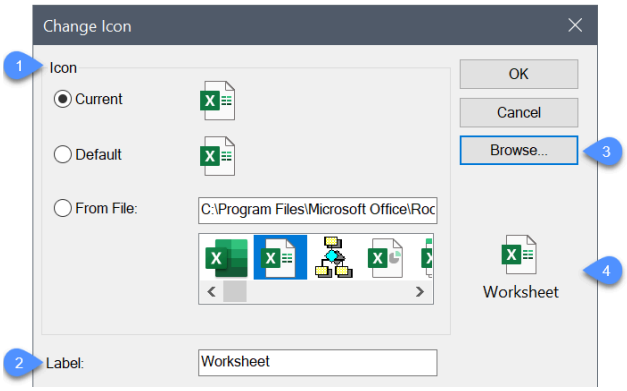
Apply your changes and close the dialog box. The custom dictionary file is only updated with added or deleted words after you choose Apply and Close.

4.12.27.6 Cancel

Close the dialog box without applying changes.

4.12.28 Change Icon dialog box

The Change Icon dialog box allows you to change the icon of an OLE Frame entity before inserting it into the drawing. The icon is related to the document type and it is defined by the source applications.



1 Icon

2 Label

3 Buttons

4 Preview

4.12.28.1 Icon

Specifies an icon to represent an embedded or linked document.

Current
Displays the current icon.

Default
Displays the default icon.

From File
Choose another icon.

4.12.28.2 Label

Optionally edit the label text.

4.12.28.3 Buttons

The Buttons area contains OK, Cancel and the Browse button.

OK
Click OK to accept the changes and to return to the previous dialog box (Insert Object).

Cancel
Click Cancel to discard the changes and to return to the previous dialog box (Insert Object).

Browse
Displays the Browse dialog box (a standard file selection dialog box), in which you can select an icon from a file.

4.12.28.4 Preview

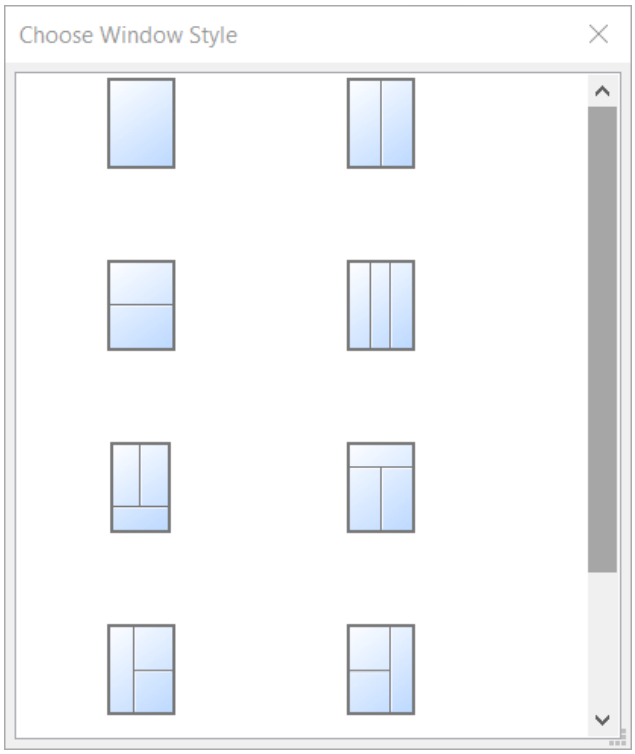
Displays the icon preview.

4.12.29 Choose window style dialog box

The Choose Window Style dialog box allows you to specify the style of a window.

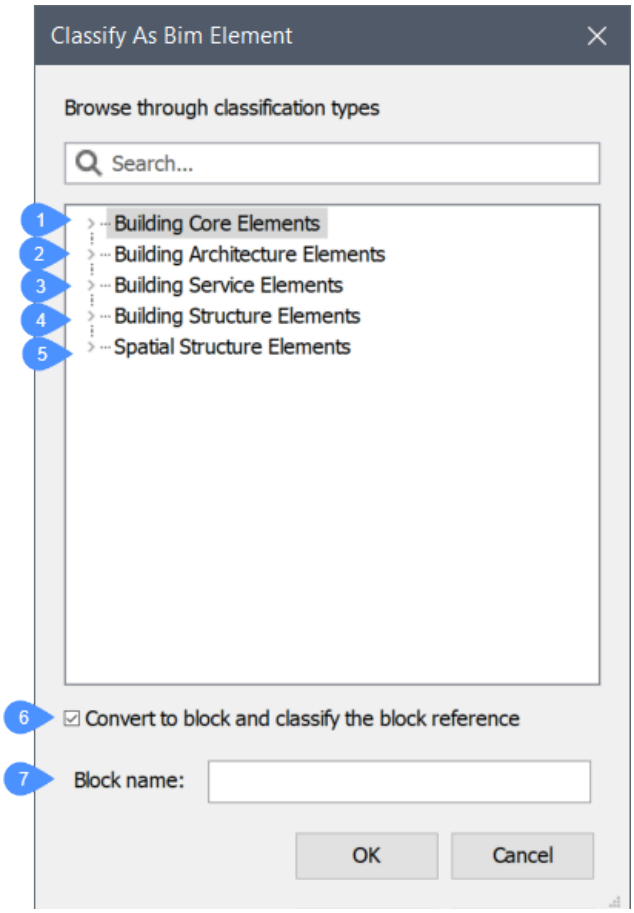
A series of parameters is created, which can be edited in the Parameters section of the Properties panel.

Rectangular windows and openings have a W (Width) and H (Height) parameter.



4.12.30 Classify As Bim Element dialog box

The **Classify As Bim Element** dialog box specifies the types of classification or converts the selection to a block and classifies the block reference.



1 Building Core Elements

2 Building Architecture Elements

3 Building Service Elements

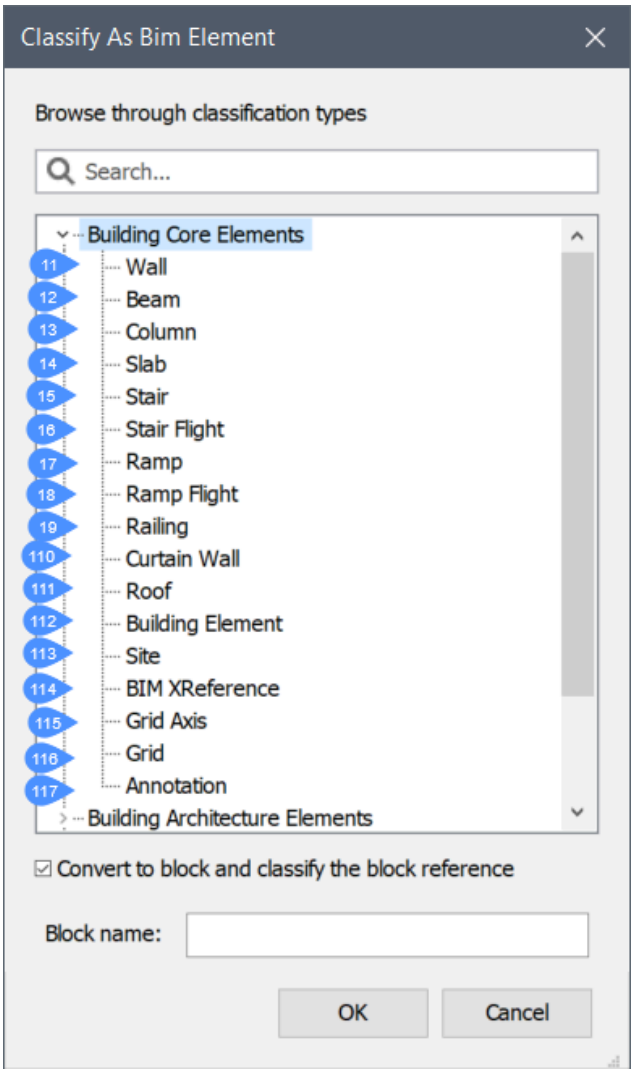
4 Building Structure Elements

5 Spatial Structure Elements

6 Convert to block and classify the block reference

7 Block name

4.12.30.1 Building Core Elements



11. Wall

12. Beam

13. Column

14. Slab

15. Stair

16. Stair Flight

17. Ramp

18. Ramp Flight

19. Railing

110. Curtain Wall

111. Roof

112. Building Element

113. Site

114. BIM XReference

115. Grid Axis

116. Grid

117. Annotation

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 2 surfaces at different levels.
Example: Slope/Ramp interrupted with landings.

Ramp Flight
Inclined slab segment, normally providing a human circulation link between 2 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.

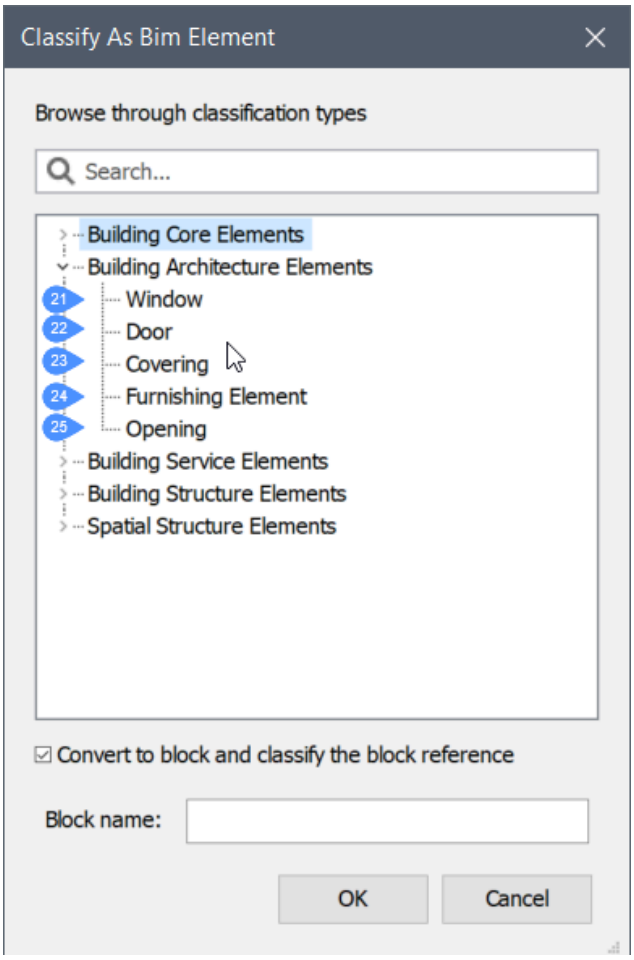
BIM XReference
The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.

Grid Axis
An individual axis that is defined in the context of a design grid. The axis definition is based on a curve of dimensionality 2.
Example: X-axis, Y-axis, Z-axis.

Grid
A planar design grid defined in 3D space used as an aid in locating structural and design elements.
Example: rectangular grid, radial grid, triangular grid.

Annotation
A graphical representation within the geometric context of a project, that adds a note or meaning to the objects of the project.
Example: additional line drawings, text, dimensioning, hatching, ...

4.12.30.2 Building Architecture Elements



21. Window

22. Door

23. Covering

24. Furnishing Element

25. Opening

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

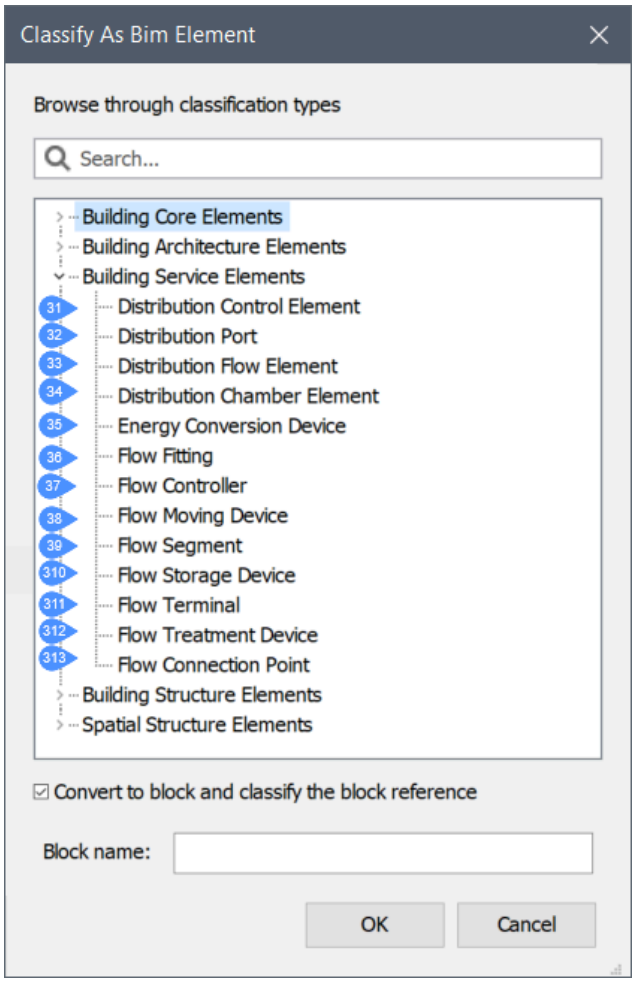
A covering is an element which covers some part of another element and is fully dependent on that other element.

Example: wall claddings, floorings, suspended ceilings, finish trim, base molding.

Furnishing Element
Generalization of all furniture related objects.
Example: closet, table, chair, bed.

Opening
Represents a void within any element that has physical manifestation.
Example: opening for a door, opening for a window.

4.12.30.3 Building Service Elements



31. Distribution Control Element

32. Distribution Port

33. Distribution Flow Element

34. Distribution Chamber Element

35. Energy Conversion Device

36. Flow Fitting

37. Flow Controller

38. Flow Moving Device

39. Flow Segment

310. Flow Storage Device

311. Flow Terminal

312. Flow Treatment Device

313. Flow Connection Point

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 3 ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member of 2 systems (GAS and DOMESTICCOLDWATER) and hosts 1 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 2 ports.

Example: section of a duct, pipe, conduit, etc.

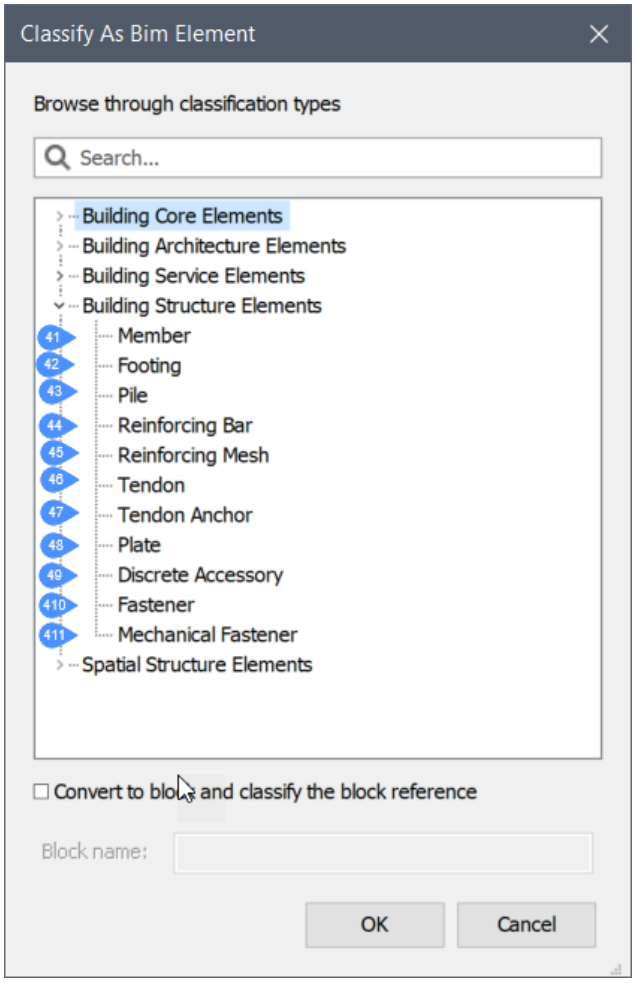
Flow Storage Device
Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow.
Example: tank, battery

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

Flow Connection Point
The connection point is used to describe the geometric constraints that facilitate the physical connection of 2 objects at a point.

4.12.30.4 Building Structure Elements



41. Member

42. Footing

43. Pile

44. Reinforcing Bar

45. Reinforcing Mesh

46. Tendon

47. Tendon Anchor

48. Plate

49. Discrete Accessory

410. Fastener

411. Mechanical Fastener

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 transferred 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 pre-stress 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 pre-stressed or post-tensioned concrete.

Plate

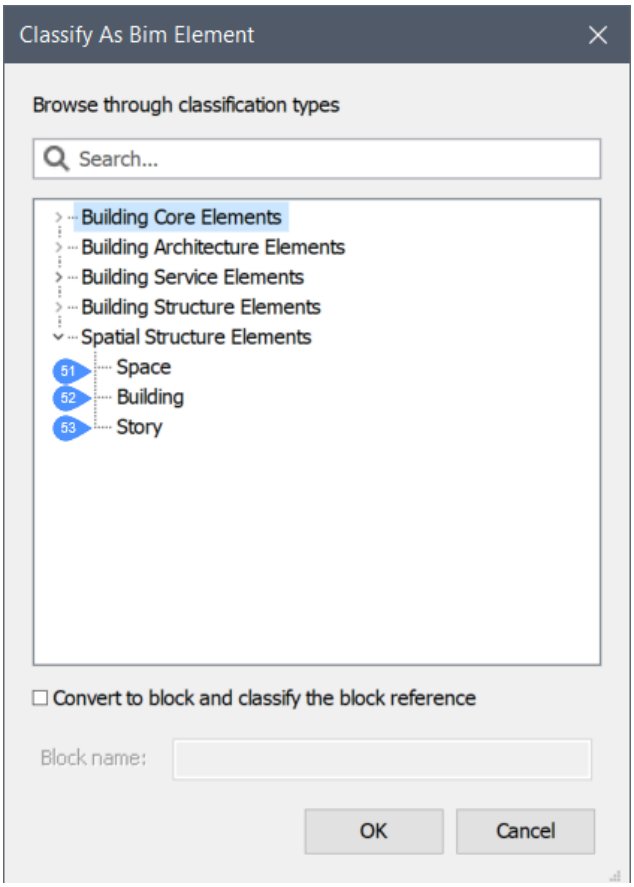
A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.

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

4.12.30.5 Spatial Structure Elements



51. Space

52. Building

53. Story

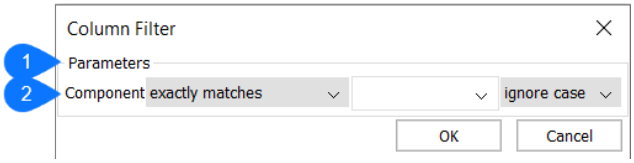
Space
BIM Space entities from enclosed boundaries (3D solids or linear 2D entities).

Building
Buildings in the model.

Story
Story of the building / spatial elements.

4.12.31 Column filter dialog box

The Column Filter dialog box configures the column filter.



1 Parameters

2 Column

4.12.31.1 Parameters

Defines the parameters to use in the column filter.

4.12.31.2 Column type

Displays the name of the column you put a filter on. Selects the type of filter that should be applied to this column and configures it.

Nota: The list of possible filters depends on the column content type.

Nota: For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

Always accepted
All column values are accepted.

Equal to
Value should be equal to the target one.

Nota: Appears with numerical columns.

Not equal to
Value should be not equal to the target one.

Nota: Appears with numerical columns.

Less than
Value should be less than the target one.

Nota: Appears with numerical columns.

Greater than
Value should be greater than the target one.

Nota: Appears with numerical columns.

Less than or equal to
Value should be less than or equal to the target one.

Nota: Appears with numerical columns.

Between
Value should be in the given range, both ends excluded.

Nota: Appears with numerical columns.

Not in range
Value should not be in the given range, both ends excluded.

Nota: Appears with numerical columns.

Exactly matches
The content of the column should exactly match the target one.

Nota: Appears with columns containing text.

Does not match
The content of the column should be different from the target one.

Nota: Appears with columns containing text.

Starts with
The content of the column should start with the given text.

Nota: Appears with columns containing text.

Does not start with
The content of the column should not start with the given text.

Nota: Appears with columns containing text.

Ends with
The content of the column should end with the given text.

Nota: Appears with columns containing text.

Does not end with
The content of the column should not end with the given text.

Nota: Appears with columns containing text.

Contains
The content of the column should contain the given text.

Nota: Appears with columns containing text.

Does not contain
The content of the column should not contain the given text.

Nota: Appears with columns containing text.

Matches pattern
The content of the column should match the given wildcard template.

Nota: Appears with columns containing text.

Does not match pattern
The content of the column should not match the given wildcard template.

Nota: Appears with columns containing text.

Included in
The content of the column should be in the given set of values.

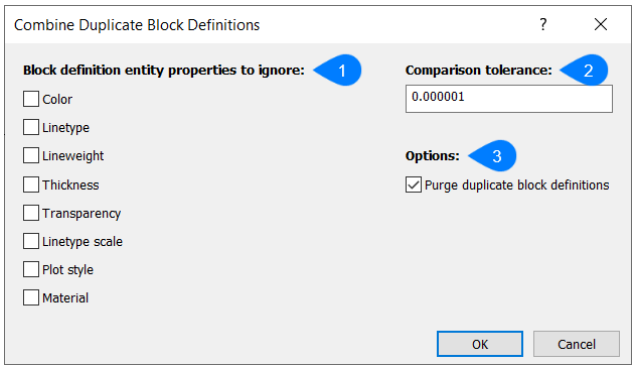
Nota: Appears with columns containing text and numerical columns.

Not included in
The content of the column should not be in the given set of values.

Nota: Appears with columns containing text and numerical columns.

4.12.32 Combine Duplicate Block Definitions dialog box

The Combine Duplicate Block Definitions dialog box allows you to search the drawing for duplicate block definitions and replace all such block instances by the most recent of these duplicates.



1 Block definition entity properties to ignore

2 Comparison tolerance

3 Options

4.12.32.1 Block definition entity properties to ignore

Specifies the block properties to ignore when searching for duplicate block definitions.

4.12.32.2 Comparison tolerance

Sets the tolerance for overlapping blocks. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

4.12.32.3 Options

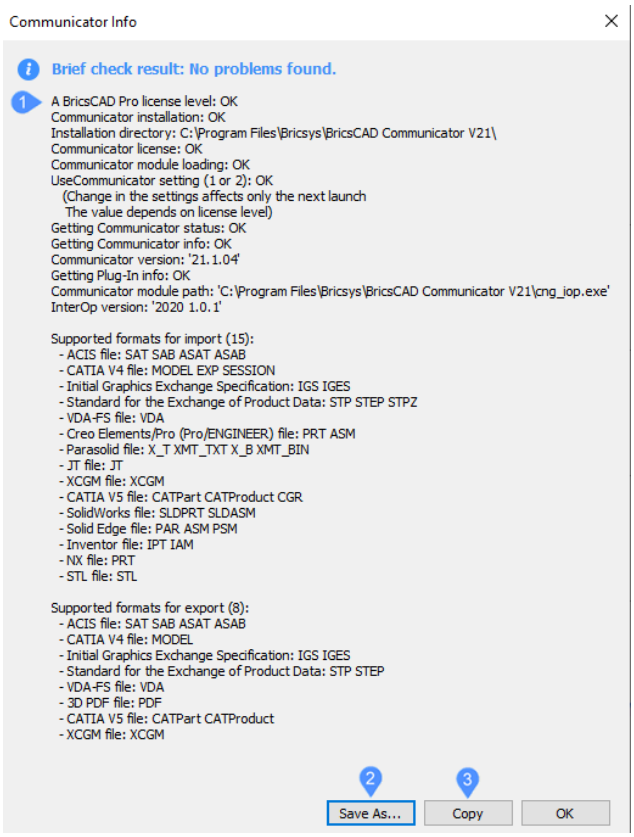
Purge duplicate block definitions

Deletes duplicate block definitions.

4.12.33 Communicator info dialog box

The Communicator Info dialog box checks whether the Communicator add-on is correctly installed and provides a diagnostic report.

The command is available on Windows only.



1 Diagnostic

2 Save As...

3 Copy

4.12.33.1 Diagnostic

Displays a diagnostic report for the Communicator add-on installation, supported formats to import and to export.

4.12.33.2 Save As...

Opens the Save Communicator Info As dialog box.

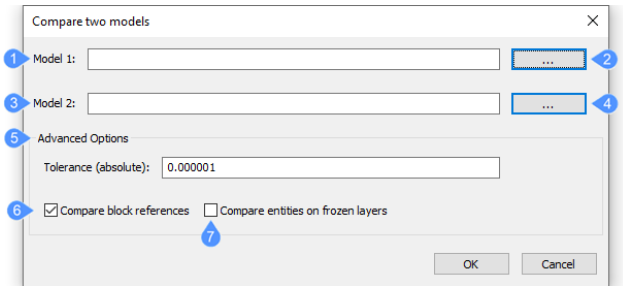
4.12.33.3 Copy

Copies the text displayed in this dialog box to clipboard.

4.12.34 Compare two models dialog box

The Compare Two Models dialog box allows you to perform a geometric comparison of solids and surfaces between two drawing files.

The result is shown in a new drawing and reported in the Mechanical Browser panel.



1 Model 1

2 Select 1st drawing

3 Model 2

4 Select 2nd drawing

5 Advanced Options

6 Compare block reference

7 Compare entities on frozen layers

4.12.34.1 Model 1

Specifies the file name of the first drawing to compare.

4.12.34.2 Select 1st drawing

Opens the Select first drawing for comparison dialog box to choose the drawing you want.

4.12.34.3 Model 2

Specifies the file name of the second drawing to compare.

4.12.34.4 Select 2nd drawing

Opens the Select second drawing for comparison dialog box to choose the drawing you want.

4.12.34.5 Advanced Options

The tolerance (absolute) expresses the threshold value for geometry comparison: determines how close two models must be to be considered unchanged.

4.12.34.6 Compare block reference

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.

4.12.34.7 Compare entities on frozen layers

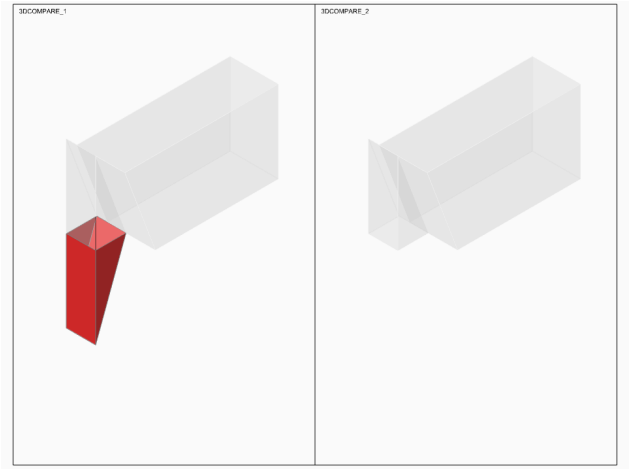
Determines if entities on frozen layers should be compared:

- **Yes** - solids and surfaces on frozen layers are compared.
- **No** - solids and surfaces on frozen layers are ignored.

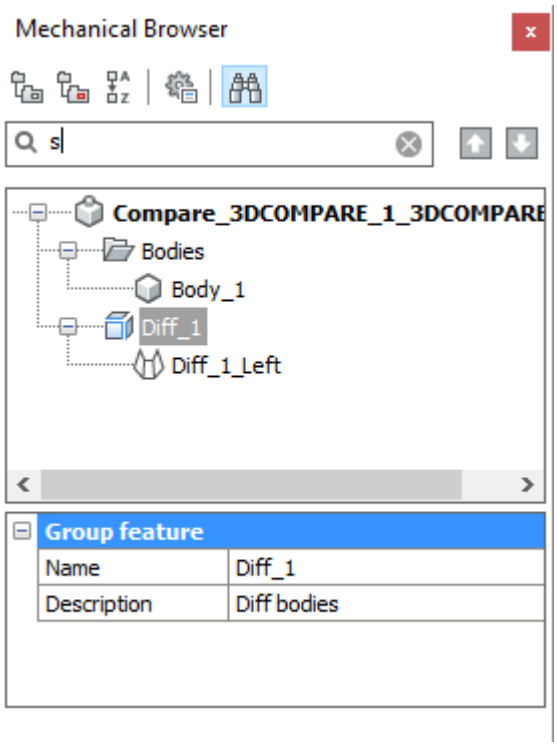
The value of the *3DCOMPAREMODE* system variable controls the display in the viewports of the Comparison layout; in the Settings dialog box, search for "3dcomparemode".

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 drawing using colors.

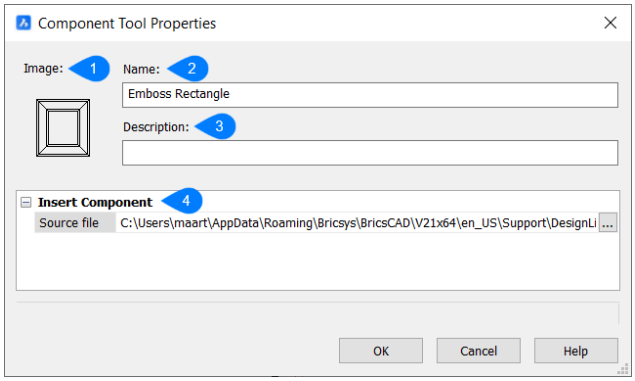


At the same time, the Mechanical Browser panel opens automatically and lists the differences between the 2 drawings.



4.12.35 Component Tool Properties

The Component Tool Properties dialog box allows you to view and change the basic properties of components in the tool palettes panel.



1 Image

2 Name

3 Description

4 Insert Component

4.12.35.1 Image

Displays the image that is used as the icon for the selected component in the tool palettes panel.

4.12.35.2 Name

Change the name of the selected component.

4.12.35.3 Description

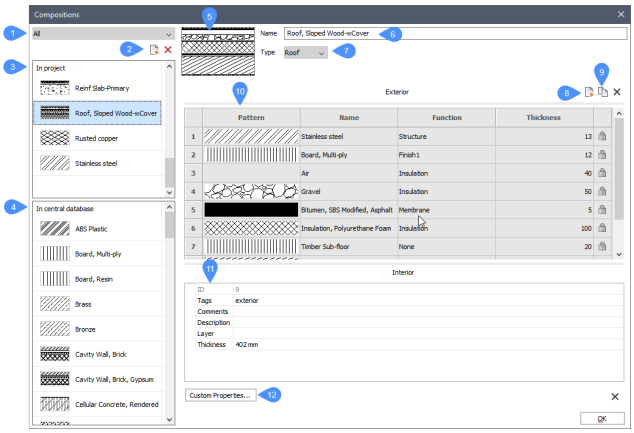
Provide an optional description of the component.

4.12.35.4 Insert Component

The browse icon opens the Choose a File dialog box to allow you to select the source file of the component.

4.12.36 Compositions dialog box

The Compositions dialog box allows you to create, modify and delete compositions.



1 Composition category selection

2 New composition

3 In project

4 In library

5 Preview pane

6 Name

7 Type

8 Add ply

9 Duplicate ply

10 Structure grid

11 Properties grid

12 Custom properties

4.12.36.1 Composition category selection

Composition categories are used to filter the list of the compositions.

Click the drop-down button and choose one of the available categories: All, Generic, Roof, Slab, and Wall.

4.12.36.2 New composition

Click this button to create a new composition.

4.12.36.3 In project

Displays the compositions of the selected category in the Project database.

Right-click on a composition to add a new composition, duplicate, delete or copy to library.

Nota: Compositions that are used in the project cannot be deleted from the project database.

4.12.36.4 In library

Displays the compositions of the selected category in the Library database.

Right-click on a composition to add a new composition, duplicate, delete or copy to project.

4.12.36.5 Preview pane

Displays the preview of the selected composition.

4.12.36.6 Name

Displays the selected composition name. You can change the default name.

4.12.36.7 Type

Displays the category for the selected composition. You can click the drop-down button and select a new category. The available categories are: Generic, Roof, Slab, and Wall.

4.12.36.8 Add ply

Opens the Physical Materials dialog box which allows you to choose a material from the library or project database.

4.12.36.9 Duplicate ply

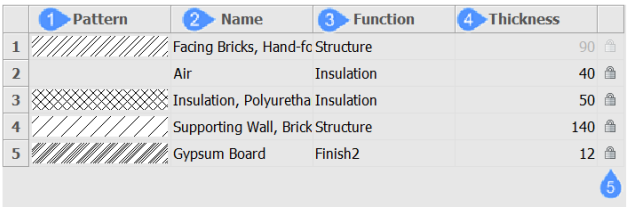
Inserts a copy of the selected ply.

4.12.36.1 Structure grid

0

Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.

The plies of a composition are ordered from Exterior (top) to Interior (bottom). You can drag the ply number on the desired position to change the ply order.



1 Pattern

2 Name

3 Function

4 Thickness

5 Lock/Unlock thickness

Pattern
Displays a preview of the ply. You can double click on the Pattern field and change its appearance from the Physical Materials dialog box.

Name
Displays the name of the ply. You can double click on the Name field and change it from the Physical Materials dialog box.

Function
You can select a function from the drop-down list. The available functions are: None, Structure, Substrate, Insulation, Finish1, Finish2, and Membrane.

Thickness
Click on the Thickness field to modify the thickness of a ply.

Nota: You can only modify the ply thickness if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

Lock/Unlock thickness
You can click the Lock/Unlock button to change its status.

Nota: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed**: all plies have locked thicknesses.
- **Minimal**: the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
- **Free**: the composition contains a single ply that has an unlocked thickness.

Nota: The thickness of a ply can only be unlocked if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

4.12.36.1 Properties grid

1
Displays the properties of the selected composition like tags, comments, description, layer, thickness and custom.

Tags
Select the Tags field and type new tags, separated by commas. Tags are case sensitive.
Click the Browse button next to the tags field to open the Tags dialog box.

Comments
Optionally, you can add comments to the selected composition.

Description
Optionally, you can add a description to the selected composition.

Layer
Optionally, you can assign a layer from the drop-down list to the selected composition.

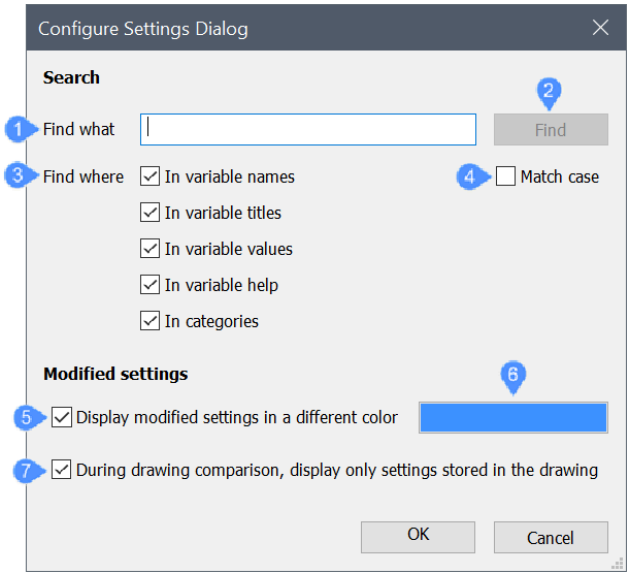
Thickness
Displays the overall thickness of a composition, which is equal to the sum of the thicknesses of the composing plies.

4.12.36.1 Custom properties

2
Opens the Select/Create a Custom Property dialog box. You can add custom properties for compositions.

4.12.37 Configure Settings dialog box

The Configure Settings dialog box allows you to control how BricsCAD searches for variables and how the differences are handled.



1 Find what

2 Find

3 Find where

4 Match case

5 Display modified settings in a different color

6 Select color

7 During drawing comparison, display only settings stored in the drawing

4.12.37.1 Find what

Type the name of the setting you are looking for.

4.12.37.2 Find

Searches in the Settings dialog box for the word you have typed.

4.12.37.3 Find where

Determines which part of the Settings dialog box should be searched.

In variable names
Thick this checkbox to search in variable names.

In variable titles
Thick this checkbox to search in variable titles.

In variable values
Thick this checkbox to search in variable values.

In variable help
Thick this checkbox to search in variable help.

In categories
Thick this checkbox to search in categories.

4.12.37.4 Match case

Thick this checkbox to specify that the capitalization of the searched phrase must match that of the words in the Settings dialog box.

4.12.37.5 Display modified settings in a different color

Thick this checkbox to see the modified settings in a different color.

4.12.37.6 Select color

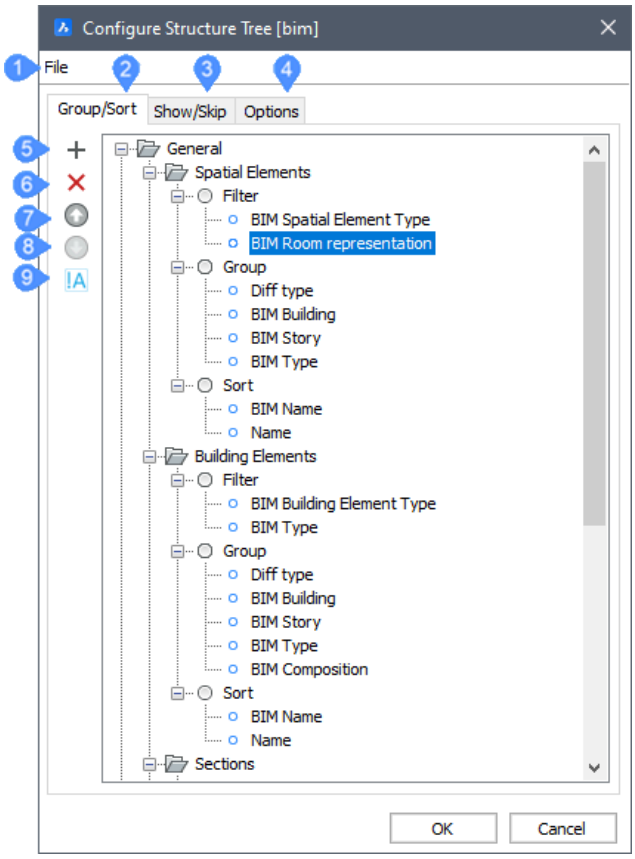
Click this button to select another color.

4.12.37.7 During drawing comparison, display only settings stored in the drawing

Thick this checkbox to see the differences for settings stored in the current drawing and another drawing that you open by running the *DWGCOMPARE* command.

4.12.38 Configure structure tree dialog box

The Configure structure tree dialog box allows you to define the tree structure of the Structural panel.



1 File

2 Group/Sort Tab

3 Show/Skip

4 Options

5 Add

6 Delete

7 Move Up

8 Move Down

9 Toggle negation

4.12.38.1 File

Click the File menu in the dialog box and choose one of the available options on the list:

- Open...: opens the Select Structure Tree Configuration File dialog box to select another *.cst configuration file.
- Save: saves the current configuration and keep the file dialog open.
- Save as: saves the current configuration under a different name and keep the file dialog open.

4.12.38.2 Group/Sort Tab

Add (+) (5)
Adds a new rule.

- 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 Rule, then click the name or right-click and choose Rename in the context menu and type a new name for the rule.
- Select the Filter node of the rule, then click the Add icon (+) or right-click and choose Add filter property in the context menu.
- Select the Group node of the rule, then click the Add icon (+) or right-click and choose Add Grouping Property in the context menu.
- Select the Sort node of the rule, 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 filter / grouping / sorting properties.

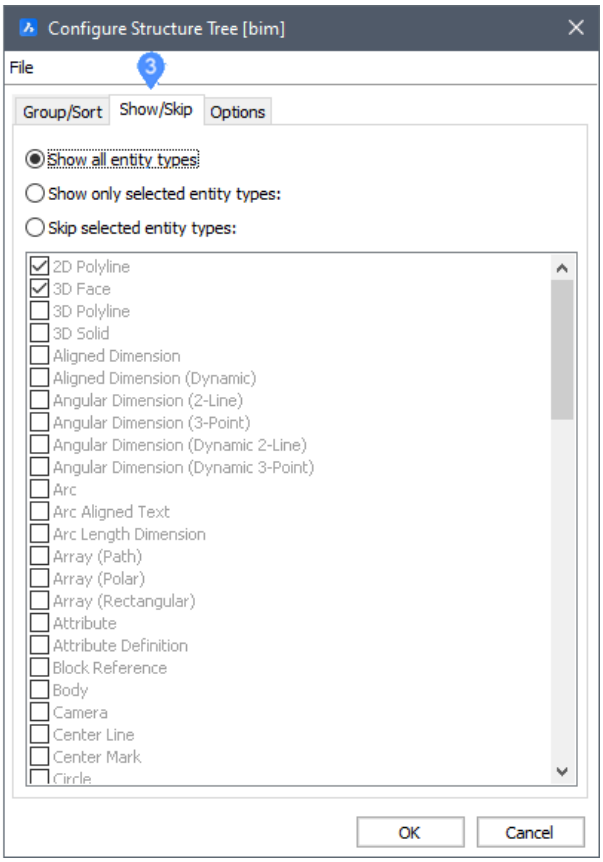
Delete (✕) (6)
Deletes a selected rule or property.

Move Up (▲) (7)
Moves Up a selected rule or property.

Move Down (▼) (8)
Moves Down a selected rule or property.

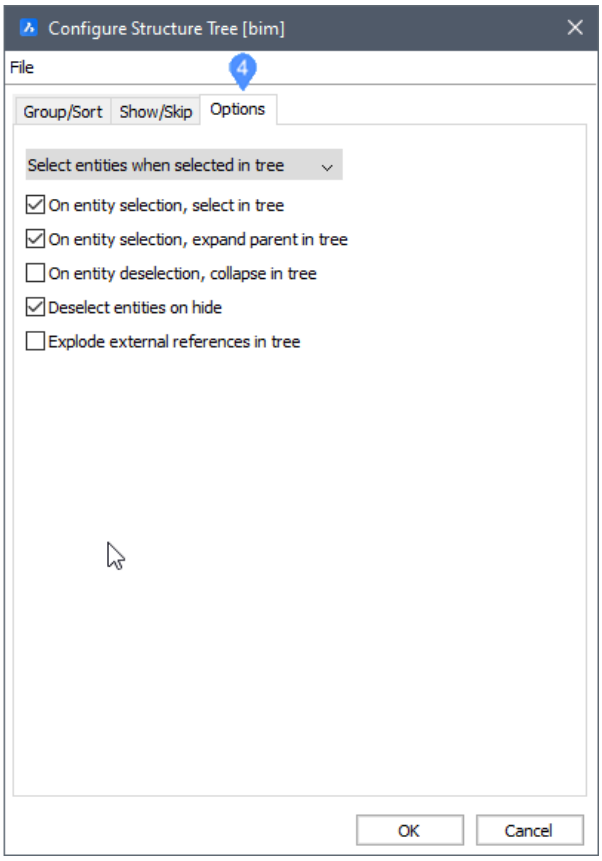
Toggle negation (9)
Toggle negation only for filter properties.

4.12.38.3 Show/Skip Tab



Select an option and then select the entity types to show or to exclude.

4.12.38.4 Options Tab

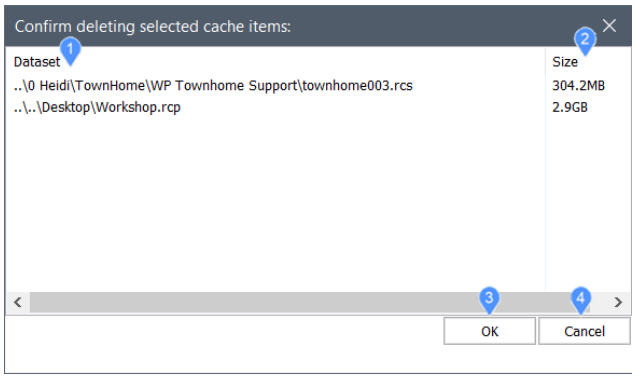


- Choose what happens in the model when entities are selected in the structure tree.
- Select what happens in the structure tree when an entity is selected/deselected in the model.

4.12.39 Confirm deleting selected cache items

TheConfirm deleting selected cache items dialog box enables you to review a list of cached point cloud files that were selected to be deleted.

This gives you a final opportunity to cancel the operation before permanently removing them from the cache.



1 Dataset

2 Size

3 OK

4 Cancel

4.12.39.1 Dataset

Displays a list of cached point cloud files that you selected to delete from the cache. It includes their names and locations.

4.12.39.2 Size

Displays the amount of cache used for each of the selected point cloud files.

4.12.39.3 OK

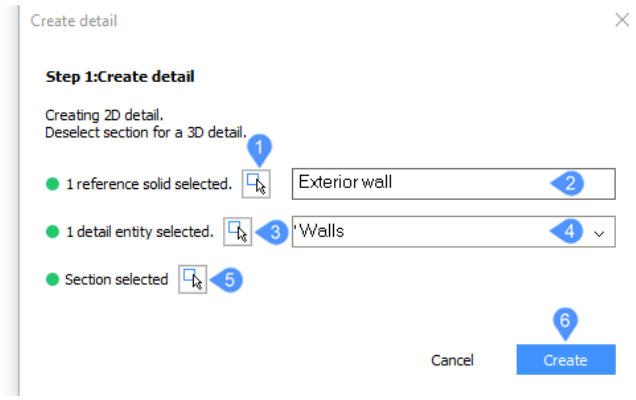
Deletes the point cloud files from the cache. Any drawings that reference these files will be unable to find them.

4.12.39.4 Cancel

Closes the dialog box without deleting point cloud files from the cache.

4.12.40 Create detail dialog box

The Create Detail dialog box allows you to create details and save them in the library.



1 Pick reference solids

2 Detail name

3 Pick detail entities

4 Create or choose category

5 Pick section

6 Create

4.12.40.1 Pick reference solids

Select reference solids, press Enter to display the dialog box and continue the procedure. These will not be copied but used to match other locations.

4.12.40.2 Detail name

Type the name you want to give to the detail.

4.12.40.3 Pick detail entities

Select detail objects (such as solids, block references, faces and edges) to be copied.

4.12.40.4 Create or choose category

Select a category in the drop-down list or type a name to create a new category.

4.12.40.5 Pick section

Select a BIM section.

Detail categories are subfolders of the Details folder, which is defined by the DETAILSPATH system variable. By default, this is *C:\ProgramData\Bricsys\Details*.

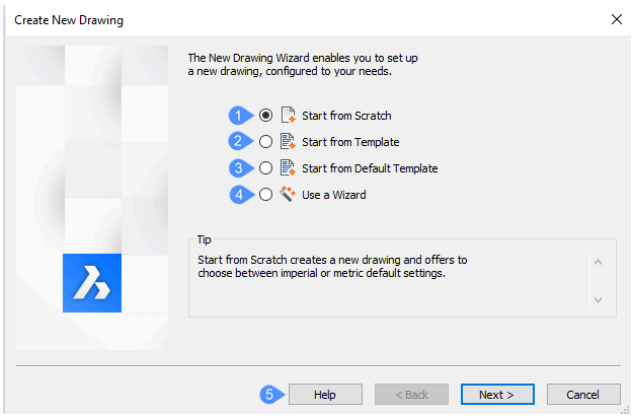
4.12.40.6 Create

Accept/Reject the changes or inflate first.

Displays the Save Detail dialog box, where you will add tags to the details in order to find it easier in the Details panel.

4.12.41 Create new drawing dialog box

The Create New Drawing dialog box allows you to create new drawings by several methods with the New Drawing Wizard.



1 Start from Scratch

2 Start from Template

3 Start from Default Template

4 Use a Wizard

5 Help

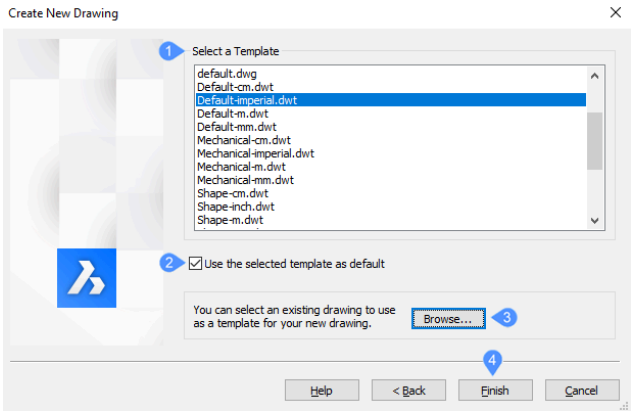
4.12.41.1 Start from Scratch

Starts the new drawing using the default template file specified by the BASEFILE system variable. Click the **Next** button and 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.

After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file.

4.12.41.2 Start from Template



1 Select a Template

2 Use default template

3 Browse...

4 Finish

Select a Template

Allows you to choose a file to act as the template. From the Template list, 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\V19x64\en_US\Templates*.

Use default template

Turn on **Use the selected template as default** option when you want to use this file as the template from now on. If so, then the program updates the value of the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

Browse...

Opens the Select a template file dialog box from where you can select the following kind of file type for a template: DWG - drawing files, DWT - template files, DXF - drawing interchange files.

Finish

Click the Finish button after selecting a Template from the list or from the dialog box.

4.12.41.3 Start from Default Template

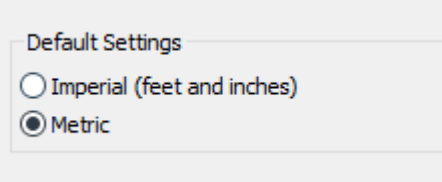
Starts new drawings with the default template defined by the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

4.12.41.4 Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way:

1 Click the **Next** button.

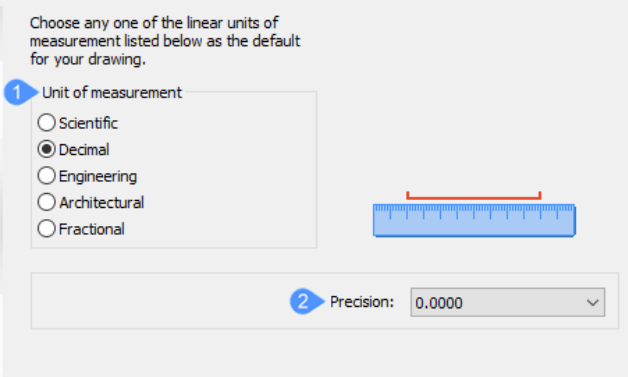
2 Choose a measurement system:



- 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.

3 Click the **Next** button.

4 Select a linear unit of measurement and display precision.



a Unit of Measurement

b Precision

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 command.

Unit Name	Measures	Example Display
Architectural	Feet, inches, fractional inches	4'-6 1/16"
Engineering	Feet, inches, decimal inches	4'-6.0625"
Fractional	Inches, fractional inches	54 1/6"
Decimal	(default) Units and decimal units	2128.4449
Scientific	Units, decimal units, exponents	2.1284E+03

Precision

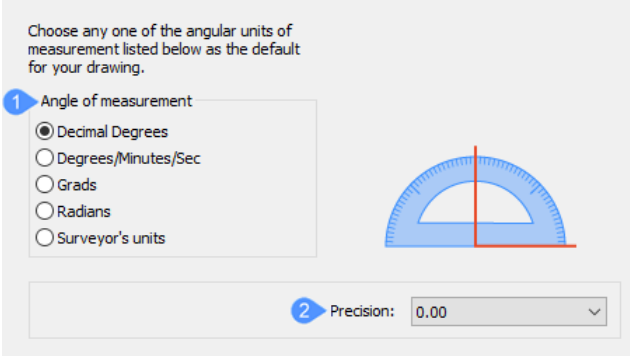
Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the Units command.

Nota: This is a display precision: the program makes calculations internally always to eight decimal places.

Nota:

1 Click the **Next** button.

2 Select an angular unit of measurement and display precision.



a Angle of measurement

b Precision

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 command.

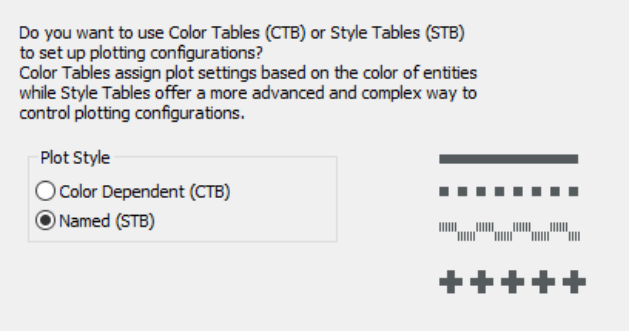
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds	Degrees,minutes,seconds,decimal secs	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units	North/South degrees towards East/West	N12d34'56"E

Precision

Click the drop-down list 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 command.

1 Click the **Next** button.

2 Determine which kinds of plot style the drawing should employ:

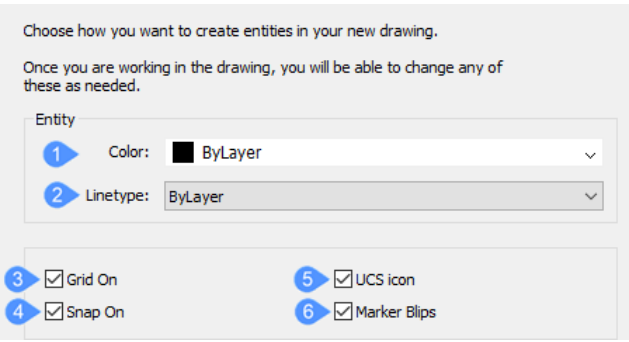


- **Color Dependent (CTB)** - these plot styles are simpler to employ, but less flexible.
- **Named (STB)** - plot styles are more flexible, but complex, because they control every aspect of every entity property during plotting.

If you are unsure, choose Color-dependent.

3 Click the **Next** button.

4 Choose the default working color, the linetype and the visual aids you want to be displayed by the drawing:



a Color
b Linetype
c Grid On
d Snap On
e UCS icon
f Marker Blips

Color

You can choose any color from the drop-down list. However, this is not recommended. Choose ByLayer so that layers control the colors of entities. See COLOR and LAYER commands.

Linetype

You can choose any linetype from the drop-down list, this again is not recommended. Choose ByLayer, for the same reason. See LINETYPE command.

Grid On

When on, displays a grid of dots (or lines) that helps you see the unit spacing, as well as drawing extents, such as every 1m; see the GRID command. It is recommended you turn this on.

Snap On
When on, sets the movement increment of the cursor, such as every 0.5m; see SNAP command. It is recommended you turn this on.

UCS icon
When on, displays the UCS icon for showing the orientation of the x, y, and z axes; see the UCSICON command. It is recommended that you turn this on for 3D drawings, but turn it off for 2D ones.

Marker Blips
When on, draws a tiny + each time you click in the drawing; see the BLIPMODE command. It is recommended you leave this one off, as it can turn the screen messy.

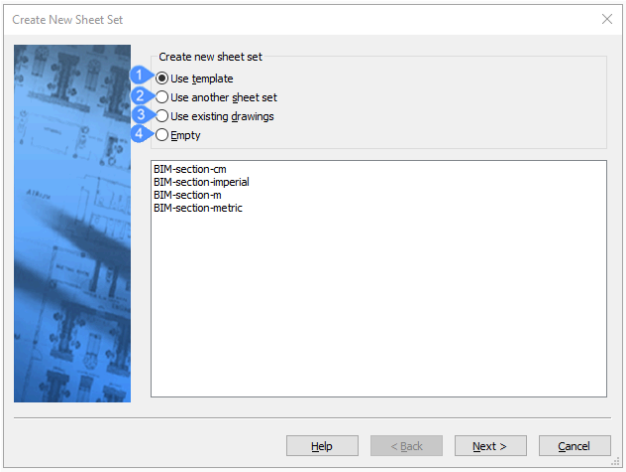
After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file and the options you choose during the wizard.

4.12.41.5 Help

Opens the Bricsys Help article about the NEWWIZ command.

4.12.42 Create new sheet set dialog box

The Create New Sheet Set dialog box provides 4 methods to create a new sheet set.



1 Use template

2 Use another sheet set

3 Use existing drawings

4 Empty

4.12.42.1 Use template

Use template
Create a new sheet set from a template.

Sheet set templates are stored as DST files in the folder set by the SheetSetTemplatePath preference (default path is *C:\Users\<UserName>\AppData\Local\Bricsys\BricsCAD\V[Version]\en_US\Templates*)

1 Select a template sheet set from the list shown in the dialog box.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Description

1 Optional, type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

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.

4.12.42.2 Use another sheet set

Creates a new sheet set from the settings in an existing sheet set.

Select sheetset to copy

1 Click **Browse** and select an existing sheetset file from Select a Sheet Set file dialog box.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder that stores the new file.

Description

1 Optional, type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

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.

4.12.42.3 Use existing drawings

Create a new sheet set from an existing set of drawings.

1 Select **Use existing drawings**.

2 Click **Next**.

Select Folder

1 Select a folder from the Choose a folder dialog box where the new sheet set will be stored.

Nota: Notice that the dialog box fills with names of drawings and their layouts.

2 Choose among the options (it is recommended 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

1 Add the drawing file's name to the start of the sheet set titles, which makes it easier to track the source of the files.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder.

Description

Optional, type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the Select Drawing Template dialog box.

2 Click **Next**.

Sheet Set Preview

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.

4.12.42.4 Empty

Creates a new sheet set from scratch.

1 Select **Empty**.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder.

Description

Optionally, type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the *Select Drawing Template* dialog box.

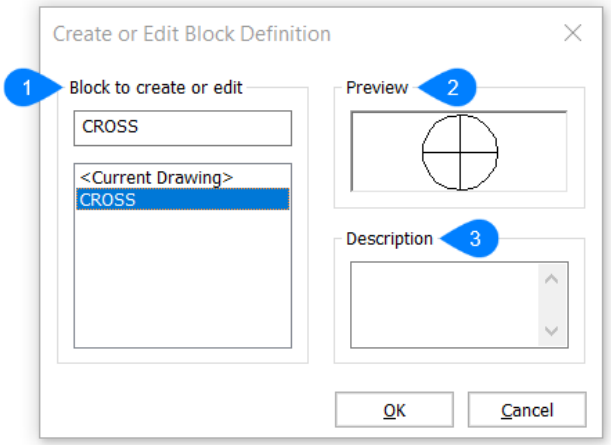
2 Click **Next**.

Sheet Set Preview

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.

4.12.43 Create or edit block definition dialog box

The Create or Edit Block Definition dialog box allows you to select an existing block definition to edit, or to enter a new name to create a new block definition.



1 Block to create or edit

2 Preview

3 Description

4.12.43.1 Block to create or edit

Specifies the name of the block you want to create or edit. To edit a block, you can select a block from the list below with the blocks available in the drawing.

4.12.43.2 Preview

Provides a preview of the selected block.

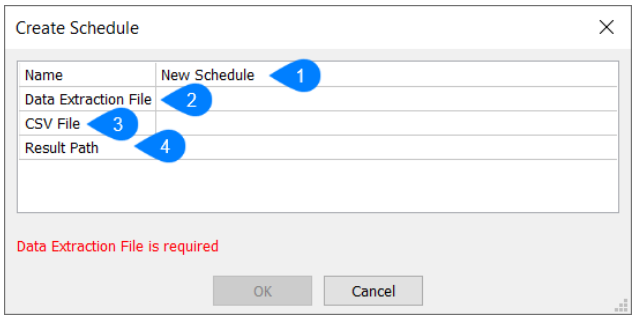
4.12.43.3 Description

Displays the description of the selected block.

Click **OK**, to open the Block Editor.

4.12.44 Create Schedule dialog box

Create a new schedule in the project browser.



1 Name

2 Data Extraction File

3 CSV File

4 Result Path

4.12.44.1 Name

Specifies the name of the new schedule.

4.12.44.2 Data Extraction File

Selects the data extraction file that will be used. Opens the Select Data Extraction Definition file dialog box where you can select a file.

4.12.44.3 CSV File

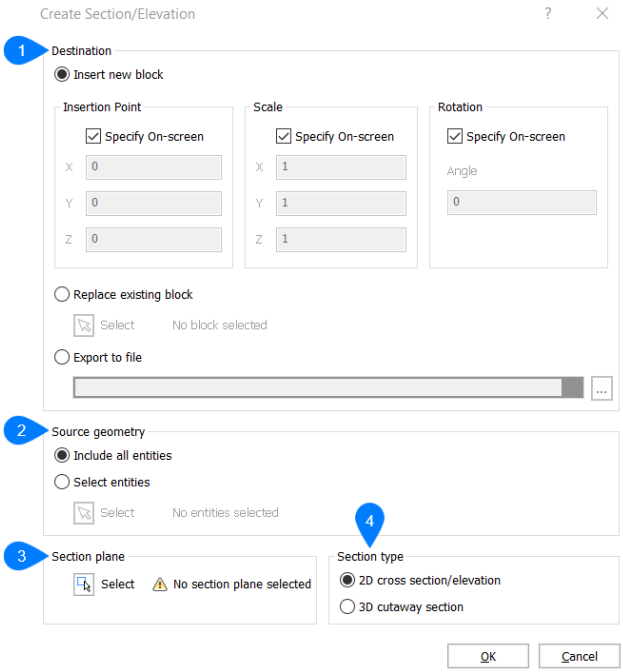
Defines the CSV file that will be used. Opens the Select Delimiter-separated format file dialog box where you can select a file.

4.12.44.4 Result Path

Defines the path where the resulting drawing will be saved. Opens the Select Drawing file dialog box where you can select a drawing to save the result.

4.12.45 Create section/elevation dialog box

The Create Section/Elevation dialog box saves section planes as blocks inserted in the current drawing or saved to disk as DWG files.



1 Destination

2 Source geometry

3 Section plane

4 Section type

4.12.45.1 Destination

Defines where the block will be placed in the drawing.

Insert new block
Inserts the created section as a new block in the drawing. The insertion point, scale and rotation can be specified.

Insertion point
Allows to specify the insertion point in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z
Defines the coordinates where the block should be inserted, only available if the Specify on-screen button is checked off.

Scale
Allows to specify the scale factor in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z
Defines the individual X,Y,Z direction scale factors, only available if the Specify on-screen button is checked off.

Rotation
Allows to specify the rotation angle in the model space after you click the OK button, if the Specify on-screen button is checked.

Angle
Defines the rotation angle of the inserted section/elevation.

Replace existing block
Replaces an existing block in the drawing. This block will be replaced by the created section

Select
Allows to select the block to replace in the model space.

Export to file
Opens the Select export file dialog box. Here you can select a file to export the created section to.

4.12.45.2 Source geometry

Specifies the entities that will be included from the drawing.

Include all entities
Selects all 3D geometry in the drawing (includes 3D solids, 3D surfaces, polyface meshes, and 3D faces).

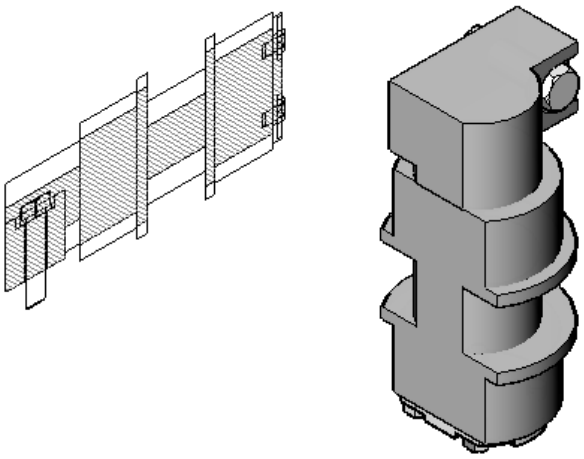
Select entities
Allows to select the entities to be included in the model space.

4.12.45.3 Section plane

Selects the section plane to be converted to a block.

4.12.45.4 Section type

Determines the kind of section plane block to create.

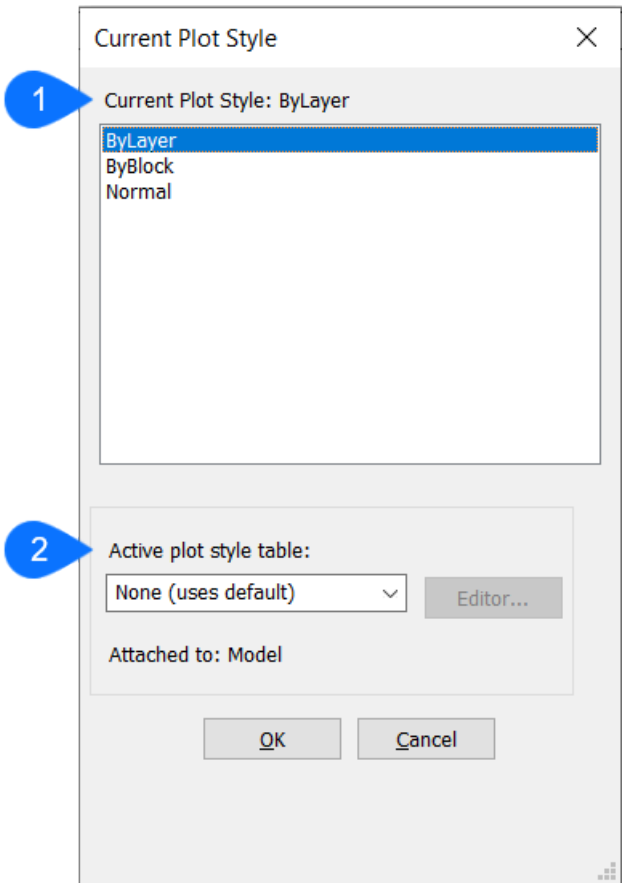


2D cross section/elevation
Uses the 2D section settings defined by the SectionPlaneSettings dialog box to create the block.

3D cutaway section
Uses the 3D section settings defined by the SectionPlaneSettings dialog box to create the block.

4.12.46 Current Plot Style dialog box

Sets the current plot style.



1 Current Plot Style

2 Active plot style table

4.12.46.1 Current Plot Style

Sets the current plot style for the drawing. You can choose between:

- **ByLayer**: uses the plot style of the layer of the entity.
- **ByBlock**: uses the plot style of the block the entity is assigned to.
- **Normal**: uses the normal plot style. The default properties of the entity are used.

4.12.46.2 Active plot style table

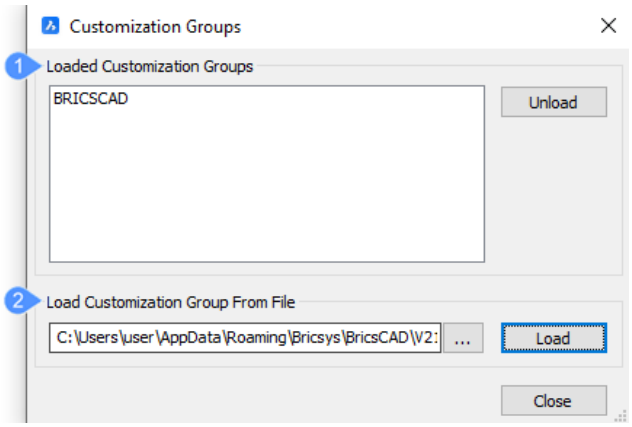
Lists the names of STB files (short for "style table"). To edit the chosen plot style table, click 'Editor...'. The Plot Style Table Editor dialog box will be displayed.

4.12.47 Customization groups dialog box

The **Customizations Groups** Dialog Box allows you to load or to unload a partial CUI file.

When you install BricsCAD the default CUI files are copied to the **Support** folder of your Roamable root folder. (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.



1 Loaded Customization Groups

2 Load Customization Group from File

4.12.47.1 Loaded Customization Groups

A list with the names of the customization files already loaded is displayed.

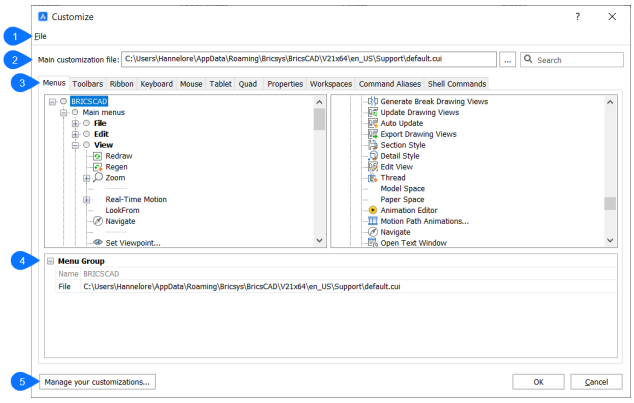
A CUI file can be unloaded by selecting it from the list and clicking the **Unload** button.

4.12.47.2 Load Customization Group from File

Click the browse button to select a partial CUI file and click the **Load** button to load it.

4.12.48 Customize dialog box

The Customize dialog box allows you to customize the user interface of BricsCAD.



1 File

2 Main customization file

3 Tab options

4 Menu Group

5 Manage your customizations

4.12.48.1 File

Shows a menu to select customization files or import workspaces.

Load main CUI file...
Opens the Select main CUI file dialog box to select a customization file. See the CUILOAD command.

Save main CUI file as...
Opens the Save main CUI file as dialog box to save the current main CUI file.

Nota: You can save the main CUI file under a different name to copy the adapted main CUI file.

Load Partial CUI file...
Opens the Choose a customization file dialog box to open a partial CUI file.

Nota: See chapter **Loading a partial CUI file** in **Loading and unloading CUI files** for a detailed explanation about the partial CUI files.

Create new partial CUI file...
Opens the Create a Customization File dialog box to create an empty CUI file.

Import workspace(s)…
Opens the Choose a Customization File dialog box to load one or more workspaces from another CUI file.

4.12.48.2 Main customization file

Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface.

Nota: Aliases and shell commands are defined in a PGP file.

Browse
Opens the Select main CUI file dialog box to load a different main CUI file.

Search
Searches the Available Tools list for command names.

4.12.48.3 Tab options

Menus
Adds, edits, moves and deletes menu items.
When right clicking a menu, menu item or submenu in the Menus tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append item or Insert item from the context menu, the Add menu item dialog box displays.

Menu Group/Menu/Menu Item/Submenu
Displays the options of the selected menu, submenu, menu item or separator.

Title
Specifies the name that is displayed by the menu, menu item or submenu.

Nota: You can prefix a letter with '&' to create an Alt-key shortcut. For example: &Line.

ID
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the menu, menu item or submenu. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

Diesel
Specifies the Diesel code to be employed by the menu, menu item or submenu.

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 for the menu, menu item or submenu. The browse icon opens the Tool Image dialog box. There are five options to select an image.

Toolbars
Adds, Moves and deletes toolbar items.
When right clicking a toolbar, tool, control, flyout or separator in the Toolbars tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append tool or Insert tool from the context menu, the Add tool dialog box displays. For options description see the Add menu item dialog box.

Menu Group/Toolbar (Button)
Displays the options of the selected toolbar, tool, control, flyout or separator.

Title
Specifies the name that is displayed by the tooltip.

ID
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

Position

Default Display
Determines whether the selected item will be added to the workspace.

Rows
Specifies the number of rows for an undocked toolbar.

Xval Yval
Specifies the distance in pixels measured from the top left corner of the screen to the toolbar. It applies to undocked toolbars.
The values are taken from the X,Y options of the toolbar as they are set in the Workspace tab.

Diesel
Specifies the Diesel code to be employed by the selected item.

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 display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

Ribbon
Manages the ribbons and/or adds panels to a ribbon tab.
When right clicking a ribbon tab or panel in the Ribbon tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add launcher, Insert toggle button or Insert command button from the context menu, the Add ribbon dialog launcher dialog box, Add ribbon toggle button or add ribbon command button displays.

- Select available tool: assigns an existing command to the new item. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new item. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop down list
- Title : specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

Menu Group/Ribbon Tab/Ribbon Panel Reference
Displays the options of the selected ribbon tab or panel.

ID
Identifies the item in the CUI file. (The ID is assigned by BricsCAD).

Collapse
Allows to choose if the panel will collapse automatically or if it will never collapse.

Label
Specifies the name of the ribbon tab or panel.

Title
Specifies the name that is displayed by the selected item.

Key Tip

Keyboard
Assigns different keyboard shortcuts to different commands.
When right clicking a keyboard shortcut in the Keyboard tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append shortcut or Insert shortcut from the context menu, the Add keyboard shortcut dialog box displays.

- Select available tool: assigns an existing command to the new keyboard shortcut. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the shortcut. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

Menu Group/Keyboard Shortcut
Displays the options of the selected toolbar, tool, control, flyout or separator.

Key
Specifies the used keyboard shortcut.

Tool ID
Identifies the keyboard 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, 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
Modifies the different actions that are connected to the mouse buttons.
When right clicking a button or button group in the Mouse tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append button or Insert button from the context menu, the Add button item dialog box displays.

Alias
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

Button
Specifies the used mouse button.

Tool ID
Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Title
Specifies the name that is displayed by the mouse button.

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

Image
Specifies the image to display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

Tablet
Customizes the tablet options.
When right clicking a button or button group in the Tablet tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: When you choose Append button or Insert button from the context menu, the Add button item dialog box displays.

Menu Group/Button Group

Alias
Defines the aliases of the selected item. The browse icon opens the ‘Edit Aliases’ dialog box where you can edit, create or delete aliases.

Quad
Adds commands to the quad tab or move commands to other tabs.
When right clicking a quad tab or quad button in the Quad tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append quad button or Insert from the context menu, the Add quad button dialog box displays.

- Select available tool: assigns an existing command to the new quad button. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new quad button. If this option is selected, Available tools is greyed out
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar

- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool

ID
Identifies the Quad item in the CUI file. (The ID is assigned by BricsCAD).

Title
Specifies the name that is displayed by the quad button.

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros related to the quad button.

Image
Specifies the image to display for the toolbar button. The browse icon opens the Tool Image dialog box. There are five options to select an image.

Entity Filter
Specifies which items will be displayed in the Quad, depending on the type of the entity.

Properties
Edits which properties are shown for different kinds of entities.

Nota: These properties will only be shown when the ROLLOVERTIPS system variable is ON.

Rollover
Specifies which properties should be displayed.

Workspaces
Controls which menu tabs are visible in different workspaces.
When right clicking an item in the Workspace tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Command Aliases
Customize aliases and add them to different commands.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add alias or Edit alias from the context menu, the Add alias or edit alias dialog box displays where you can add or edit an alias.

Alias
Specifies the name of the alias, a command abbreviation.

Nota: The list of Aliases and commands will be ordered alphabetically according to the aliases when you click on **Alias**.

Command
Specifies the command to which the alias is assigned.

Nota: The list of Aliases and commands will be ordered alphabetically according to the commands when you click on **Command**.

Add
Adds an alias.

Edit
Edits the selected alias.

Delete
Deletes the selected alias.

Commands
Lists all the available commands to add an alias to.

Shell Commands
Create and edit shell commands. Shell commands run programs external to BricsCAD.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add shell command or Edit shell command from the context menu, the Add shell command or edit shell command dialog box displays where you can add or edit a shell command.

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.

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*

Prompt
Specifies the prompt to display in the Command line, for example: "Enter file to edit:"

Add
Adds a shell command through a dialog box.

Edit
Edits the shell command through a dialog box.

Delete
Deletes the selected shell command. Gives a warning where you can confirm you to delete the selected item.

Options Within the Context Menu

Append …
Adds the mentioned item below the last item.

Insert ...
Adds the mentioned item above the selected item.

Add …
Adds the mentioned item in alphabetical order.

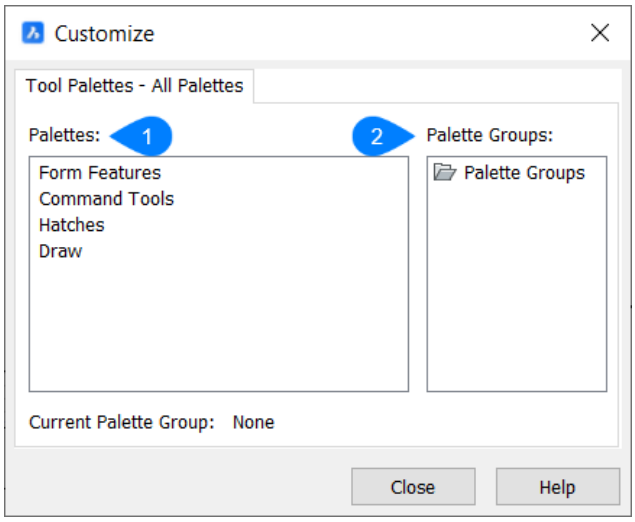
Delete ...
Deletes the menu, submenu, item or separator selected item. Gives a warning where you can confirm you to delete the selected item.

4.12.48.4 Manage your customizations

Displays the Manage Customizations dialog box for confirming and reversing changes made to the user interface.

4.12.49 Customize (Tool Palettes) dialog box

The Customize (Tool Palettes) dialog box allows you to customize your own tabs and tools to use in the tool palettes panel.



1 Palettes

2 Palette Groups

4.12.49.1 Palettes

Displays a list of all palettes available in the tool palettes panel. By right clicking on one of the palettes, several options may be available in the context menu.

Rename
Renames the Palette by entering the new name.

New Palette
Adds a new Palette to the list with a generic name that can be changed right away.

Delete
Deletes the selected palette

Export
Exports a palette through the Export Palette dialog box.

Import
Imports a palette through the Import Palette dialog box.

4.12.49.2 Palette Groups

Displays the loaded palette groups. By right clicking on one of the groups, several options may be available in the context menu.

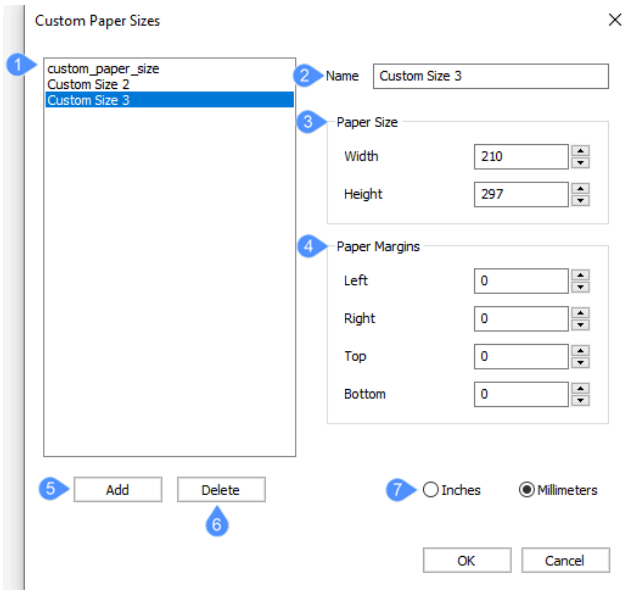
New group
Adds a new sub-group to the list with a generic name that can be changed right away.

> ***Nota:*** For the additional sub group, the available options in the context menu are the same as those for palettes.

Import
Imports a palette through the Import Palette Group dialog box.

4.12.50 Custom paper sizes dialog box

The Custom Paper Sizes Dialog Box allows you to create custom paper sizes when Print as PDF.pc3 is selected in the Printer/Plotter Configuration list.



1 Custom Paper Sizes list

2 Name

3 Paper Size

4 Paper Margins

5 Add

6 Delete

7 Units

4.12.50.1 Custom Paper Sizes list

Displays the list of all custom paper sizes created by the user.

4.12.50.2 Name

Allows you to name the custom paper size.

4.12.50.3 Paper Size

You can insert the width and the height of the page in the appropriate fields.

4.12.50.4 Paper Margins

You can set the paper margins by entering the desired values in the fields. Only positive values are allowed.

4.12.50.5 Add

Allows you to create the new custom paper size and adds it to the list.

4.12.50.6 Delete

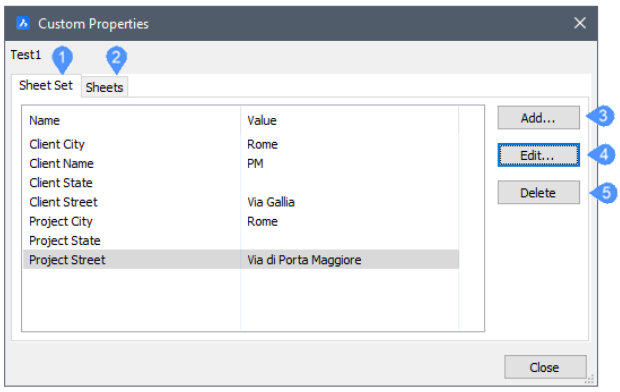
Allows you to select a custom paper size from the list and delete it. You can delete one item at a time.

4.12.50.7 Units

You can choose whether to use inches or millimeters for your custom paper size.

4.12.51 Custom properties dialog box

The **Custom Properties** dialog box allows you add custom properties for sheet set and sheets.



1 Sheet Set Tab

2 Sheets Tab

3 Add

4 Edit

5 Delete

4.12.51.1 Sheet Set Tab

Manages the custom sheet set properties.

4.12.51.2 Sheets Tab

Manages the custom sheets properties.

4.12.51.3 Add

Adds a custom property. Displays the **Add Custom Property** dialog box in which you add a property name and its initial value.

4.12.51.4 Edit

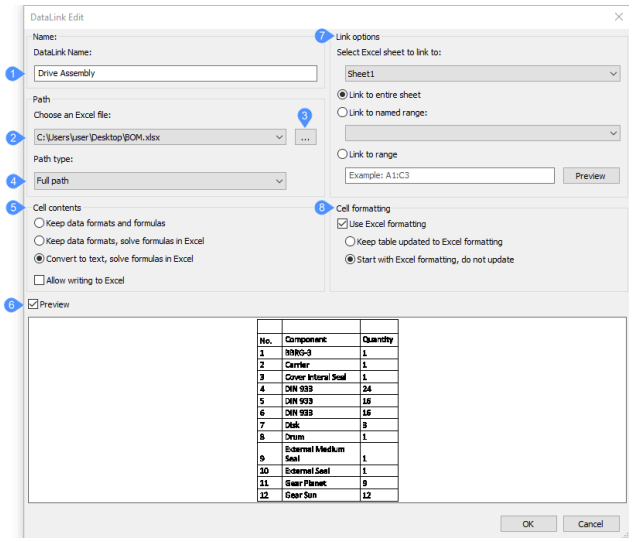
Allows to edit the value of a selected custom property. Displays the **Custom Sheet Property** or **Custom Sheet Set Property** dialog box in which you edit the name of the property.

4.12.51.5 Delete

Deletes a selected custom property.

4.12.52 DataLink edit dialog box

The **DataLink Edit** dialog box allows you to link data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Name

2 Path

3 Browse

4 Path type

5 Cell contents

6 Preview

7 Link options

8 Cell formatting

4.12.52.1 Name

You can type a name in the **DataLink Name** field.

4.12.52.2 Path

Displays you the path where your Excel file is located.

4.12.52.3 Browse

Allows you to select a data file. *.xls, *.xlsx and *.csv files are accepted.

4.12.52.4 Path type

Allows you to choose a path type option:

- **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.

4.12.52.5 Cell contents

- **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.

4.12.52.6 Preview

If this button is selected, a preview of the data file is displayed.

4.12.52.7 Link options

Allows you to select Excel sheet to link to:

- **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.

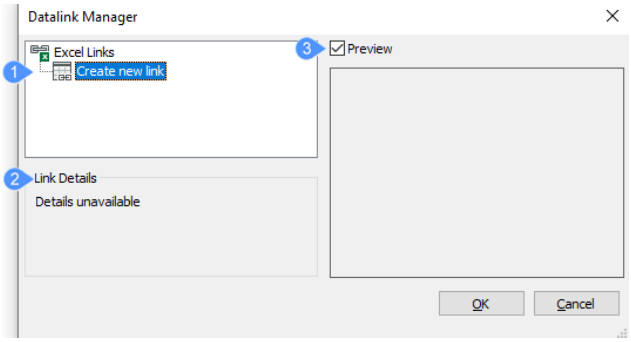
4.12.52.8 Cell formatting

You must select the **Use Excel formatting** option to be able to select one of the following options:

- **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.

4.12.53 Datalink manager dialog box

The **Datalink Manager** dialog box links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Create new link

2 Link details

3 Preview

4.12.53.1 Create new link

Opens the **DataLink Edit** dialog box.

4.12.53.2 Link details

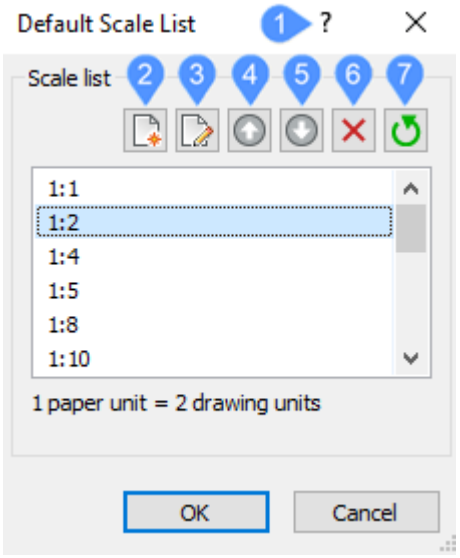
Displays link details.

4.12.53.3 Preview

Display a preview of the datalink.

4.12.54 Default scale list dialog box

The **Default Scale List** dialog box allows you to edit the list of scale factors displayed by commands, such as PRINT and PAGESETUP and by annotative scale factors.



1 Command reference

2 Add new scale

3 Edit scale

4 Move up

5 Move down

6 Delete

7 Reset

4.12.54.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

4.12.54.2 Add new scale

Allows you to add a new scale factor to the list. Opens the **Add Scale** dialog box.

4.12.54.3 Edit scale

Allows you to edit the factors of an existing scale. Opens the **Edit Scale** dialog box.

4.12.54.4 Move up

Moves the selected scale factor up the list.

4.12.54.5 Move down

Moves the selected scale factor down the list.

4.12.54.6 Delete

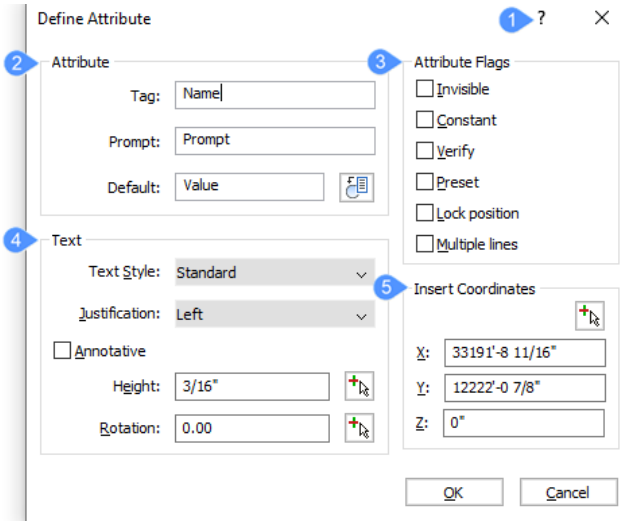
Deletes the selected scale factor(s). Press the Ctrl key to select and delete more than one factor at a time. Scale 1:1 cannot be deleted.

4.12.54.7 Reset

Returns the list to its original condition, adding back in the deleted factors and removing those that were added, but are not being used.

4.12.55 Define attribute dialog box

The **Define Attribute** Dialog Box defines the options for the attribute values.



1 Command reference

2 Attribute options

3 Attribute Flags options

4 Text options

5 Insert Coordinates options

4.12.55.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

4.12.55.2 Attribute options

- **Tag**

  Specify 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**

  Specify 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 then uses the tag as the prompt during attribute insertion.

- **Default**

  Specify the default value. This is displayed in angle brackets, like <360>. As with other defaults, the user only needs to press Enter to accept this value.

  Click the **Insert Field** icon to assign a field value to the attribute (see the FIELD command).

4.12.55.3 Attribute Flags options

- **Invisible**

  Hide the attribute from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the ATTDISP command.

- **Constant**

  Specify a default value that the user cannot change.

- **Verify**

  Forces to enter the value a second time; this helps to ensure that the value is entered correctly.

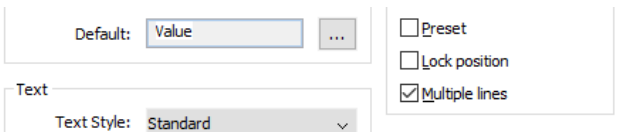
- **Preset**

  Inserts attributes without prompting the user; the attributes can be changes later with the ATTEDIT 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 to a single line.

  When the **Multiple lines** box is checked, the button icon displays next to the dimmed **Default** text field.



When you click the button, the **Define Attribute** dialog box temporarily closes to let you define a multiline text entity.

The MTEXT 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.

4.12.55.4 Text options

- **Text style**

  Specify 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 command.)

- **Justification**

  Specify the justification of the attribute text, such as **Center** or **Fit**.

- **Height**

  Specify the height of the attribute text, unless overridden by the text style. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Rotation**

  Specify the rotation angle of the rotation text. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.

- **Annotative**

  Specify the **Annotative** property of the attribute text.

4.12.55.5 Insert Coordinates options

- **Select insertion point**

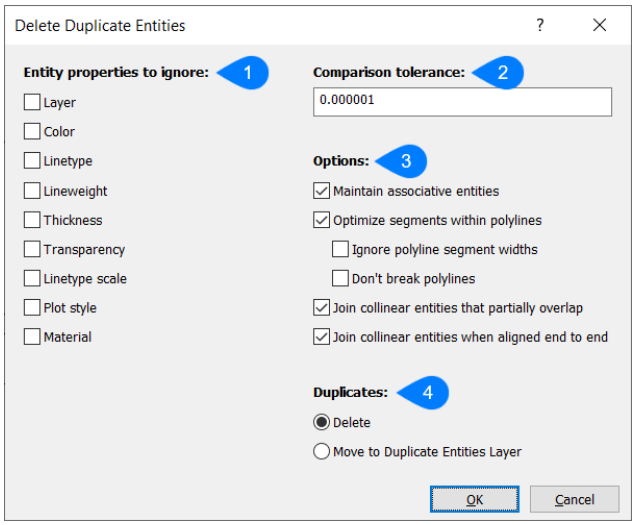
  Specify the insertion point directly in model.

- **X/Y/Z**

  Specify the coordinates of the attribute's insertion point.

4.12.56 Delete duplicate entities dialog box

The Delete Duplicate Entities dialog box allows you to delete duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones.



1 Entity properties to ignore

2 Comparison tolerance

3 Options

4 Duplicates

4.12.56.1 Entity properties to ignore

Selected properties are ignored when comparing duplicate or overlapping entities.

4.12.56.2 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.

4.12.56.3 Options

Determines how special cases are handled.

Maintain associative entities
Associative entities are not processed.

Optimize segments within polylines
Evaluates segments of selected polylines individually, duplicate vertices and segments are removed.

- **Ignore polyline segment widths**: ignores the Width property of overlapping polyline segments.
- **Don't break polylines**: polylines entities are kept intact.

Join collinear entities that partially overlap
Unifies partially overlapping entities into a single entity.

Join collinear entities when aligned end to end
Unifies collinear lines and polylines having coinciding endpoints into a single entity or segment.

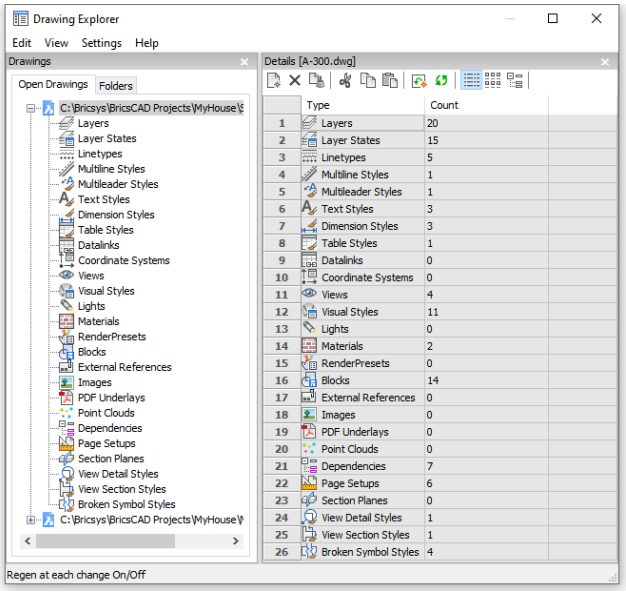
4.12.56.4 Duplicates

Determines what to do with duplicates: either delete them or move them to a Duplicate Entities layer.

4.12.57 Drawing explorer dialog box

4.12.57.1 Opening the Drawing Explorer dialog box

- Launch the EXPLORER command.
- Launch a definition category command, such as LAYERS or EXPBLOCKS.
- Choose a definition category in the **Tools** > **Drawing Explorer** menu.
- Choose **Drawing Explorer** in the **General** tab of the No Selection Quad.



Categories	Description
Layers	Reports the number of layers defined in the selected drawing. Click to explore the layers; see the LAYER command.
Layer States	Reports the number of layer states defined in the selected drawing. Click to explore the layer states; see the LAYERSTATE command.
Linetypes	Reports the number of linetypes loaded into the selected drawing. Click to explore the linetypes; see the LINETYPE command.
Multiline Styles	Reports the number of multiline styles defined in the selected drawing. Click to explore multiline styles; see the MLSTYLE command.
Multileader Styles	Reports the number of multileader styles defined in the selected drawing. Click to explore multileader styles; see the MLEADERSTYLE command.
Text Styles	Reports the number of text styles defined in the selected drawing. Click to explore text styles; see the STYLE command.

Categories	Description
Dimension Styles	Reports the number of dimension styles defined in the selected drawing. Click to explore dimension styles; see the DIMSTYLE command.
Table Styles	Reports the number of table styles defined in the selected drawing. Click to explore table styles; see the TABLESTYLE command.
Data Links	Reports the number of data in the drawing linked to external spreadsheet files. Click to explore data links; see the DATALINK command.
Coordinate Systems	Reports the number of named user coordinate systems (UCS) defined in the selected drawing. Click to explore coordinate systems; see the EXPUCS command.
Views	Reports the number of named views in the selected drawing. Click to explore saved views; see the VIEW command.
Visual Styles	Reports the number of visual styles defined in the selected drawing. Click to explore visual styles; see the VISUALSTYLES command.
Lights	Reports the number of lights placed in the selected drawing. Click to explore lights; see the LIGHTLIST command.
Materials	Reports the number of materials defined in the selected drawing. Click to explore materials; see the MATERIALS command.
Render Presets	Reports the number of render presets defined in the drawing. Click to explore render presets; see the RENDERPRESETS command.
Blocks	Reports the number of blocks defined in the selected drawing. Click to explore blocks; see the EXPBLOCKS command.
External References	Reports the number of external references (xrefs) attached to the selected drawing. Click to explore xrefs; see the XREF command.
Images	Reports the number of images attached to the selected drawing. Click to explore the images; see the IMAGE command.
PDF Underlays	Reports the number of PDF underlays attached to the selected drawing. Click to explore PDF underlays; see the PDF command.
Point Clouds	Reports the number of point clouds attached to the selected drawing. Click to explore point cloud; see the POINTCLOUD command.

Categories	Description
Dependencies	Reports the number of external files that the selected drawing depends on. Click to explore dependencies; see the ETRANSMIT command.
Page Setups	Reports the number of the page setups attached to the selected drawing. Click to explore page setups; see the PAGESETUP command.
Section Planes	Reports the number of section planes defined in the selected drawing. Click to explore the section planes; see the SECTIONPLANESETTINGS command.
View Detail Styles	Reports the number of view detail styles defined in the selected drawing. Click to explore the view detail styles; see the VIEWDETAILSTYLE command.
View Section Styles	Reports the number of view section styles defined in the selected drawing. Click to explore the view section styles; see the VIEWSECTIONSTYLE command.
Broken Symbol Styles	Reports the number of available styles for the VIEWBREAK command. Click to explore the broken symbol styles.

4.12.57.2 The Drawing Explorer layout

The **Drawing Explorer** window consists of a number of sub-windows or panes.

You can select which panes you want to have open or closed in the **View** menu. BricsCAD remembers the display settings for each category.

Item	Description
Drawing	Displays the **Drawing** pane. • **Open Drawings**: a list of all drawings that are currently open. • **Folders**: - **Local Folders**: your favorite local drawing folders. - **Bricsys 24/7**: your drawing folders on Bricsys 24/7.
Details	Displays the **Details** pane. Shows the details of the selected drawing or the details of the selected category.
Preview	Displays the **Preview** pane. Shows a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material.
Editor	Displays the **Editor** pane. Allows you to define or edit the properties of the selected category.

Item	Description
Status bar	Displays the Status bar at the bottom of the **Drawing Explorer** dialog box.

	View options for the Details pane
Details	Displays the details of definitions in a grid.
Icons	Displays definitions as an array of icons; applies only to blocks and external references.
Tree	Displays definitions 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.
Regen at each change	Toggles the automatic updating of the display at each change.

Hide Xref symbols	Do not display definitions in Xrefs.

Nota:

- If you click a definition category of an open drawing in the **Drawings** sub-window when the **Details** sub-window is closed, the **Details** window is re-opened, showing the content of the selected 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.

4.12.57.3 The Settings menu

Item	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.

Item	Description
Options	Applies to Blocks and External References only. Specifies default settings for inserting blocks. See the EXPBLOCK command.

4.12.57.4 The Details toolbar

Icon	Name	Shortcut	Description
	New	Ctrl + N	Creates a new definition with a generic name, such as NewLayer 1 or NewStyle1. Use the Rename button to rename it.
	Delete	Ctrl + D	Erases the selected definition 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. Some items cannot be erased, such as layer 0, Global material, visual style 2D Wireframe, …
	Purge	Ctrl + P	Purges unused definitions from the drawing. Some definitions cannot be erased, such as layer 0 and text style Standard. See the PURGE command.
	Cut	Ctrl + X	Copies the selected definition(s) to the Clipboard, and then deletes it from the drawing following the same procedure as the Delete tool.
	Copy	Ctrl + C	Copies the definition(s) to the Clipboard for pasting elsewhere in the Drawing Explorer.
	Paste	Ctrl + V	Pastes the definition(s) from the Clipboard, 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: • **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
	Details View		Displays the details of definitions in a grid.
	Icon View		Displays definitions as an array of icons; applies only to blocks and external references.

Icon	Name	Shortcut	Description
	Tree View		Displays definitions 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 the automatic updating of the display at each change.

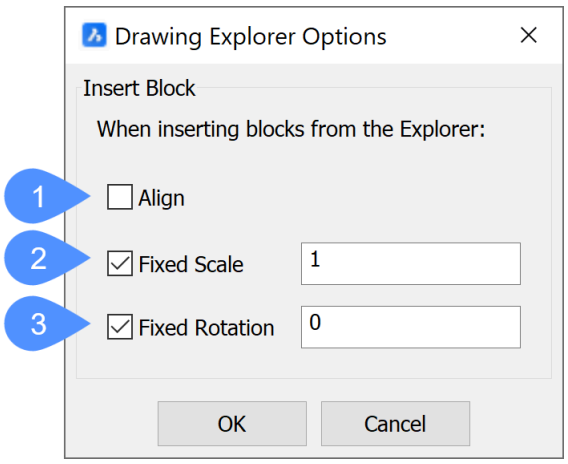
4.12.57.5 The Context menu

Select a definition, then right-click.

Item	Shortcut	Description
New	Ctrl + N	Creates a new definition as a copy of the selected definition.
Delete	Ctrl + D	Deletes the selected definitions. Definitions that are in use cannot be deleted.
Purge	Ctrl + P	Deletes unused definitions in the current selection.
Cut	Ctrl + X	Cuts the selected definitions to the clipboard.
Copy	Ctrl + C	Copies the selected definitions to the clipboard.
Paste	Ctrl + V	Pastes definitions from the clipboard.
Rename	F2	Renames the selected definition; alternatively, click on the name twice, slowly, to rename it.
Select All	Ctrl + A	Selects all definitions in the current category.
Invert Selection		Inverts the selected definition(s), e.g. select a few layers, then use this to select all other layers.
Current 		Makes the definition current, such as making the selected layer the current (working) layer.

4.12.58 Drawing explorer options dialog box

The **Drawing Explorer Options** dialog box allows you to align or set the scale and the rotation of the inserted blocks.



1 Align

2 Fixed Scale

3 Fixed Rotation

4.12.58.1 Align

Tick the checkbox to align blocks with the snapped entity.

4.12.58.2 Fixed Scale

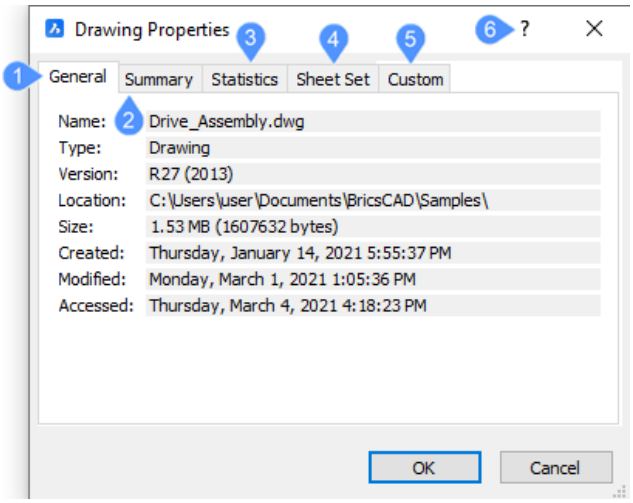
Tick the checkbox to specify the scale factor to be used for all block insertions.

4.12.58.3 Fixed Rotation

Tick the checkbox to specify the rotation angle to be used for all block insertions.

4.12.59 Drawing properties

The **Drawing Properties** dialog box allows you to see general information about the drawing and user-defined properties.



1 General

2 Summary

3 Statistics

4 Sheet Set

5 Custom

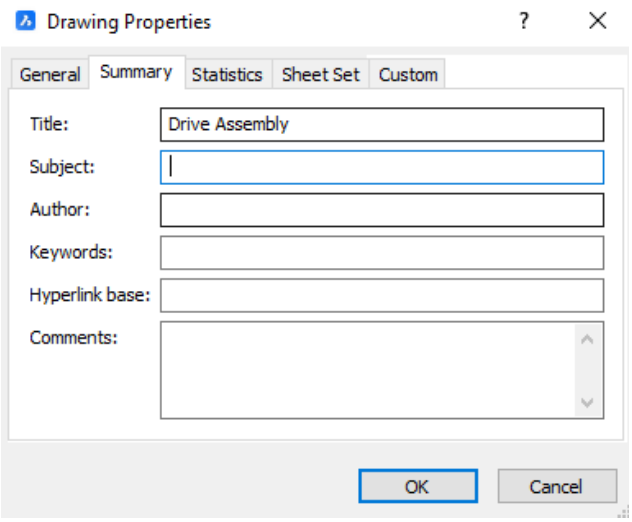
6 Command reference

4.12.59.1 General

Displays the general properties of the drawing, such as creation date and time.

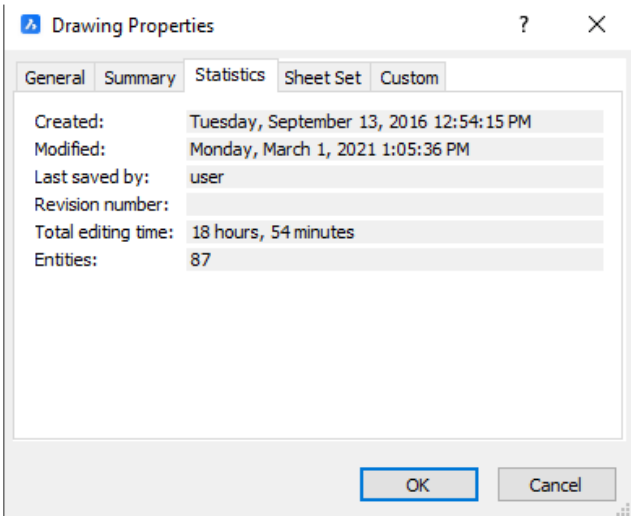
4.12.59.2 Summary

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.



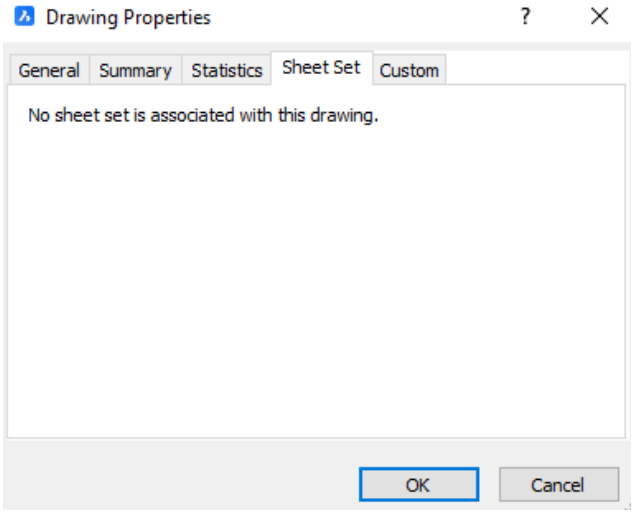
4.12.59.3 Statistics

Displays statistical information about the drawing, such as when it was created, modified.



4.12.59.4 Sheet Set

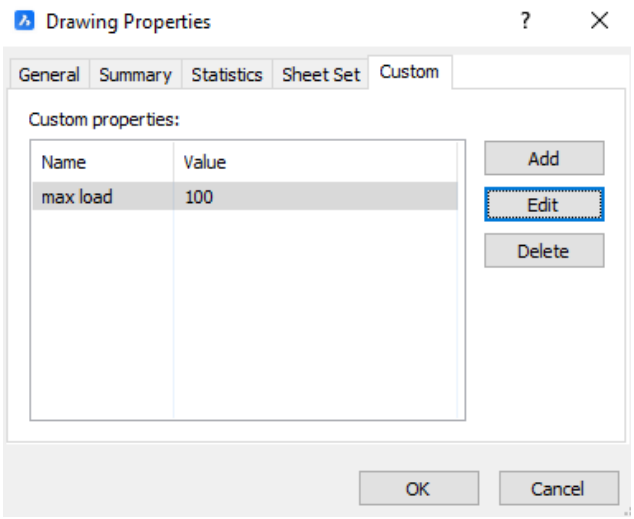
Displays sheet set data when the drawing is associated with a sheet set.



4.12.59.5 Custom

Displays the custom properties of the drawing. Click the **Add** button to add a new property or the **Edit** button to edit a property.

The Custom Property dialog box appears when clicking the **Add** and **Edit** buttons.

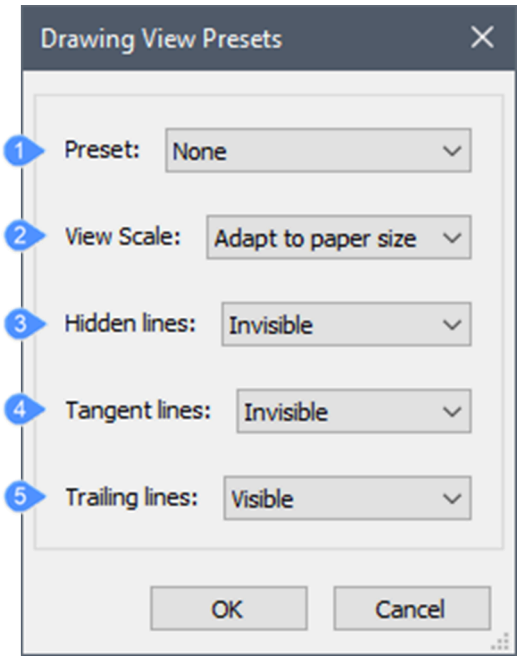


4.12.59.6 Command reference

Opens the Bricsys Help article about the DWGPROPS command.

4.12.60 Drawing view presets dialog box

Specifies the types of generated drawings and their placement in the layout.



1 Preset

2 View Scale

3 Hidden lines

4 Tangent lines

5 Trailing lines

4.12.60.1 Preset

Select a preset view arrangement:

- **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 system variable.

4.12.60.2 View Scale

Sets the default scale factor for the drawing views from drop-down list. The scale list can be edited by the SCALELISTEDIT command.

4.12.60.3 Hidden lines

Sets the default setting for hidden lines in drawing views.

4.12.60.4 Tangent lines

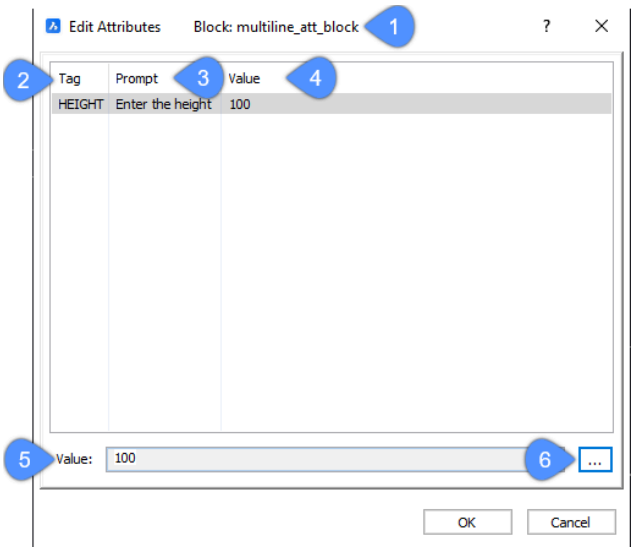
Sets the default for **Tangent Lines** in drawing views.

4.12.60.5 Trailing lines

Sets the default for **Trailing Lines** in drawing views.

4.12.61 Edit attributes dialog box

The Edit Attributes dialog box allows you to edit the values of attributes.



1 Block name

2 Tag

3 Prompt

4 Value

5 Value

6 Text Formatting

4.12.61.1 Block name

Displays the name of the selected block.

4.12.61.2 Tag

Specifies the name of the attribute.

Nota: This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.

4.12.61.3 Prompt

Specifies the user prompt.

Nota: 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.

4.12.61.4 Value

Displays the value that belongs to a certain attribute.

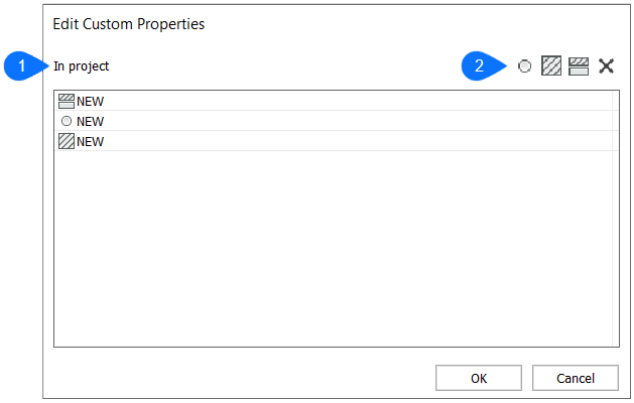
4.12.61.5 ... Text Formatting

Changes the value of the attribute text.

Nota: To be able to use this option, the Multiple lines attribute flag must be selected, in the Define Attribute dialog box, when defining an attribute.

4.12.62 Edit custom properties dialog box

The Edit Custom Properties dialog box allows you to add custom properties to materials and compositions.



1 In project

2 Add Generic/ Material/Composition/Delete properties

4.12.62.1 In project

Displays the custom properties available in the project.

4.12.62.2 Add/Delete Properties

Add Generic Custom Property
Creates a new generic custom property, which can be assigned to building elements in the project.

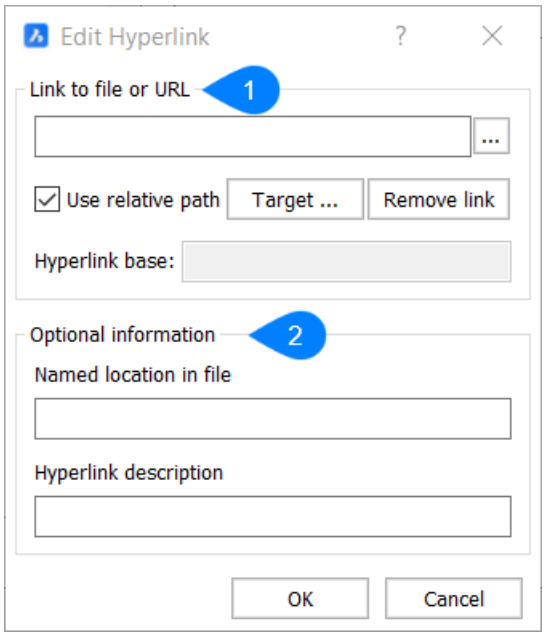
Add Material Custom Property
Creates a new material custom property, which can be assigned to materials in the project.

Add Composition Custom Property
Creates a new composition custom property, which can be assigned to compositions in the project.

Delete
Deletes the selected property.

4.12.63 Edit hyperlink dialog box

The Edit Hyperlink dialog box allows you to change the hyperlink to a file or URL.



1 Link to file or URL

2 Optional information

4.12.63.1 Link to file or URL

Specifies the URL of the file either by entering the URL or file path or through the Select File dialog box by pressing the Browse button.

Use relative path
Specifies a relative path by selecting the target file.

Nota: To remove the link, press **Remove** link.

Hyperlink base
Displays the hyperlink base path.

4.12.63.2 Optional information

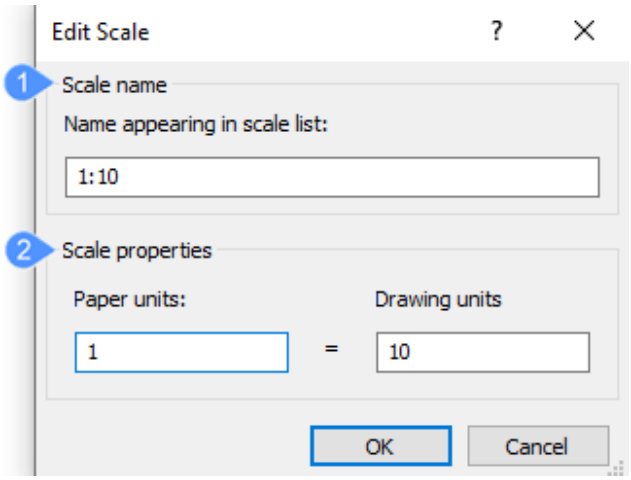
Specifies additional information

Named location file
Specifies a bookmark name. The # character specifies locations (bookmarks) within files or Web pages.

Hyperlink description
Adds an additional description of the hyperlink.

4.12.64 Edit scale dialog box

The **Edit Scale** dialog box allows you to edit the name and the scale properties of a scale factor.



1 Scale name

2 Scale properties

4.12.64.1 Scale name

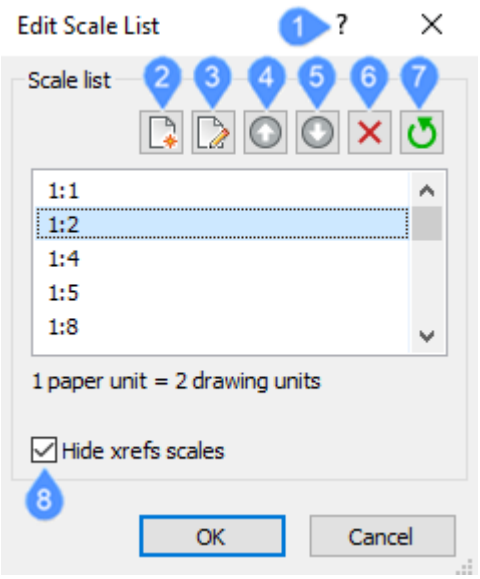
Edit the name by typing the new name.

4.12.64.2 Scale properties

Change the scale factor in paper (size when plotted) and/or drawing units.

4.12.65 Edit scale list dialog box

The **Edit Scale List** dialog box allows you to add, edit and delete scales.



1 Command reference

2 Add a new scale

3 Edit scale

4 Move up

5 Move down

6 Delete the selected scale

7 Reset

8 Hide xrefs scales

4.12.65.1 Command reference

Opens the Bricsys Help article about the SCALELISTEDIT command.

4.12.65.2 Add a new scale

Opens the **Add scale** dialog box.

4.12.65.3 Edit scale

Opens the **Edit Scale** dialog box.

4.12.65.4 Move up

Moves the selected scale up the list.

4.12.65.5 Move down

Moves the selected scale down the list.

4.12.65.6 Delete the selected scale

Deletes the selected scale. Press the CTRL key to select and delete more than one scale at a time.

Note that the scale 1:1 cannot be deleted.

4.12.65.7 Reset

Returns the list to its original condition, adding back in the deleted scales and removing all added but unused scales.

4.12.65.8 Hide xrefs scales

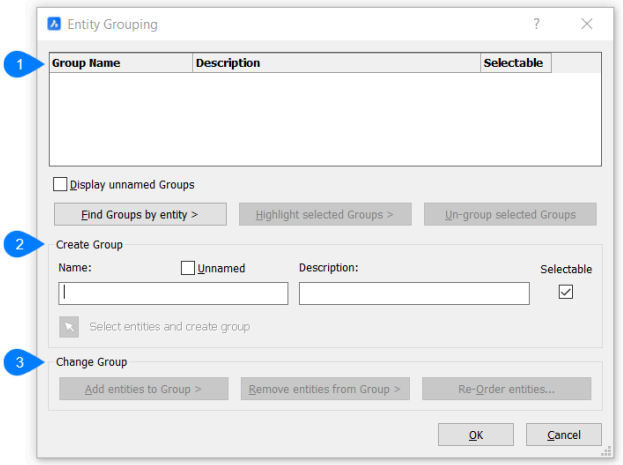
Determines if the scale factors from xrefs are displayed in the list:

- **Yes**: scale factors from externally referenced drawings are not shown in the list.
- **No**: scales in the attached xrefs not in the current drawing are listed here.

Toggles the HIDEXREFSCALES variable.

4.12.66 Entity grouping dialog box

The Entity grouping dialog box allows you to create, modify and delete named groups of entities.



1 Existing groups

2 Create Group

3 Change Group

4.12.66.1 Existing groups

Lists all existing groups with the Group Name and a short description.

Display unnamed Groups
Toggles the display of unnamed groups.

Find Groups by entity
Reports the name(s) of groups when entities in the drawing are selected.

Highlight selected Groups
Highlights all entities that belong to the selected group.

Ungroup selected Groups
Removes group status from selected entities.

4.12.66.2 Create Group

Name
Specifies the name of a group.

Unnamed
Toggles whether the group has a name.

Description
Adds an optional description of the group.

Selectable
Determines how groups are selected when the PICKSTYLE system variable is set to 1 or 3.

Select entities
Selects entities in the workspace which should be part of the group.

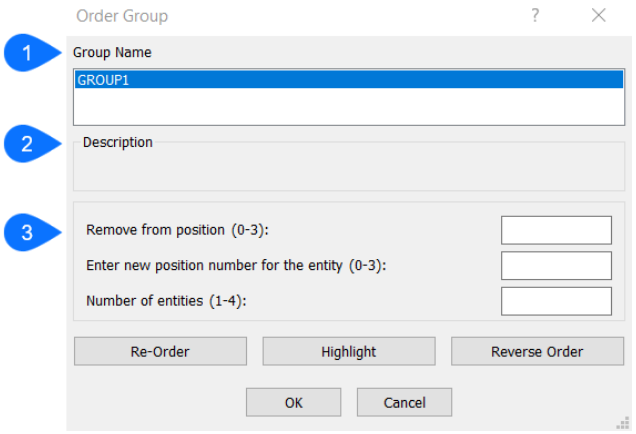
4.12.66.3 Change Group

Add entities to group
Adds entities to the selected group.

Remove entities from group
Removes entities from the selected group.

Re-Order entities
Changes the order of entities in groups trough the Order Group dialog box.

- Options in the Order Group dialog box

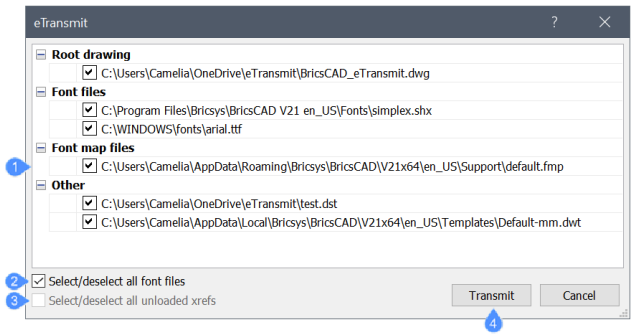


a Group Name
b Description
c Options

- **Group Name**: lists the names of named and unnamed groups.
- **Remove from position**: specifies the position number of the entity to be reordered.
- **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 reordering dictated by the previous three fields.
- **Highlight**: highlights entities in the group; dismisses the Order Group dialog box.
- **Reverse Order**: reverses the order of entities in the group.

4.12.67 eTransmit dialog box

The eTransmit dialog box 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.



1 Available files

2 Select/deselect all font files

3 Select/deselect all unloaded xrefs

4 Transmit

4.12.67.1 Available files

Select the files to be included in the eTransmit package (usually all of them).

4.12.67.2 Select/deselect all font files

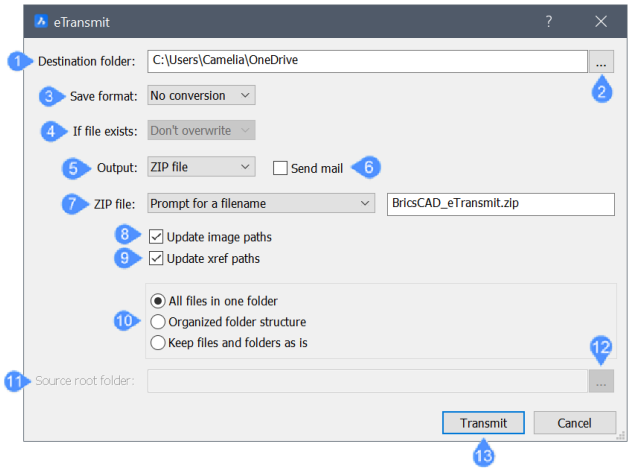
Tick the checkbox to automatically select all font files.

4.12.67.3 Select/deselect all unloaded xrefs

Tick the checkbox to automatically select all xref files.

4.12.67.4 Transmit

Opens the following dialog box:



1 Destination folder

2 Browse (destination folder)

3 Save format

4 If file exists

5 Output

6 Send mail

7 ZIP file

8 Update image paths

9 Update xref paths

10 Path options

11 Source root folder

12 Browse (root folder)

13 Transmit

Destination folder

Specifies the output folder for the eTransmit package. If the destination folder does not exist, you are prompted to create the folder.

Browse (destination folder)

Opens the Select the destination folder dialog box to allow you to choose the destination folder for the transmitted package.

Save format

Specifies the format of the drawing files in the eTransmit package.

If file exists

Specifies what to do if a file with the same name already exists in the output folder.

This option is available when Folder is chosen as output.

Select an option from the drop-down list:

- **Overwrite**: replaces the existing files with new ones.

- **Don't overwrite**: all files are transmitted, except files with the same file name as an existing file in the folder.
- **Abort**: the operation is aborted and no files are saved if a file with the same name already exists in the destination folder.

Output
Specifies whether to create a ZIP file or copy the files to the output folder.

Send mail
Tick the checkbox to send a mail with the zipped eTransmit package in attachment using your default mail application.
This option is available when ZIP is chosen as output.

ZIP file
Specifies what to do when a ZIP file with the same name already exists in the output folder.
Choose an option from the drop-down list:

- **Prompt for a file name**: opens a dialog box where you can choose a name and a path.
- **Overwrite if necessary**: replaces the existing file with the new one if a file with the same name already exists.
- **Increment name if necessary**: adds a suffix to the file name.

Update image paths
Tick the checkbox to update the saved paths for image files according to the new location of the transmitted files.

Update xref paths
Tick the checkbox to update the saved paths to externally-referenced files according to the new location of the transmitted files.

Path options
All files in one folder
All files in the eTransmit package are copied to the destination folder.

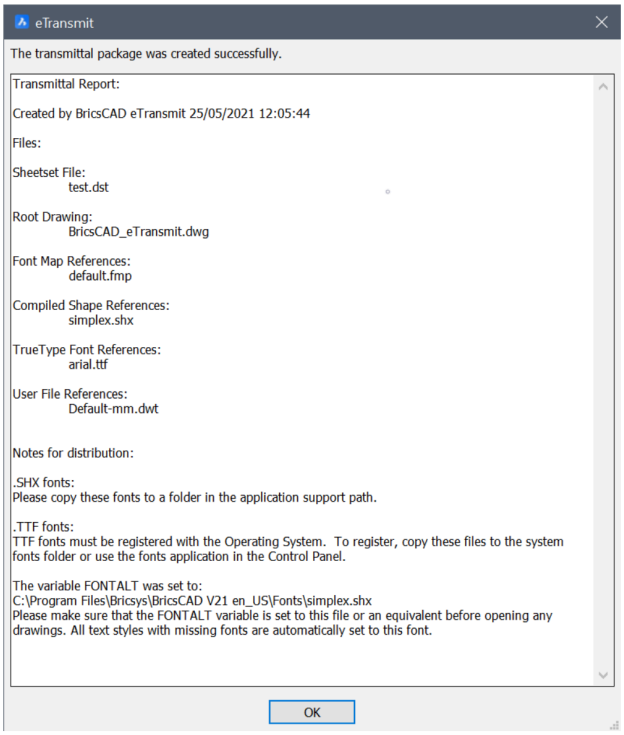
Organized folder structure
Creates a hierarchical folder tree with subfolders for all drawing file dependencies.

Keep files and folders as is
Keeps the folder structure of all files in the eTransmit package. This procedure allows you 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.
This option is available when **Organized folder structure** is checked.

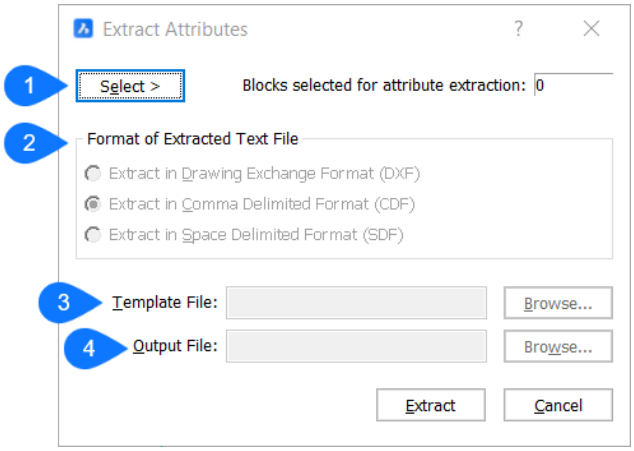
Browse (root folder)
Opens the Select the source root folder dialog box to allow you to choose a root folder.

Transmit
Transmits the files according to the selected options and displays a report of the transmission. The report contains a complete list of all files included and notes for distribution.



4.12.68 Extract attributes dialog box

The Extract Attributes dialog box allows you to copy data from attributes to a text file.



1 Select

2 Format of Extracted Text File

3 Template File

4 Output File

4.12.68.1 Select

Specifies the set of blocks with attributes to copy data from.

Nota: If blocks without attributes are selected, BricsCAD ignores them.

4.12.68.2 Format of Extracted Text File

Specifies the format of the Extracted text file. Several formats are available:

- DXF: drawing Exchange Format
- CDF: extract in Comma Delimited Format
- SDF: space Delimited Format

4.12.68.3 Template File

Specifies the location path of the template file by entering the path or through the Attribute Extract Template file dialog box after pressing the Browse button.

4.12.68.4 Output File

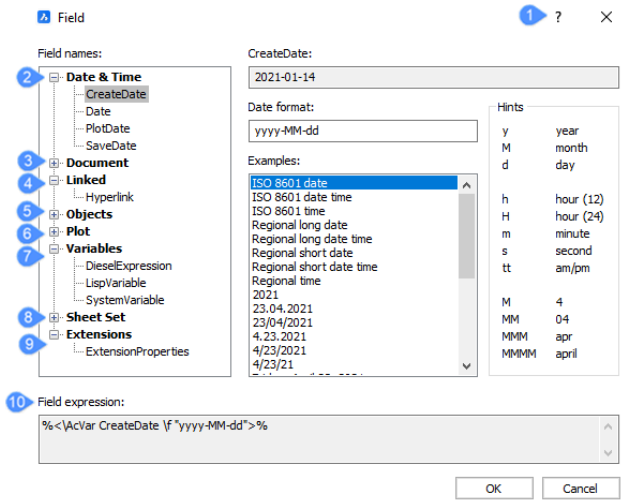
Specifies the location path of the output file by entering the path or through the Attribute Extract Output file dialog box after pressing the Browse button.

4.12.68.5 Extract

Exports the attribute values to the output file.

4.12.69 Field dialog box

The **Field** dialog box allows you to insert a field in a multiline text, table cells, attributes, and so on.



1 Command reference

2 Date & Time

3 Document

4 Linked

5 Objects

6 Plot

7 Variables

8 Sheet Set

9 Extensions

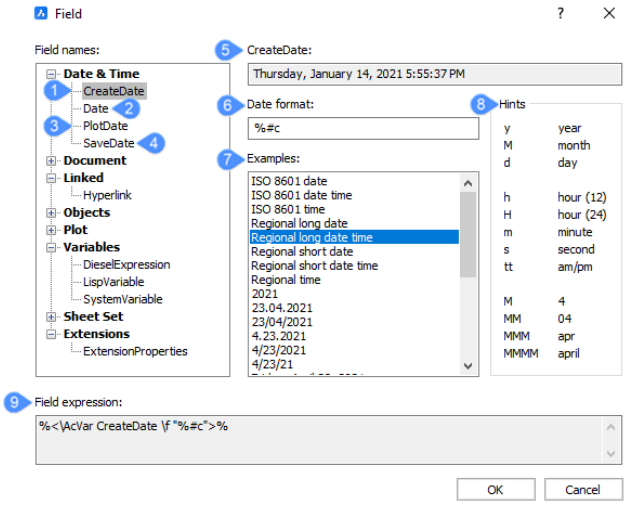
10 Field expression

4.12.69.1 Command reference

Opens the Bricsys Help article about the FIELD command.

4.12.69.2 Date & Time

For the **Date & Time** category, only the field number 1 changes based on the selected item.



1 CreateDate

2 Date

3 PlotDate

4 SaveDate

5 CreateDate

6 Date format

7 Examples

8 Hints

9 Field expression

CreateDate
Displays a random date.

Date
Displays the current date.

PlotDate
Displays the date the drawing was last plotted.

SaveDate
Displays the date the drawing was last saved.

Date format
Select a date format in the list or type a format in this field.

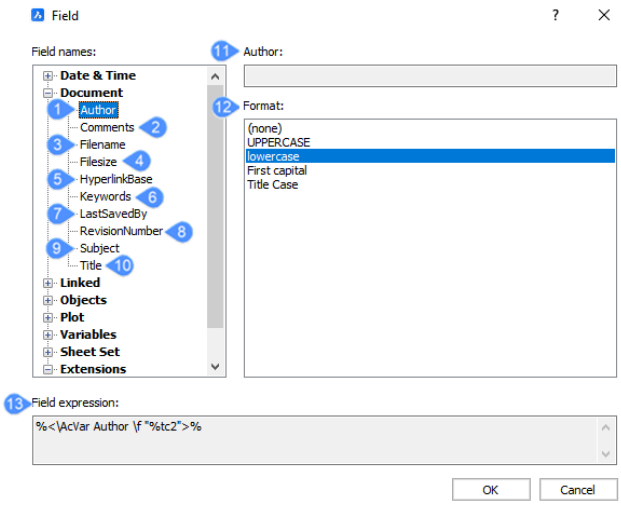
Examples
Displays date formats examples.

Hints
Explains the meaning of digits used in the date format acronym/abbreviation definition.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.3 Document

It includes 10 categories and 3 areas, excepting the Filename (no. 3) category that will be documented separately.

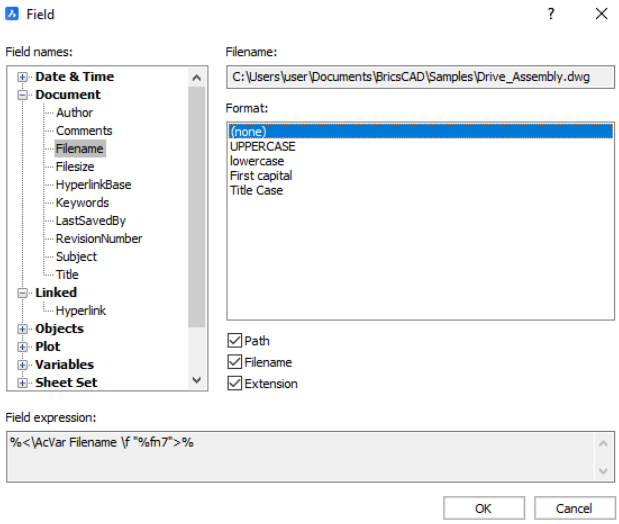


1 Author
2 Comments
3 Filename
4 Filesize
5 HyperlinkBase
6 Keywords
7 LastSavedBy
8 RevisionNumber
9 Subject
10 Title
11 Author
12 Format
13 Field expression

Author
Displays the author of the document.

Comments
Displays the comments of the document.

Filename
Shows the filename. It can include the path and the extension of the file.



Filesize
Displays the file size expressed in Bytes, Kilobytes and Megabytes.

HyperLinkBase
Displays the default path for relative hyperlink in drawings.

Keywords
Displays the keywords for the opened file.

LastSavedBy
Displays the name of the person who have last saved the file.

RevisionNumber
Displays the drawing revision number.

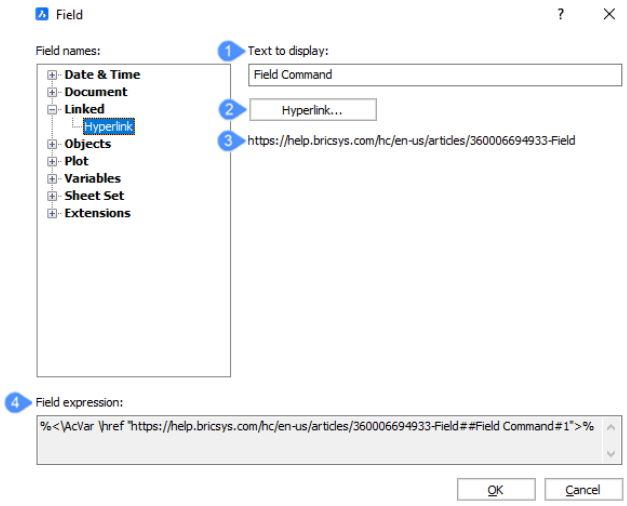
Subject
Displays the subject of the drawing.

Title
Displays the title of the drawing.

Format
Displays the format of the selected category.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.4 Linked



1 Text to display

2 Hyperlink

3 URL

4 Field expression

Text to display
You can insert in this field the text you want to be displayed for this hyperlink.

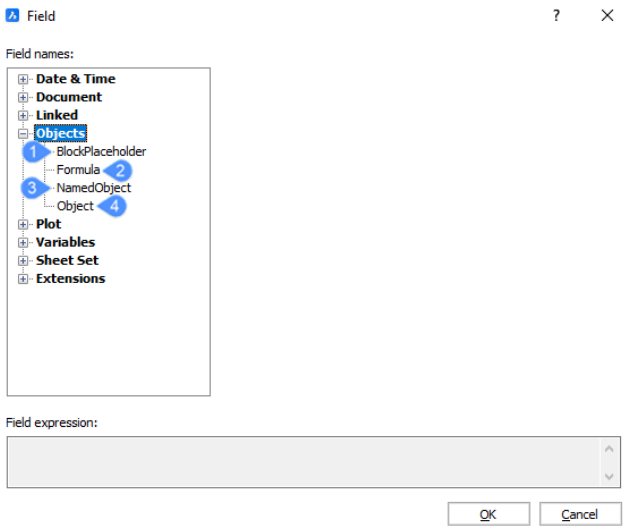
Hyperlink
Opens the **Edit Hyperlink** dialog box.

URL
Displays the URL.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.5 Objects

It includes 4 categories.



1 BlockPlaceHolder

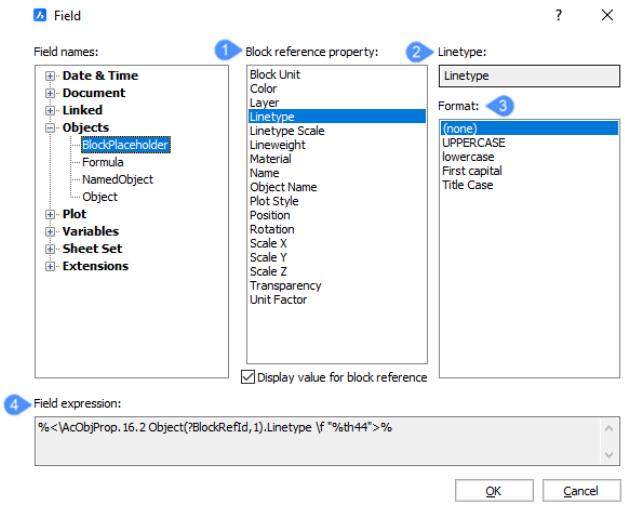
2 Formula

3 NamedObject

4 Object

BlockPlaceHolder

See the ATTDEF command to create block property fields in an attribute. When such attribute is included in a block definition, the field displays the current value of the block property.



1 Block reference property

2 Block name

3 Format

4 Field expression

- **Block reference property**: displays the block reference property.
- **Block name**: displays the name of the block.
- **Format**: displays the name of the block in different letter cases.

- **Field expression**: displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

Formula
Creates a formula field. You can use the values of table cells in a formula field. Click the **Average**, **Sum**, **Count** or **Cell** button. The **Field** dialog closes to let you select cells in a table in the drawing.

NamedObject
Creates a field that shows the current name of a named entity.

- Click the **Named object type** field, then select the object type in the list.

  All named objects of the selected type are listed in the **Name list** box.
- Select a name in the list.
- Choose a format.
- Click the **OK** button to place the field.

Object
Creates a field that shows a property of a selected entity.

- Click the select button, next to the **Object type** field.

  The **Field** dialog temporarily closes.
- Select an entity in the drawing.
- The properties of the selected entity are listed in the Property list.
- Select a property in the list.
- Choose a format.
- Click the **OK** button to place the field.

4.12.69.6 Plot

Plot fields apply to paper space layouts.

Allows to create fields which show the current print settings of a layout.

4.12.69.7 Variables

Allows to create fields which show the current value of Diesel Expressions, Lisp Variables and System Variables.

4.12.69.8 Sheet Set

Allows to create fields which show default and custom sheet set properties.

4.12.69.9 Extensions

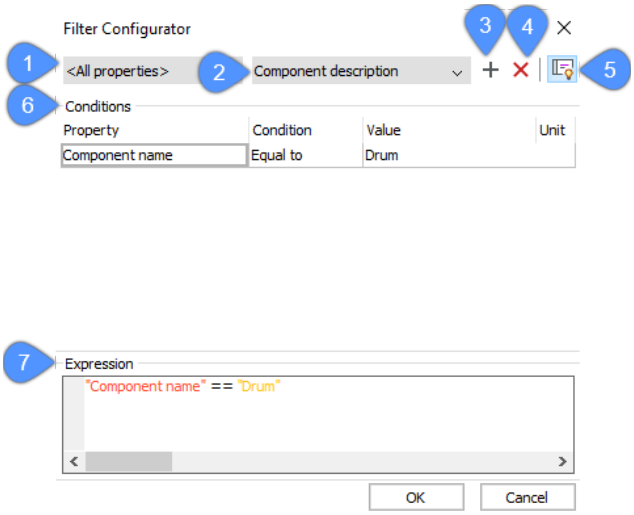
Allows to create fields showing properties of entities created by extensions, such as BIM or Mechanical.

4.12.69.1 Field expression 0

Displays the expression of the field.

4.12.70 Filter Configurator dialog box

The Filter Configurator dialog box allows you to specify the conditions to filter components in the BOM table.



1 Columns

2 Properties

3 Add

4 Delete

5 Expression

6 Conditions

7 Expression Editor

4.12.70.1 Columns

Selects columns to apply the filter to.

4.12.70.2 Properties

Specifies the available properties that can be used to apply the filter.

Nota: When specified the options above, the condition can be added by pressing the plus sign.

Filter conditions

Displays the conditions were added. This includes the properties, it's condition, value and unit.

4.12.70.3 Add

Adds a condition for the selected property.

4.12.70.4 Delete

Deletes the selected condition(s) from the filter.

4.12.70.5 Expression

Shows or hides the expression editor.

4.12.70.6 Conditions

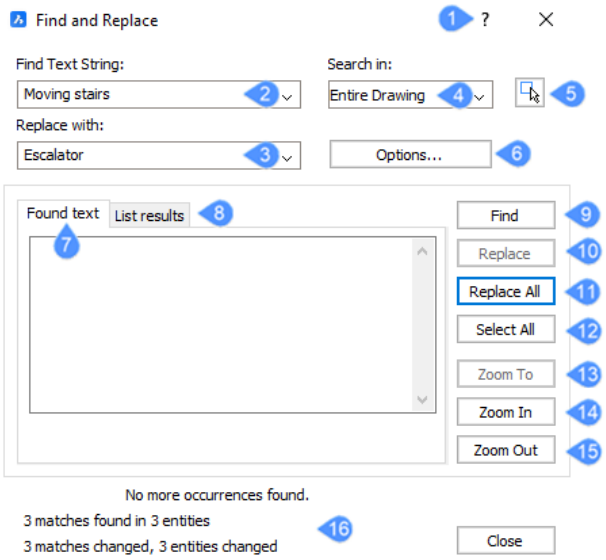
Displays all the added conditions that will be used to filter the content of the BOM table.

4.12.70.7 Expression Editor

Displays the Expression Editor where you can configure and preview the filtering expression.

4.12.71 Find and replace dialog box

The **Find and Replace** dialog box allows you to search for text in blocks, attributes, dimensions and hyperlinks, and optionally to replace it with another text string.



1 Command reference

2 Find Text

3 Replace With

4 Search In

5 Select Entities

6 Options

7 Found text

8 List results

9 Find

10 Replace

11 Replace All

12 Select All

13 Zoom To

14 Zoom In

15 Zoom Out

16 Search report

4.12.71.1 Command reference

Opens the Bricsys Help article about the FIND command.

4.12.71.2 Find Text

Specifies the text to be located.

4.12.71.3 Replace With

Specifies the text with which to replace the located text. Do not fill in this field when only searching for text.

4.12.71.4 Search In

Specifies the range of search:

- **Entire Drawing**: searches all text in the drawing.
- **Current Layout**: searches only among the selected entities.
- **Current Selection**: searches only in the current selection that can be made using button no. 5.

4.12.71.5 Select Entities

Dismisses the dialog box so that you can select the entities to search. After choosing one or more entities, press ENTER to return to the dialog box.

4.12.71.6 Options

Opens the **Find and Replace Options** dialog box that allows you to specify the kinds of text entities to search.

4.12.71.7 Found text

Displays the text string found.

4.12.71.8 List results

Displays the text string found, its location and the object type.

4.12.71.9 Find

Finds the next occurrence of the text string.

Shortcut Keystroke: CTRL+F.

4.12.71.1 Replace
0

Replaces the found text string with the replacement string.

Shortcut Keystroke: CTRL+R.

4.12.71.1 Replace All

1

Replaces all occurrences of the text string with the replacement string.

Shortcut Keystroke: CTRL+A.

4.12.71.1 Select All

2

Selects all occurrences of the text string.

4.12.71.1 Zoom To

3

Zooms to the part of the drawing containing the found text string.

4.12.71.1 Zoom In

4

Zooms in on the center of the window by a factor of 2.

4.12.71.1 Zoom Out

5

Zooms out from the center of the window by a factor of ½.

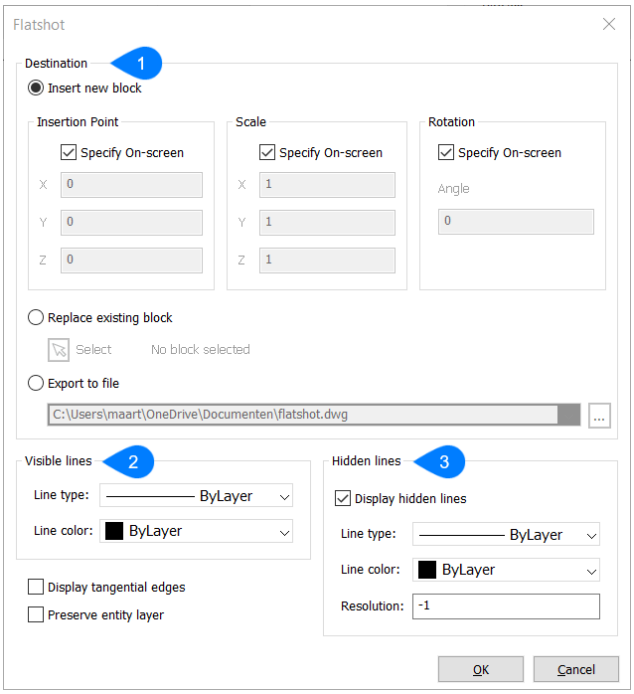
4.12.71.1 Search report

6

Displays the number of matches found in the selected entities, but also the number of matches and entities changed after replacement.

4.12.72 Flatshot dialog box

The Flatshot dialog box allows you to create a flattened, hidden line representation of 3D solids, 3D surfaces, and polyface meshes as a block or a new drawing.



1 Destination

2 Visible lines

3 Hidden lines

4.12.72.1 Destination

Determines where the block should be placed. Several options are available

Insert new block
Inserts the flatshot block as a new block in the current drawing by specifying some characteristics

Replace existing block
Replaces a block already inserted in the drawing.

Export to file
Saves the flatshot block as a dwg-file trough the Select Export File dialog box.

4.12.72.2 Visible lines

Sets the linetype and color for the visible lines. Choose the linetype and color from the drop lists.

Nota: A new linetype can be loaded into the drawing trough the Load Linetypes dialog box by selecting Load. A new line color can be selected trough the Select Color dialog box by selecting Select color.

4.12.72.3 Hidden lines

Toggles the display of hidden lines and sets their properties.

Display hidden lines
Toggles the display of hidden lines.

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.

4.12.72.4 Display tangential edges

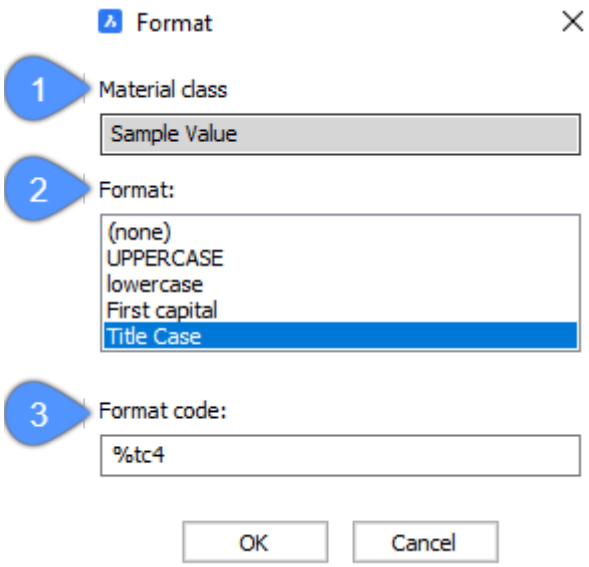
Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces.

4.12.72.5 Preserve entity layer

When on, maintains the layers of entities in the block.

4.12.73 Format dialog box

The Format dialog box allows you to specify the format of text strings in the specified column of a BOM table.



1 Material class

2 Format

3 Format code

4.12.73.1 Component

Specifies a component.

4.12.73.2 Format

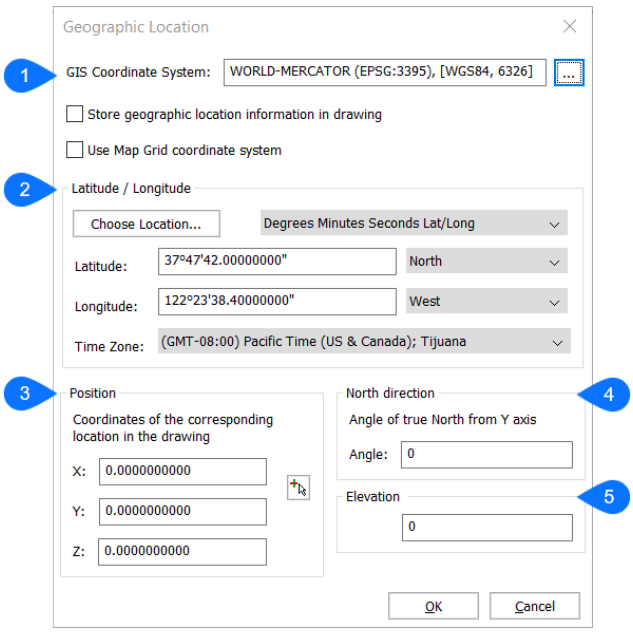
Specifies the desired format for the selected component from the list.

4.12.73.3 Format code

Changes depending on the format type you choose for all the materials in the BOM table.

4.12.74 Geographic location dialog box

The Geographic Location dialog box allows you to set the geographic location of the drawing by applying the longitude and latitude to a point in the drawing.



1 GIS Coordinate System

2 Latitude / Longitude

3 Position

4 North direction

5 Elevation

4.12.74.1 GIS Coordinate System

Reports the GIS system being used by the drawing.

Nota: Click the browse icon to change the system.

Store geographic location information in drawing
When checked, a red dot indicates the position in the drawing.

Use Map Grid coordinate system
Uses the location data from the GIS Coordinate System.

4.12.74.2 Latitude / Longitude

Choose Location
Specifies the graphic location trough the Choose Geographic Location dialog box.

Degrees Minutes Seconds Lat/Long
Displays the location in a Degrees, Minutes and Seconds format.

Decimal Lat/Long
Displays the location in a decimal format.

Latitude
Sets the altitude. The valid range is 0 to 90.

Longitude
Sets the longitude. The valid range is 0 to 180.

Time zone
Specifies the time zone from the Time Zone drop list.

Nota: This setting is saved to the TIMEZONE system variable.

4.12.74.3 Position

Specifies the position of the location in the drawing by entering coordinate values.

Nota: You can also select a specific point in the workspace to set the position.

4.12.74.4 North direction

Specifies the angle of the sun from north, in the context of the world coordinate system.

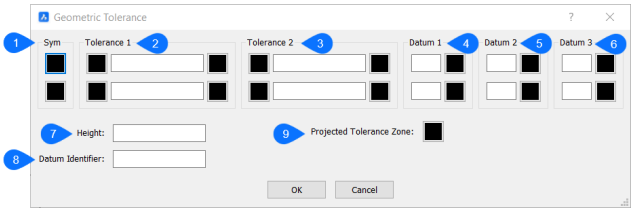
Nota: The setting is saved to the NORTHDIRECTION system variable.

4.12.74.5 Elevation

Specifies an elevation for the long-lat point. The value can be positive or negative.

4.12.75 Geometric tolerance dialog box

The Geometric Tolerance dialog box allows you to place tolerance symbols in drawings.



1 Symbol

2 Tolerance 1

3 Tolerance 2

4 Datum 1

5 Datum 2

6 Datum 3

4.12.75.1 Symbol

Specifies a tolerance symbol trough the Symbol dialog box.

4.12.75.2 Tolerance 1 & 2

Specifies the tolerance specifications (diameter, value and material condition).

Diameter
Toggles the diameter symbol.

Value
Specifies the tolerance value.

Material condition
Specifies the material condition trough the Material condition dialog box.

4.12.75.3 Datum 1, 2 & 3

Specifies the datum reference (value and material condition).

Value
Specifies the datum value.

Material condition
Specifies the material condition trough the Material condition dialog box.

Height
Specifies the height of the tolerance symbols.

Datum Identifier
Specifies datum identifier, 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

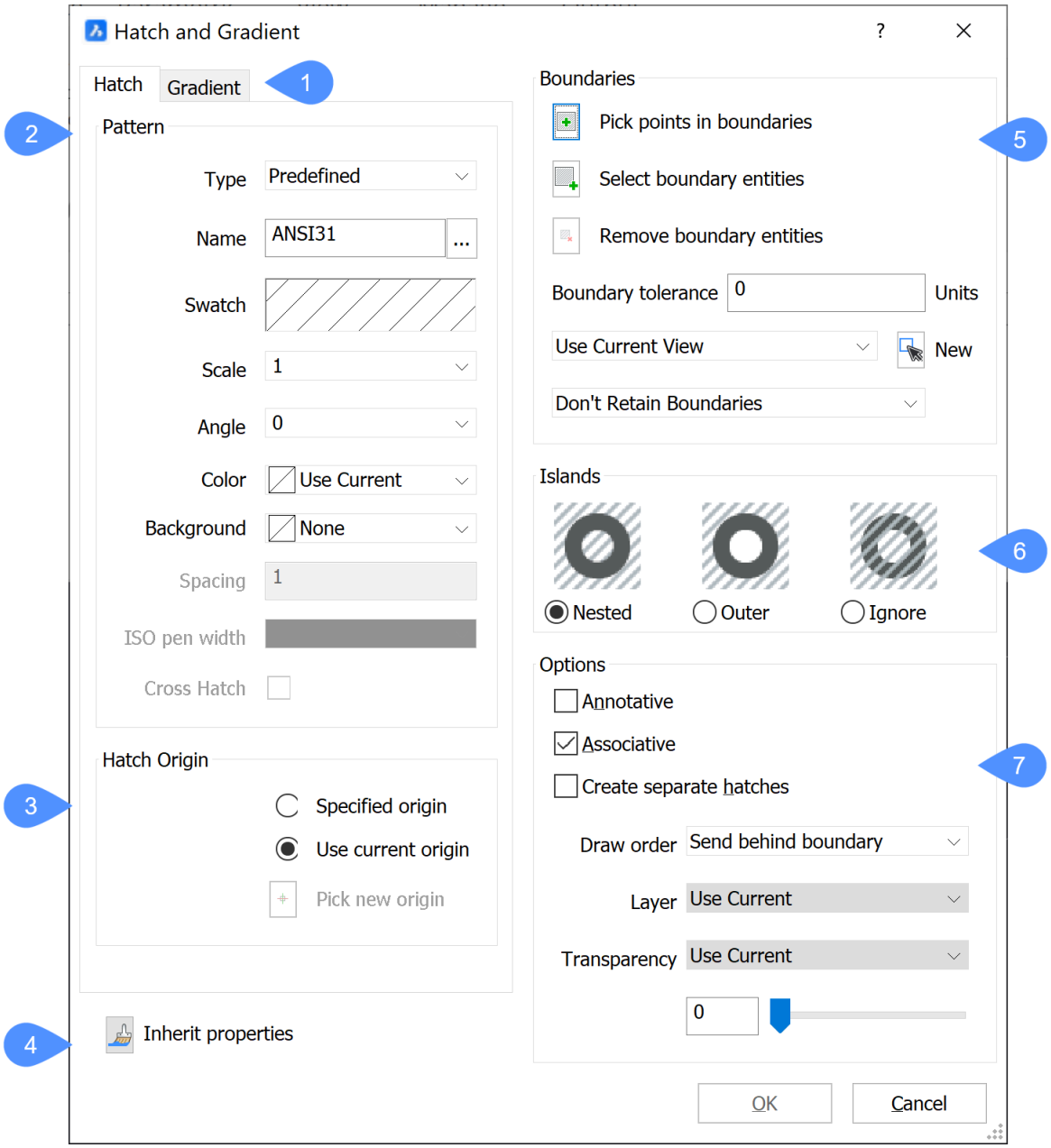
Symbol	Characteristic	Type
	Flatness	Form
	Circularity or Roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular Run-out	Run-out
	Total Run-out	Run-out

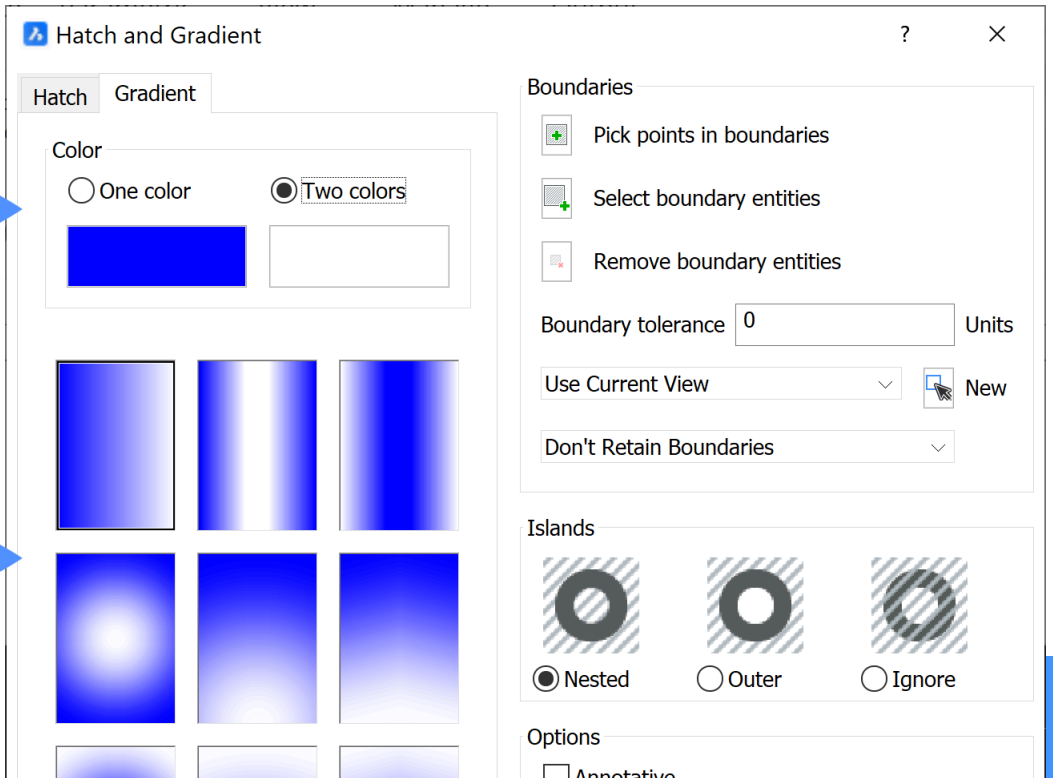
Material Condition Symbols

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.

4.12.76 Hatch and gradient dialog box

The Hatch and gradient dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.


Hatch and Gradient
Hatch
Gradient
1
2
Pattern
Type
Predefined
Name
ANSI31
Swatch
Scale
1
Angle
0
Color
Use Current
Background
None
Spacing
1
ISO pen width
Cross Hatch
3
Hatch Origin
Specified origin
Use current origin
Pick new origin
4
Inherit properties
Boundaries
5
Pick points in boundaries
Select boundary entities
Remove boundary entities
Boundary tolerance
0
Units
Use Current View
New
Don't Retain Boundaries
Islands
6
Nested
Outer
Ignore
Options
7
Annotative
Associative
Create separate hatches
Draw order
Send behind boundary
Layer
Use Current
Transparency
Use Current
0
OK
Cancel


Hatch and Gradient
Hatch
Gradient
Color
8
One color
Two colors
9
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

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

4.12.76.1 Pattern

Specifies the hatch pattern properties.

Type
Specifies the type of hatch.

User defined
Constructs the pattern from the user-defined parameters of Angle, Spacing, Color, Background and Cross Hatch.

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 system variable.

Name
Specifies the name of the predefined hatch pattern by one of these methods. Click the Browse button to display the Hatch Pattern Palette dialog box, to choose a pattern.

Nota: This option is unavailable for the User-defined patterns.

Swatch
Displays the Hatch Pattern Palette dialog box.

Nota: This option is unavailable for the User-defined patterns.

Scale
Sets the scale factor of the pattern. To use an enlarged hatch pattern, enter a value bigger than 1.

Nota: The scale factor of hatch patterns is generally the same as that for text and linetypes.

Angle
Sets the angle of the pattern.

Nota: This angle is measured relative to the positive x-axis of the current UCS.

Color
Sets the color of the hatch pattern lines.

Nota: For the Use Color option, the current color specified by the CECOLOR system variable is used.

Background
Sets the color of the background of the hatch.

Spacing
Specifies the distance between hatch lines.

Nota: This option is available for only user defined patterns.

ISO pen width
Specifies the lineweight.

Nota: This option is available only for ISO hatch patterns.

Cross Hatch
Determines whether the pattern is crossed (repeated at 90 degrees to the original).

Nota: This option is available only for user defined patterns.

4.12.76.2 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

4.12.76.3 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

4.12.76.4 Boundaries

Specifies the boundary conditions of the hatch.

Pick points in boundaries
Specifies the closed areas (boundaries) in which to place the pattern.

Nota: 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.

Nota: 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.

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.

Nota: The value is saved in the HPGAPTOL system variable.

Use current view / boundary set
Specifies where the program should search for entities that make up the boundary.

New
Creates a new selection set of entities that make up the pattern boundary.

Nota: 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).

4.12.76.5 Islands

Specifies how BricsCAD will respond when other boundaries are present within the closed hatch boundary.

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.

4.12.76.6 Options

Annotative
Toggles the Annotative property of the hatch. When turned on, BricsCAD applies the current annotative scale.

Nota: 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.

Nota: This option is not available for annotative hatches.

Create separate hatches
Toggles how multiple hatches are treated. If turned on, BricsCAD creates a separate hatch entity for each boundary in the selection set.

Draw order
Specifies where the pattern should be placed relative to overlapping entities.

Layer
Specifies the layer on which the hatch should be placed.

Transparency
Set the transparency property of the hatch.

Nota: For the Use current option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

4.12.76.7 Gradient Color

Specifies the gradient color. The gradient can either exist of one color to create a solid fill hatch or exist out of 2 colors.

4.12.76.8 Gradient pattern

Specifies the pattern of the gradient.

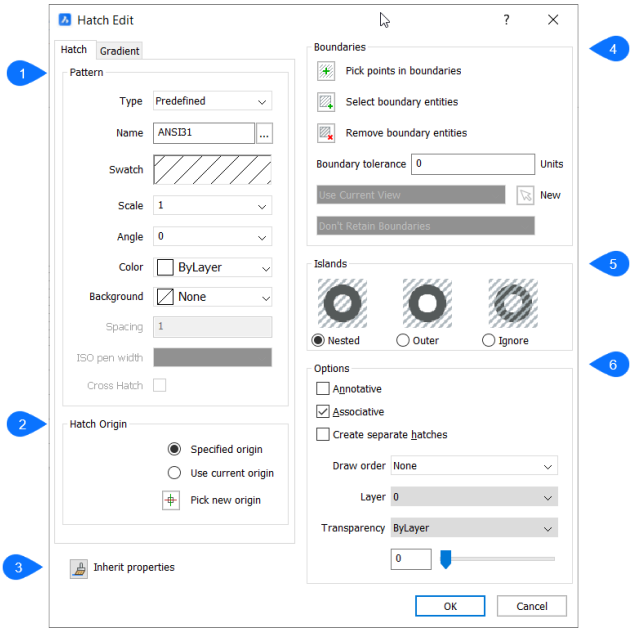
4.12.76.9 Gradient Orientation

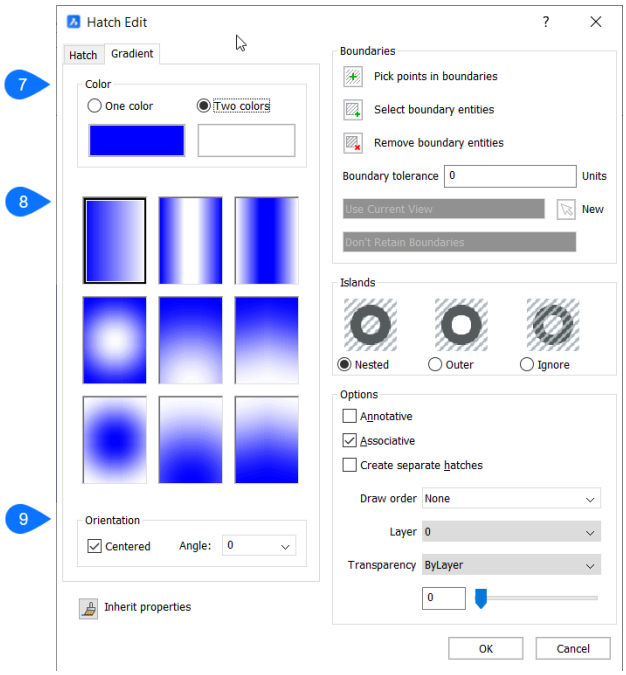
Specifies how the fill is placed in the boundary.

- Centered: centers gradient fill in the area to be filled; when off, starts the fill from the upper left of the boundary.
- 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.

4.12.77 Hatch Edit dialog box

The Hatch Edit dialog box allows you to edit an already existing hatch in the drawing. All options resemble the options of the Hatch and Gradient dialog box.





1 Pattern

2 Hatch origin

3 Inherit properties

4 Boundaries

5 Islands

6 Options

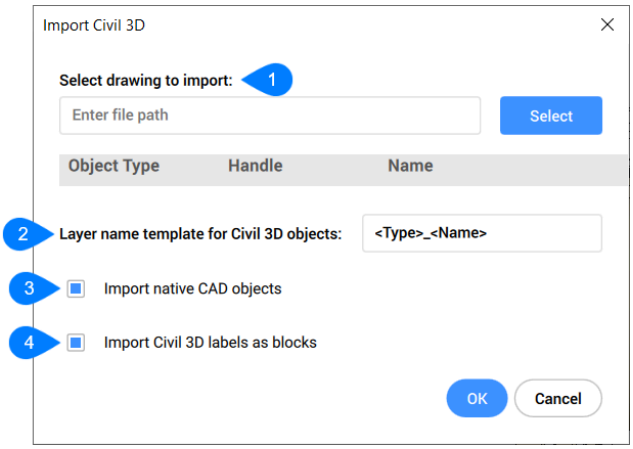
7 Color

8 Pattern

9 Orientation

4.12.78 Import Civil 3D dialog box

The Import Civil 3D dialog box allows you to create BricsCAD civil entities from an Autodesk® Civil 3D drawing.



1 Select drawing to import

2 Layer name template for Civil 3D objects

3 Import native CAD objects

4 Import Civil 3D labels as blocks

4.12.78.1 Select drawing to import

Specifies the drawing to import by clicking the Select button, which opens the Open File dialog box.

4.12.78.2 Layer name template for Civil 3D objects

Edits the template that is used for the layer name of all Civil 3D objects.

4.12.78.3 Import native CAD objects

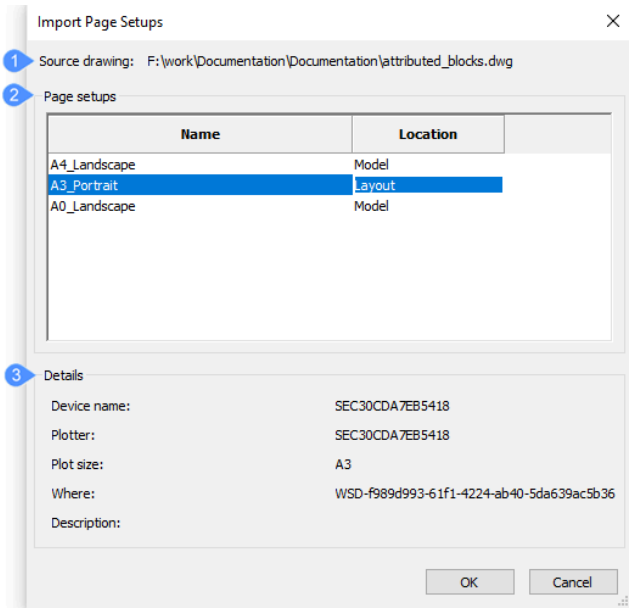
Toggles the option to also import native CAD objects.

4.12.78.4 Import Civil 3D labels as blocks

Toggles the option to import all Civil 3D labels as blocks.

4.12.79 Import page setups dialog box

The **Import Page Setups** dialog box allows you to select one or more page setups (using the CTRL key) from the list and import them into the current drawing.



1 Source Drawing

2 Page setups list

3 Details

4.12.79.1 Source Drawing

Displays the source of the drawing used to import the page setup(s).

4.12.79.2 Page setups list

Specifies the name and the location of each page setup from the list:

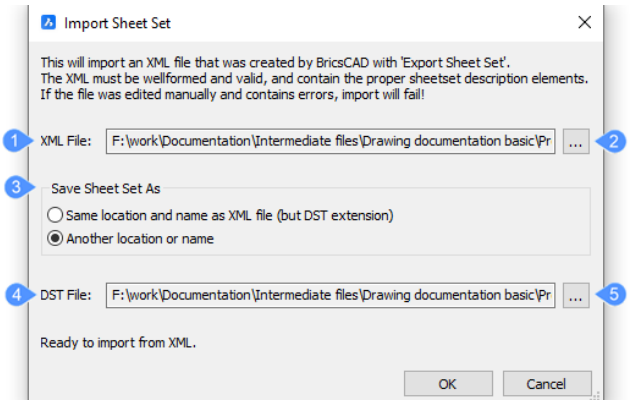
- **Name**: specifies the names of page setups in the drawing. When the drawing contains no page setups, then the dialog box is blank.
- **Location**: specifies the layout and model locations of page setups.

4.12.79.3 Details

Includes information about the device name, plotter, plot size, location and a description field.

4.12.80 Import sheet set dialog box

The **Import Sheet Set** dialog box allows you to import a sheet set file that has been created using the **Export Sheet Set As** dialog box.



1 XML File path

2 Select XML File

3 Save Sheet Set As

4 DST File path

5 Save Sheet Set As

4.12.80.1 XML File path

Displays the path of the *.xml file after selecting it using the **Select XML File** button.

4.12.80.2 Select XML File

Opens the **Select XML File** dialog box.

4.12.80.3 Save Sheet Set As

Choose whether to save the imported XML file as DST file using:

- The same location and name as XML file. Sections named 4 and 5 are unavailable.
- Another location or name. For the second option, click the button number 5.

4.12.80.4 DST File path

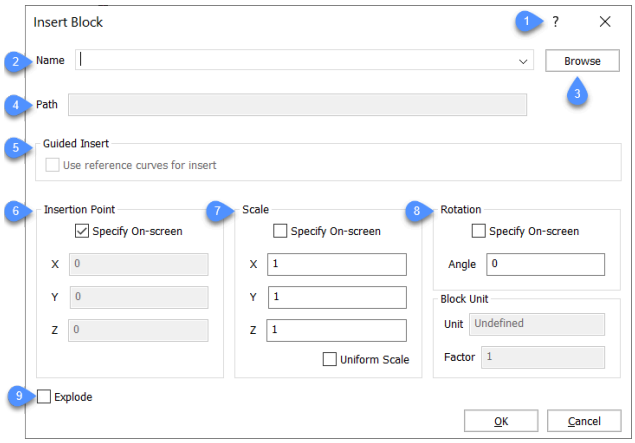
When you choose to save the imported XML file as DST file to a new location or name is displayed the path of this file.

4.12.80.5 Save Sheet Set As

Opens the **Save Sheet Set As** dialog box.

4.12.81 Insert block dialog box

The **Insert Block** dialog box allows you to insert blocks into the current drawing. You can also insert entire DWG and DXF files as blocks into the drawing.



Nota: If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through **Edit Attributes** dialog box, depending on the value of the value of the ATTDIA (attribute dialog) variable.

1 Command reference

2 Name

3 Browse

4 Path

5 Guided Insert

6 Insertion Point

7 Scale

8 Rotation

9 Explode

4.12.81.1 Command reference

Opens the Bricsys Help article about the INSERT command.

4.12.81.2 Name

Specifies the name of the block, whose definition exists in the drawing, or is a DWG or DFX file on the computer or network.

4.12.81.3 Browse

Selects a DWG or DXF file from your computer or network. Opens the **Select Drawing File** dialog box.

4.12.81.4 Path

Displays the path to the block, if the block was opened from a DWG or DXF file.

4.12.81.5 Guided Insert

When a dwg is selected that contains guide curves, the **Guided Insert** option becomes selectable. It is used for the insertion of parametric blocks.

- **Use reference curves for insert**: Specifies if a reference should be used for insert or not.

4.12.81.6 Insertion Point

Specifies the insertion point of the block in the drawing.

Nota: 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.

4.12.81.7 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.
- **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 the same scale factor is used for all axes.

4.12.81.8 Rotation

Specifies the rotation angle of the block about its insertion point, starting with the x-axis as 0 degrees.

Nota: Positive angles rotate the block counter-clockwise and negative angles rotate the block clockwise.

4.12.81.9 Block unit

Controls the automatic scaling of the block with respect to the INSUNITS system variable of the current drawing.

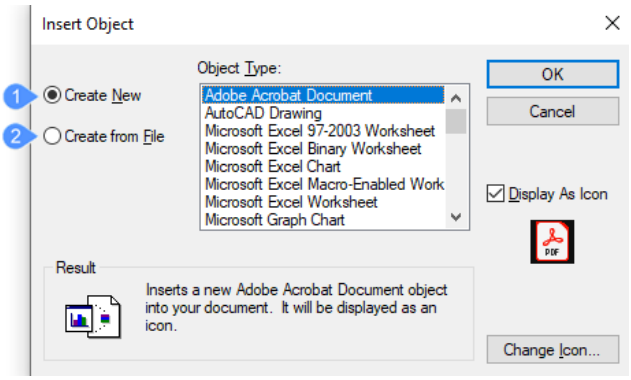
4.12.81.1 Explode

0

Toggles whether the block is inserted exploded.

4.12.82 Insert object dialog box

The Insert Object dialog box allows you to insert a linked or embedded document in the current drawing.

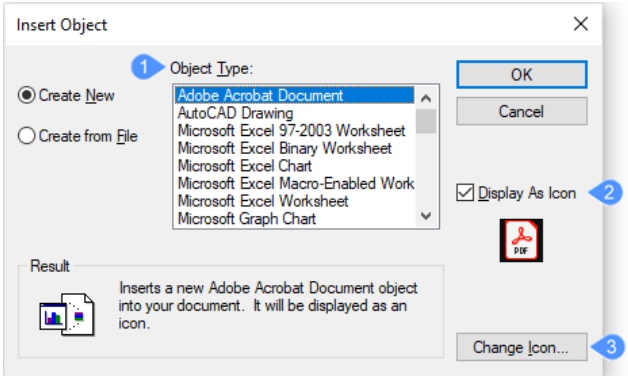


1 Create new Object

2 Create from File

Nota: The object types listed varies according the software installed on your computer.

4.12.82.1 Create new Object



1 Display as icon

2 Change icon

Display as icon
Displays the object as an icon. The icon is related to the document type and is defined by source application.

Change icon
Opens the Change Icon dialog box.

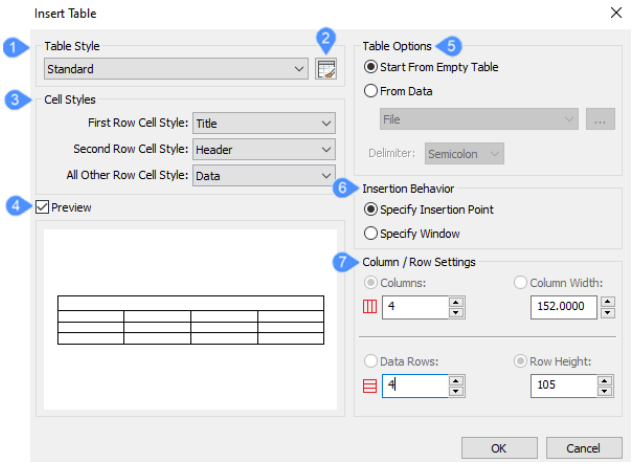
4.12.82.2 Create from File

Link
Toggles linkage to the source document:

- **On**: links the object in the drawing to the source file, so that when the source file is changed, the linked object also changes.
- **Off**: does link not the object.

4.12.83 Insert table dialog box

The **Insert table** dialog box allows you to create a table entity in the drawing either from scratch or from data in an external file or a data link.



1 Table Style

2 Table Style Explorer

3 Cell Styles

4 Preview

5 Table Options

6 Insertion Behavior

7 Column / Row settings

4.12.83.1 Table Style

Chooses an existing table style from the drop-down list.

4.12.83.2 Table Style Explorer

Allows you to create or edit table styles. Opens the **Table Styles** from the **Drawing Explorer** dialog box.

4.12.83.3 Cell Styles

A table style can have different sub-styles for the first two rows named **Title**, **Header** and **Data**:

- **First Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Title**. A **Title** row is the topmost row of the table.
- **Second Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Header**. A **Header** row is the first row of a column.
- **All Other Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Data**. A **Data** row is all other rows in the table.

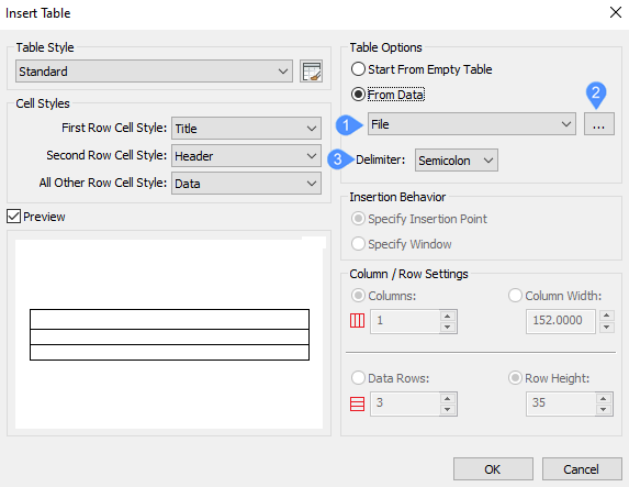
4.12.83.4 Preview

Displays a preview of the table depending on the options and settings you set in this dialog box.

4.12.83.5 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.



a File
b Open
c Delimeter

- **File**

  Read data for the new table from a file.

- **Open**

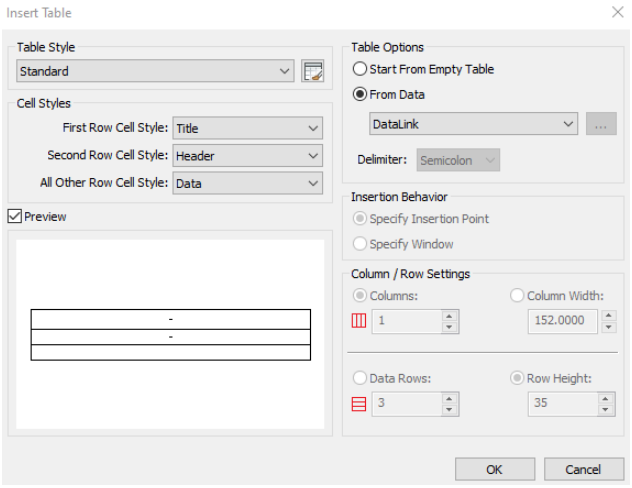
  Displays the **Open** dialog box from where you will select the file.

- **Delimeter**

  Specifies the delimiter character used to separate fields in CSV data files. Select from the drop-down list the delimiter you want to use.

  Nota: Available only for the **File** option.

- **DataLink**



When you select to read data from a datalink the **Datalink Manager** dialog box opens.

4.12.83.6 Insertion Behavior

(This option is available only when you create a new empty table.)

- **Specify Insertion Point**: specifies where the upper-left corner of the table will be inserted in the drawing.
- **Specify Window**: specifies the area that will be used by the table, this is useful when you need the table to fit a specific are, such as in a revision table or title block.

Nota: As you move the mouse, a fictive table dynamically resizes itself.

4.12.83.7 Column / Row settings

(This option is available only when you create a new empty table.)

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

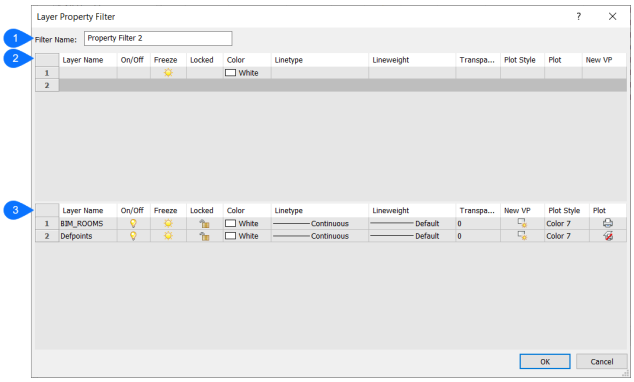
And choose one of:

- **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.

4.12.84 Layer Property Filter dialog box

The Layer Property Filter Dialog Box allows you to create a new layer filter definition, based on the properties of the layer.



1 Filter Name

2 Filter Properties

3 Filtered Layers

4.12.84.1 Filter Name

Sets the name of the filter.

4.12.84.2 Filter Properties

Sets the properties to filter the layers. Layers that fulfill all indicated properties of the filter will be added to the layer filter.

Layer Name

Filters the layers based on their name. You can use wild cards to specify more than one layer, for example Doors-* for Doors-Exterior and Doors-Interior.

- # - matches any digit. For example, ## matches 12 and 34 but not 5 or 123 (ASCII code: 35).
- @ - matches any alphabetic character, so not digits or symbols. For example, @@@@@ matches all 5-letter layer names. (ASCII code: 64).
- . - matches non-alphanumeric characters, so symbols but not letters or numbers. For example, .* matches all layer names starting with a non-alphanumeric character such as _SITEPLAN and +1_LEVEL (ASCII code: 46).
- * - matches any string, including letters, numbers, and characters. For example, A* matches all layer names starting with the letter A (ASCII code: 42).
- ? - 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 ? specifies a single letter (ASCII code: 63).
- ~ - negative match, in which any string but the pattern is returned. For example, ~*A* matches all layer names not containing the letter A (ASCII code: 126).
- [ ] - Matches any one of the characters enclosed inside the square brackets. For example, LA[YB] matches LA**Y**and LA**B**(ASCII code: 91 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** (ASCII code: 91 126 93).

- [-] - 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 (ASCII code: 91 45 93).
- ‘ - Matches the next string exactly to find 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: 96).

Nota: 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.

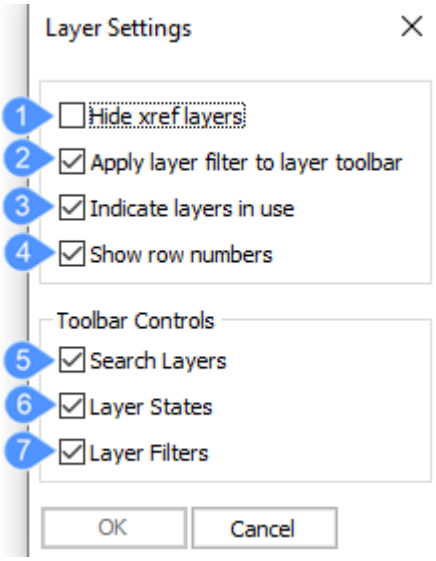
Properties
Specifies which layers to include according to their properties, such as freeze and color.

4.12.84.3 Filtered Layers

Displays the layers that fulfill the requirements of the filter.

4.12.85 Layer settings dialog box

The **Layer Settings** dialog box allows you to apply settings on the **Layers** Panel.



1 Hide xref layers

2 Apply layer filter to layer toolbar

3 Indicate layers in use

4 Show row numbers

5 Search Layers

6 Layer States

7 Layer Filters

4.12.85.1 Hide xref layers

When on, hide layers that came from externally referenced drawings.

4.12.85.2 Apply layer filter to layer toolbar

When on, shortens the list of layers displayed, according to the conditions of the filter. When off, displays all layer names.

4.12.85.3 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.

4.12.85.4 Show row numbers

Toggles the display of the number at the start of each row.

4.12.85.5 Search Layers

Toggles the display of the **Search** field on and off.

4.12.85.6 Layer States

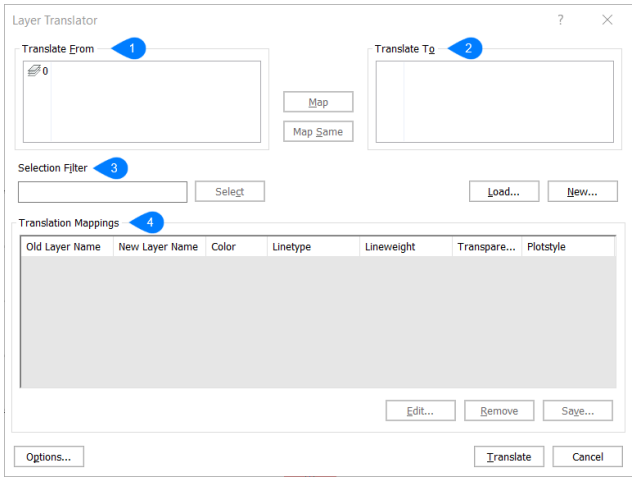
Toggles the display of the **Layer States** drop-down list.

4.12.85.7 Layer Filters

Toggles the display of the **Layer Filters** drop-down list.

4.12.86 Layer translator dialog box

The Layer Translator dialog box allows you to apply layer names and properties from another drawing to the current drawing.



1 Translate from

2 Translate to

3 Selection filter

4 Translation mappings

4.12.86.1 Translate from

Lists all the layer names in the current drawing.

Nota: You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.

Nota: New layers can be created by simply entering the layer name and properties to which you want to map existing layer.

4.12.86.2 Translate to

Specifies which layer you want the chosen layer(s) to map to.

Nota: Select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose Map Same any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right.

4.12.86.3 Translation mappings

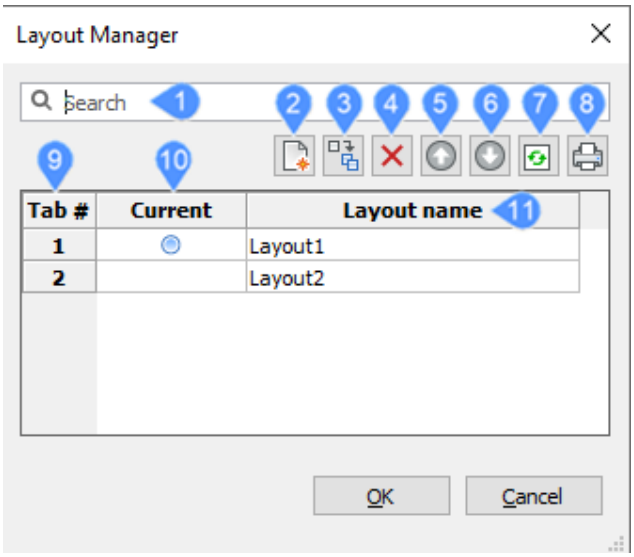
Displays an overview of which layer and its properties are translated to another layer.

4.12.86.4 Options

Displays the Options dialog box, which offers additional controls for layer mapping.

4.12.87 Layout manager dialog box

The Layout Manager dialog box allows you to create, copy and delete layouts.



1 Search

2 Add new layout

3 Copy selected layout

4 Delete

5 Move up

6 Move down

7 Clear selection

8 Publish

9 Tab

10 Current

11 Layout name

4.12.87.1 Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

4.12.87.2 Add new layout

Creates a new layout.

4.12.87.3 Copy selected layout

Makes a copy of the selected layouts and adds them to the end of the list.

4.12.87.4 Delete

Deletes the selected layouts.

4.12.87.5 Move up

Moves the selected layouts up the list.

4.12.87.6 Move down

Moves the selected layouts down the list.

4.12.87.7 Clear selection

Removes highlighting from layout names, thereby unselecting them.

4.12.87.8 Publish

Opens the **Publish** dialog box.

4.12.87.9 Tab

Displays the number of the layout.

4.12.87.1 Current

0

Click in the **Current** column to make the layout current.

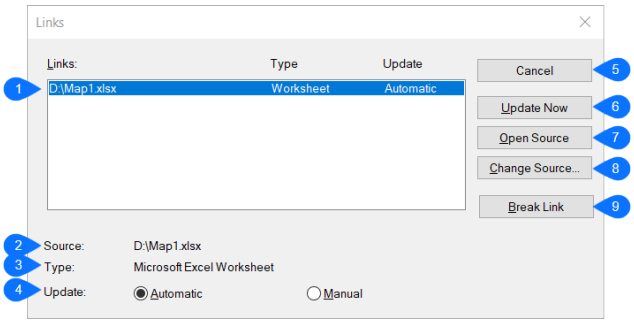
4.12.87.1 Layout name

1

Displays the name of the layout.

4.12.88 Links dialog box

The Links dialog box allows you to edit the links of OLE entities that were embedded or linked to drawings.



1 Links

2 Source

3 Type

4 Update

5 Cancel

6 Update Now

7 Open Source

8 Change Source

9 Break Link

4.12.88.1 Links

Provides a list of all links loaded in the file.

4.12.88.2 Source

Reports the path to the source file name.

4.12.88.3 Type

Reports the file type.

4.12.88.4 Update

Determines how the OLE entity in the drawing is updated when the source changes.

4.12.88.5 Cancel

Cancels the linking operation.

4.12.88.6 Update Now

Updates linked OLE entities.

4.12.88.7 Open Source

Opens the source file in the source application. This allows you to edit the source file.

4.12.88.8 Change Source

Changes the source for the linked OLE entity.

4.12.88.9 Break Link

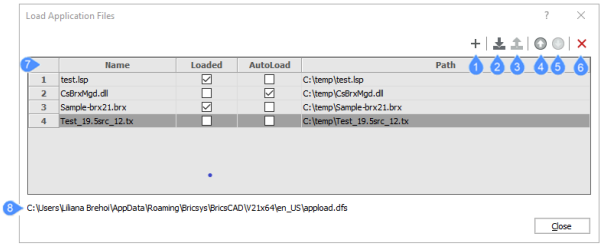
Breaks the link with the source file.

Nota: When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.

4.12.89 Load application files dialog box

The **Load Application Files** dialog box allows you to load and unload the following applications type 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.



1 Add application file

2 Load selected application

3 Unload selected application

4 Move application upward

5 Move application downward

6 Remove selected application

7 Application Files List

8 Application File Path

4.12.89.1 Add application file

Opens the **Select Application Files** dialog box.

4.12.89.2 Load selected application

Loads the selected application(s) into BricsCAD. Only compatible applications with BricsCAD will be loaded.

4.12.89.3 Unload selected application

Unloads the selected application(s) from BricsCAD. The LISP applications have to be removed in order to be unloaded in the current BricsCAD session.

4.12.89.4 Move application upward

Moves the selected application(s) up in the list.

4.12.89.5 Move application downward

Moves the selected application(s) down in the list.

4.12.89.6 Remove selected application

Removes the selected application(s) from the list.

4.12.89.7 Application Files List

Displays a list with application files that are available for load, unload, move up and down or remove.

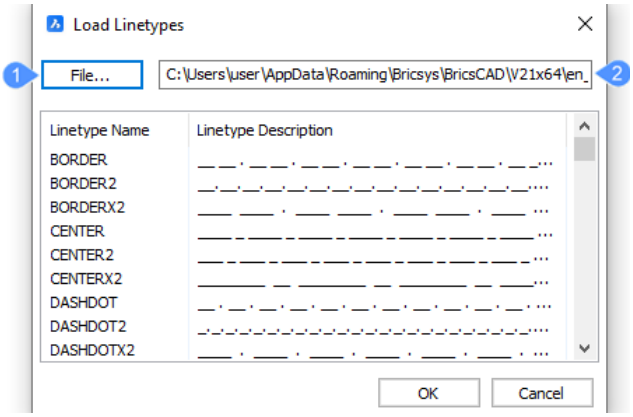
- **Name**: specifies the name and the extension of the application file from the list.
- **Loaded**: specifies if a file is loaded or unloaded in the current BricsCAD session.
- **Autoload**: specifies if a file will be loaded automatically in the next BricsCAD session.
- **Path**: displays the location folder of the added files.

4.12.89.8 Application File Path

Displays the location folder for appload.dfs file that contains all applications added, using the **Select Application Files** dialog box. You can open the file using Notepad.

4.12.90 Load linetypes dialog box

The **Load Linetypes** dialog box allows you to load additional linetype definitions.



1 File

2 File's path

4.12.90.1 File

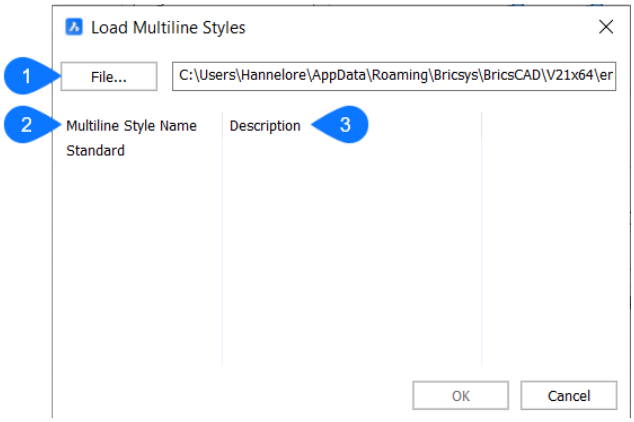
Opens the **Choose a Linetype File** dialog box.

4.12.90.2 File's path

Displays the path of the selected *.lin (linetype) file.

4.12.91 Load Multiline Styles dialog box

The Load Multiline Style dialog box allows you to load multiline styles from a multiline (*mln) file.



1 File

2 Multiline Style Name

3 Description

4.12.91.1 File

Displays the Choose a Multiline Style File dialog box where you can select a multiline style file to open.

4.12.91.2 Multiline Style Name

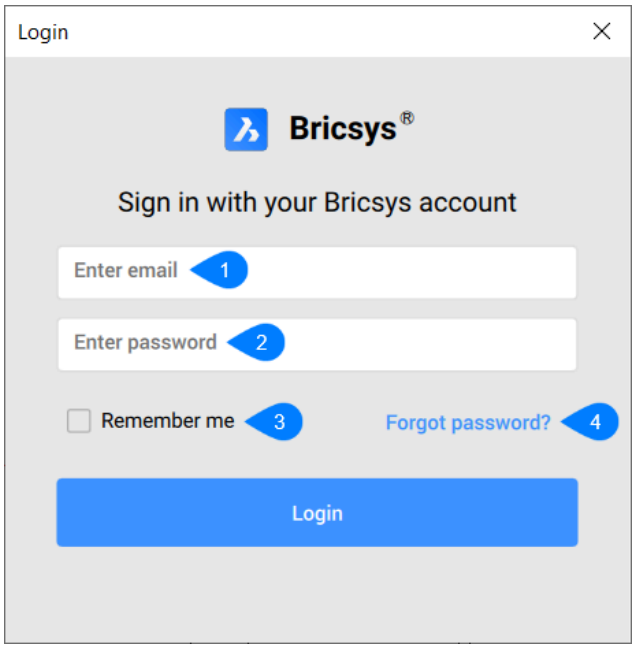
Displays the names of the available multiline style files in the drawing. You can select a multiline style file from this list.

4.12.91.3 Description

Displays a description of the multiline style files, if available.

4.12.92 Login dialog box

The Login dialog box allows you to login to BricsCAD with your personal Bricsys account.



1 Enter email

2 Enter password

3 Remember me

4 Forgot password?

4.12.92.1 Enter email

Specifies the email address connected to your Bricsys account.

4.12.92.2 Enter password

Specifies the password.

4.12.92.3 Remember me

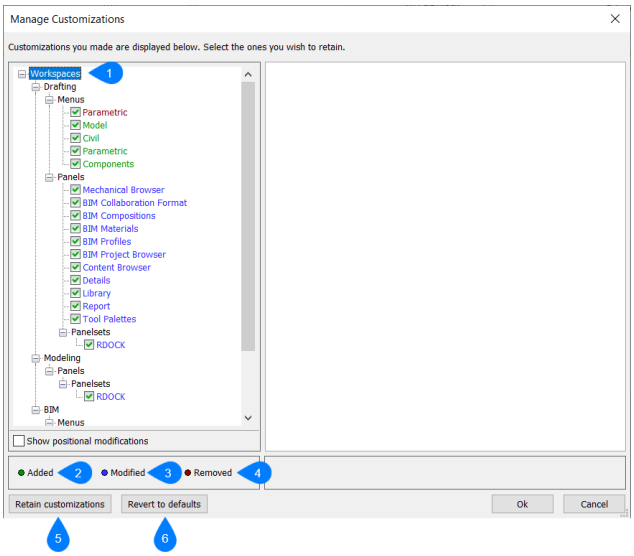
Check this box to keep being logged in.

4.12.92.4 Forgot password?

Redirects you to an internet browser to set up a new password.

4.12.93 Manage Customizations dialog box

The Manage Customizations dialog box provides a list of all customizations for confirming and reversing changes made to the user interface.



1 Customizations

2 Added

3 Modified

4 Removed

5 Retain customizations

6 Revert to defaults

4.12.93.1 Customizations

Lists all added, modified or removed customizations.

Nota: You can check a customization to retain the modification in the CUI.

4.12.93.2 Added

Items you have created display in green.

4.12.93.3 Modified

Items you have modified display in blue.

4.12.93.4 Removed

Items you have removed display in red.

4.12.93.5 Retain customizations

Keeps your local default CUI as it is.

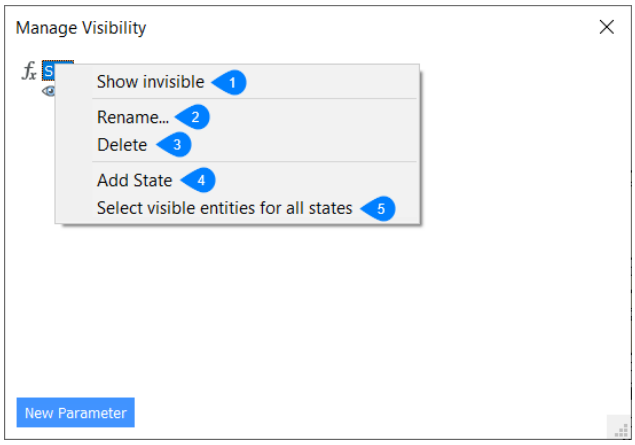
4.12.93.6 Revert to defaults

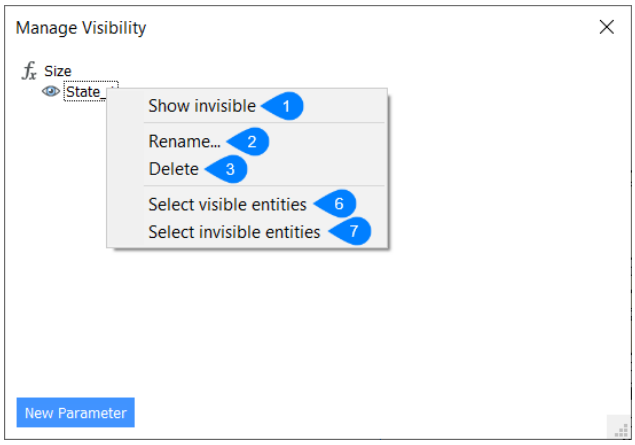
Reverts your local default CUI to the default one installed by BricsCAD in:

C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\default.cui.

4.12.94 Manage visibility

The Manage Visibility dialog box displays a list of visibility states in the drawing.





1 Show invisible

2 Rename

3 Delete

4 Add State

5 Select visible entities for all states

6 Select visible entities

7 Select invisible entities

4.12.94.1 Show invisible

Shows all invisible entities grayed-out.

4.12.94.2 Rename

Displays the Rename dialog box. Type a name to replace the existing name.

4.12.94.3 Delete

Deletes the parameter and its visibility states.

4.12.94.4 Add State

Displays the Rename dialog box. Type a name for the new state to replace the default name.

4.12.94.5 Select visible entities for all states

First choose Show invisible, then select the entities to remove all visibility states from the selected parameter.

4.12.94.6 Select visible entities

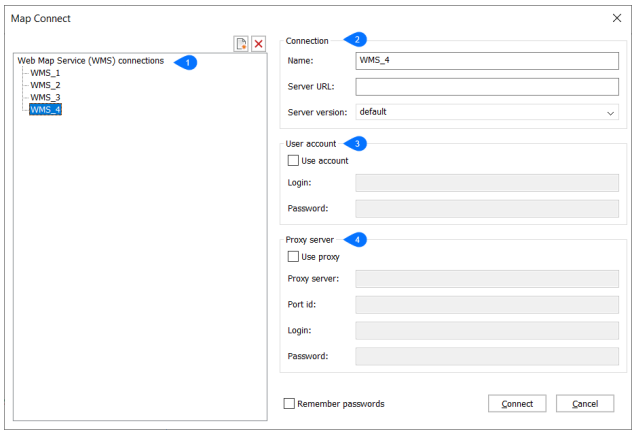
Visualizes all entities that are selected.

4.12.94.7 Select invisible entities

Hides all entities that are selected.

4.12.95 Map Connect dialog box

The Map Connectdialogboxallowsyoutoset up a connection with a Web Map Service.



1 WMS connections

2 Connection

3 User account

4 Proxy server

4.12.95.1 WMS connections

Displays a list of the different possible Web Map Service connections.

4.12.95.2 Connection

Allows you to create a Web Map Service (WMS) connection.

Name

Specifies the name for the new connection.

Server URL
Specifies the URL of the server you want to connect to.

Server version
Specifies the version of the server or else select one from the drop-down list.

4.12.95.3 User account

Allows you to log in to your account.

Use account
Toggles whether the login or password is used when making the connection.

Login
Specifies the login information.

Password
Specifies the password of your account.

4.12.95.4 Proxy server

Allows you to enter the information about a proxy server.

Use proxy
Toggles whether proxy is used to connect to the Web Map Server.

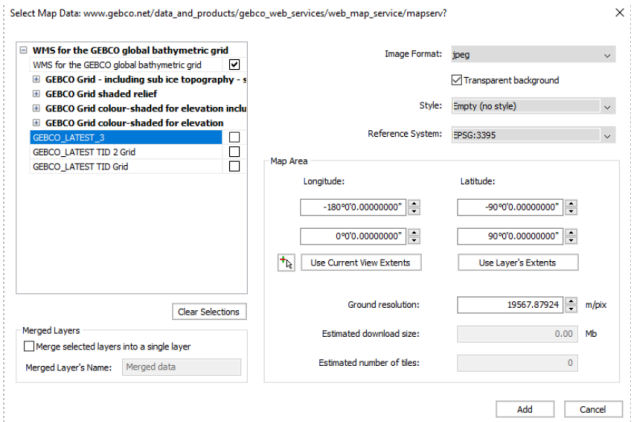
Proxy server
Specifies the URL of the proxy server.

Port id
Specifies the address of the proxy server.

Login
Specifies the login of the proxy server.

Password
Specifies the password of the proxy server.

Remember passwords
Check this box to keep being logged in. When is set to Off the program does not remember your passwords for greater security.
After connecting to a Web Map Server, the following dialog box is displayed:



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.

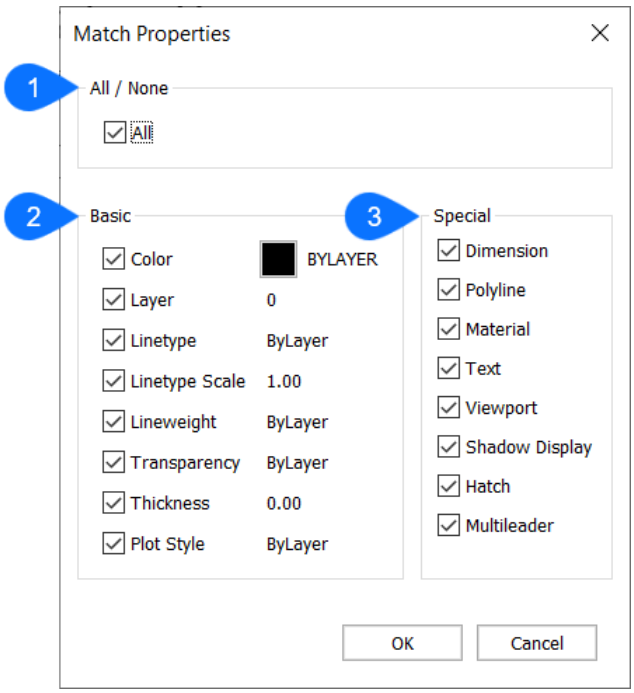
The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved.

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\V21x64\en_US\Support*

The geodatabase.xml files lists cities and their WGS84 geo coordinates used by the **Choose Location** function of the Geographic Location dialog box.

4.12.96 Match Properties dialog box

The Match Properties dialog box allows you to select which properties you want to copy from one entity to one or more other entities.



1 All/None

2 Basic

3 Special

4.12.96.1 All/None

When checked on, all properties are automatically selected. When checked off, all entities are deselected.

4.12.96.2 Basic

Selects the basic entity properties to copy. On the right side, the entity properties of the source entity are displayed.

Color
When checked on, the color of the source entity will be copied to the selected entities. The color of the source entity is displayed on the right.

Layer
When checked on, the layer of the source entity will be copied to the selected entities. The layer of the source entity is displayed on the right.

Linetype
When checked on, the linetype of the source entity will be copied to the selected entities. The linetype of the source entity is displayed on the right.

Linetype Scale
When checked on, the linetype scale of the source entity will be copied to the selected entities. The linetype scale of the source entity is displayed on the right.

Lineweight
When checked on, the lineweight of the source entity will be copied to the selected entities. The lineweight of the source entity is displayed on the right.

Transparency
When checked on, the transparency of the source entity will be copied to the selected entities. The transparency of the source entity is displayed on the right.

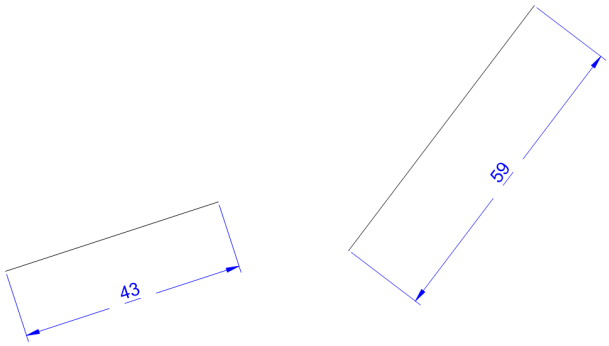
Thickness
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

Plot Style
When checked on, the plot style of the source entity will be copied to the selected entities. The plot style of the source entity is displayed on the right.

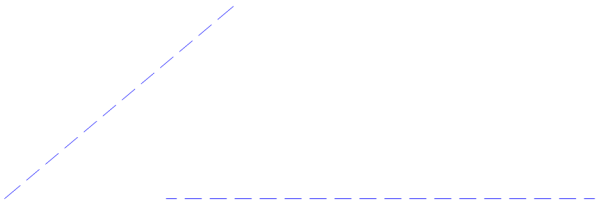
4.12.96.3 Special

Selects other less basic properties to copy.

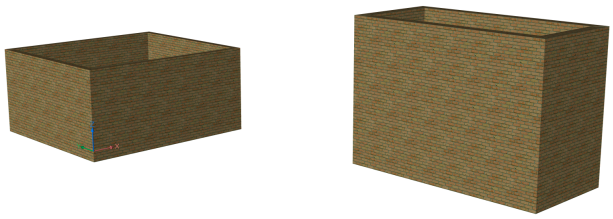
Dimension
When checked on, the dimension style and annotative properties of dimensions, leaders and tolerances is copied.



Polyline
When checked on, the width and linetype generation settings are copied. Other settings are ignored, such as taper and spline.



Material
When checked on, the material properties are copied. Adds or removes materials, depending on whether the source entities has a material definition or not.



Text
When checked on, text style and annotative property of text and mtext is copied. Other properties, like the color, are not copied.



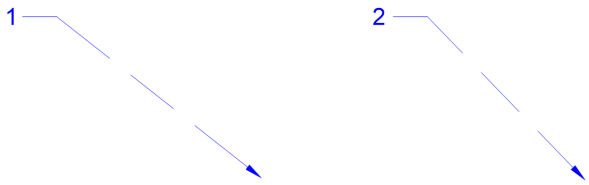
Viewport
When checked on, the viewport-specific properties are copied, such as on or off, snap and grid status, and scale factor. Other properties, such as clipping or frozen layer states, are not copied.

Shadow Display
When checked on, the shadows of the source entity are copied.

Hatch
When checked on, all hatch properties and annotative scaling are copied.

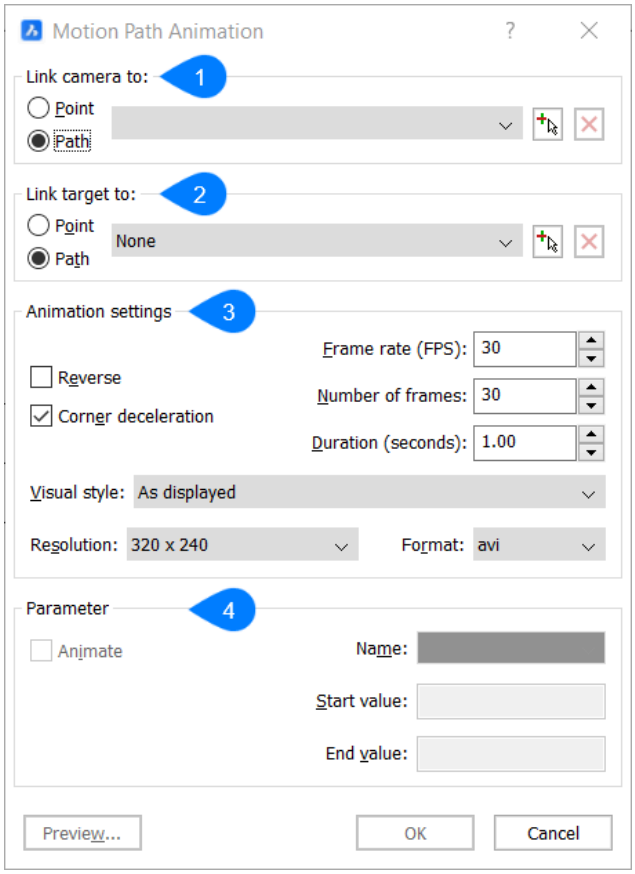


Multileader
When checked on, the multileader style and annotative scaling is copied.



4.12.97 Motion path animation

The Motion Path Animation dialog box allows you to animate a camera moving along a path or panning in a 3D model and saves it to a movie file.



1 Link camera to

2 Link target to

3 Animation settings

4 Parameter

4.12.97.1 Link camera to

Defines the location of the camera by selecting a point or a path in the model space.

Nota: After selecting a point or path, the option to name the entity is available.

Point
Links a camera to a static point in the drawing.

Path
Links a camera to a linear entity in the drawing.

4.12.97.2 Link target to

Defines the target point of the camera by selecting a point or path in the model space.

Nota: After selecting a point or path, the option to name the entity is available.

Nota: 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 should be linked to either a point or a path.

Point
Links the target to a static point in the drawing.

Path
Link the target to a linear entity in the drawing.

4.12.97.3 Animation settings

Allows to control the output of the movie file.

Frame rate
Defines the number of frames per second. When the frame rate is changed, the number of frames will be adjusted according to the duration.

Number of frames
Specifies the total number of frames in the movie. When the number of frames is changed, the duration will be changed accordingly.

Duration
Specifies the duration of the movie in seconds. When the duration is changed, the number of frames will be changed accordingly.

Reverse
Reverses 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 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.

4.12.97.4 Parameter

Allows to animate the change of a parameter.

Animate
Check the box to create realistic movies with moving mechanisms.

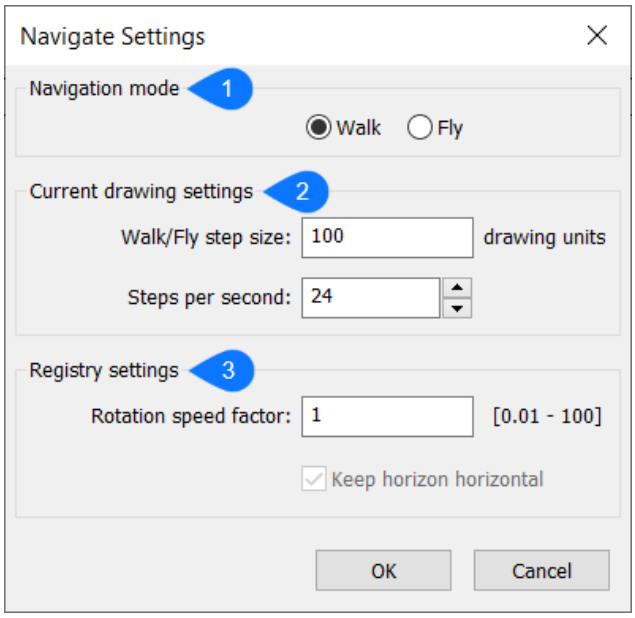
Name
Specify the name of the movie.

Start value
Specifies a start frame for the selected parameter.

End value
The parameter step is set to the Start-End range divided by the number of frames. On each frame, the parameter value should be incremented with the step.

4.12.98 Navigate Settings dialog box

The Navigate Settings Dialog box allows you to adjust the navigation settings in a drawing.



1 Navigation mode

2 Current drawing settings

3 Registry settings

4.12.98.1 Navigation mode

Toggles between Walk and Fly modes:

- Flight mode: permits movements in all three dimensions.
- Walk mode: restricts movement to the xy-plane.

4.12.98.2 Current drawing settings

Sets the settings in the current drawing, and so will be different in other drawings.

Walk/Fly step size
Specifies the distance the view travels with each press of the key. The distance is measured in drawing units.

Steps per second
Specifies how quickly the view changes when a travel key is held down.

4.12.98.3 Registry settings

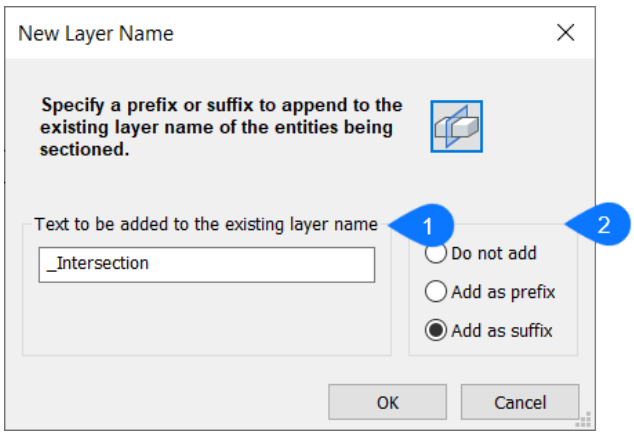
Settings which are saved with the registry, they will be the same in other drawings.

Rotation speed factor
Specifies how quickly the view rotates.

Keep horizon horizontal
When toggled on, the flight level is kept. This option is grayed-out for Walk mode.

4.12.99 New Layer Name dialog box

The New Name dialog box allows you to specify the text which is added to the name of the layer when new layers are created as a copy from the existing layer for section planes.



1 Text to be added to the existing layer name

2 Manner to add the text

4.12.99.1 Text to be added to the existing layer name

Specifies the text that will be added to the existing layer name.

4.12.99.2 Manner to add the text

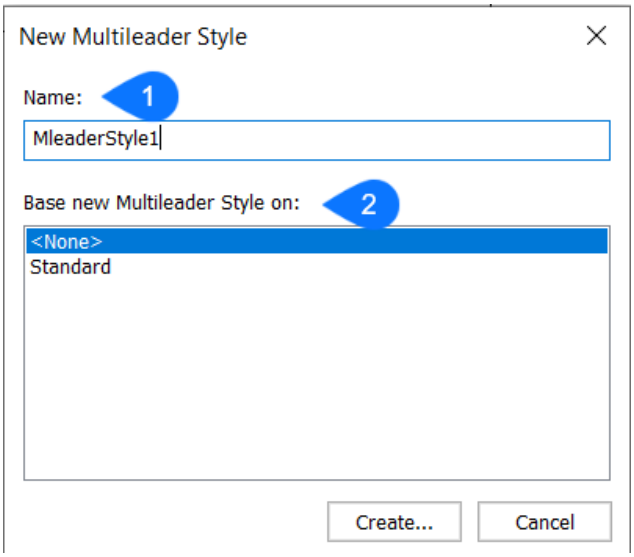
Do not add
No text will be added to name of the layer when a copy is created.

Add as prefix
The specified text will be added as a prefix to the existing name.

Add as suffix
The specified text will be added as a suffix to the existing name.

4.12.100 New Multileader Style dialog box

The New Multileader Style dialog box allows you to create a new multileader style.



1 Name

2 Base new Multileader Style on

4.12.100. Name

1

Sets the name of the new multileader style.

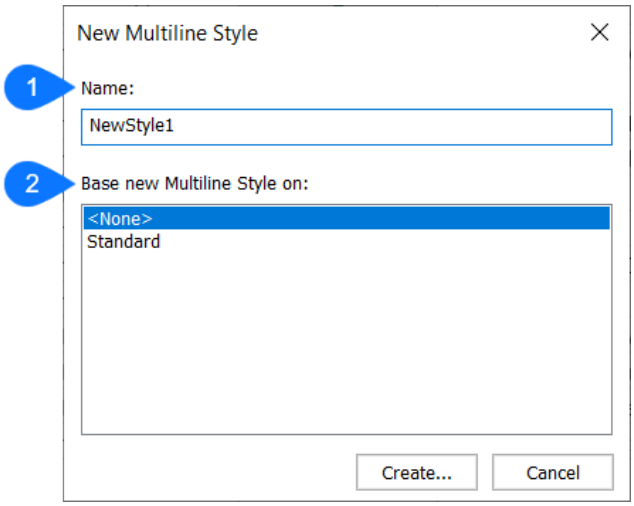
4.12.100. Base new Multileader Style on

2

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

4.12.101 New Multiline Style dialog box

The New Multiline Style dialog box allows you to create a new multiline style definition.



1 Name

2 Base new Multiline Style on

4.12.101. Name

1

Sets the name of the new multiline style.

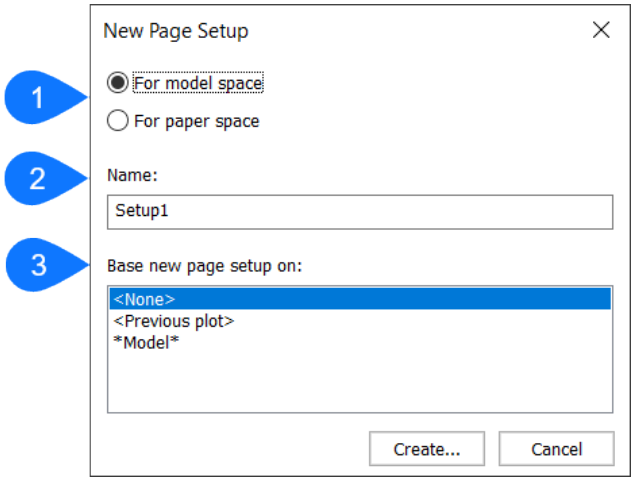
4.12.101. Base new Multiline Style on

2

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

4.12.102 New Page Setup dialog box

The New Page Setup dialog box allows you to create a new page setup for the paper space or model space.



1 Model space or Paper space

2 Name

3 Base new page setup on

4.12.102. Model space or Paper space

1

Sets the space to create a new page setup to the model space or paper space.

4.12.102. Name

2

Sets the name of the new page setup.

4.12.102. Base new page setup on

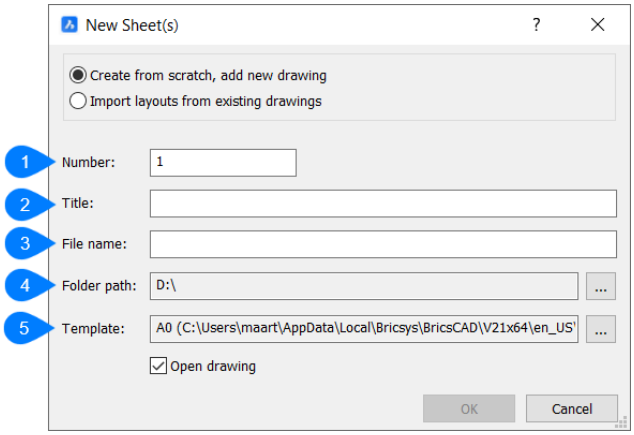
3

Selects an existing page setup to create a new one as a copy of the selected one. To start from scratch, select <None>. To use the previous plot, select <Previous plot>.

4.12.103 New Sheet(s) dialog box

The New sheets dialog box allows you to create (a) new sheet(s) within an existing sheet set.

When starting from scratch, the dialog box includes 5 areas. By selecting the Import layouts from existing drawings option, the dialog box includes 4 areas.



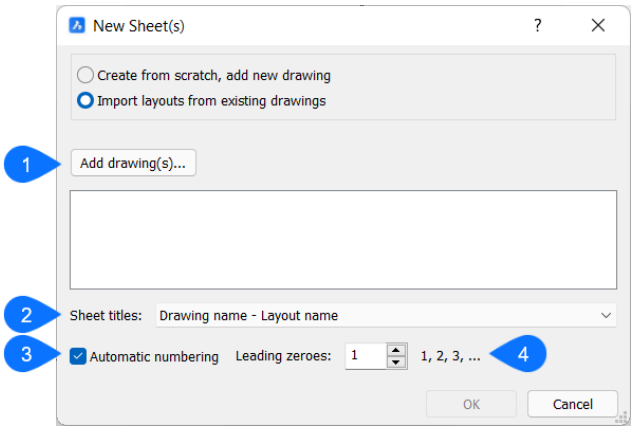
1 Number

2 Title

3 File name

4 Folder path

5 Template



1 Add drawing(s)...

2 Sheet titles

3 Automatic numbering

4 Leading zeroes

4.12.103. Create from scratch

1

Allows you to create a new sheet from scratch

- **Number**: defines the number of sheets.
- **Title**: defines the title of the new sheet(s).
- **File name**: defines the name of the file in which the new sheet(s) will be saved.
- **Folder path**: defines the location of the file.
- **Template**: choose the template of the new sheet(s).

4.12.103. Import layout

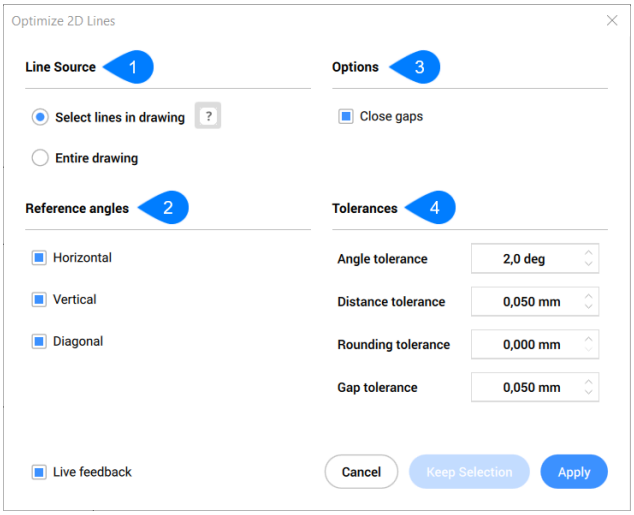
2

Allows you to create a new sheet with properties of an existing layout

- **Add drawing(s)...**: selects drawings to import a layout from trough the Select Drawing(s) dialog box.
- **Sheet titles**: specifies how the selected drawing should be listed by selecting one of the options in the drop-down list.
    - **Drawing name**: displays the drawing name of the selected drawings.
    - **Drawing name- Layout name**: displays the drawing name and the layout name of the selected drawings.
    - **Layout name**: displays the layout name of the selected drawings.
- **Automatic numbering**: numbers the layouts automatically.
- **Leading zeroes**: specifies how the Number value of new sheets is formatted by adding zeroes at the start of a sheet number.

4.12.104 Optimize 2D lines dialog box

The Optimize 2D Lines dialog box allows you to correct inaccuracies in drawings, such as small gaps between lines or near vertical, horizontal and diagonal lines. It includes 4 areas.



1 Line Source

2 Reference angles

3 Options

4 Tolerances

4.12.104. Line Source

1

Specifies the lines in the drawing to work with.

Select lines in drawing

Selects line entities in drawing trough the model space. Press enter when done.

Entire drawing

All entities in the drawing are taken into account.

4.12.104. Reference angles

2

Specifies the angles for which lines will be corrected.

Horizontal

Corrects near horizontal lines according to the angle tolerance.

Vertical

Corrects near vertical lines according to the angle tolerance.

Diagonal

Corrects near 45° lines according to the angle tolerance.

Live feedback

Highlights the lines which match the selected options.

4.12.104. Options

3

Close gaps

Closes small gaps between line entities when checked.

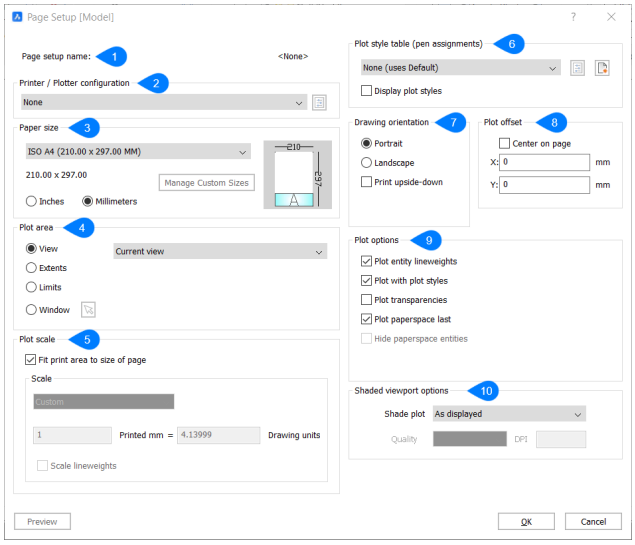
4.12.104. Tolerances

4

Specifies the tolerance value.

4.12.105 Page setup dialog box

The Page Setup dialog box allows you to edit the page setup of the current layout or model space in a dialog box.



1 Page setup name

2 Printer / Plotter configuration

3 Paper size

4 Plot area

5 Plot scale

6 Plot style table

7 Drawing orientation

8 Plot offset

9 Plot options

10 Shaded viewport options

4.12.105. Page setup name

1

Displays the name of the current setup.

4.12.105. Printer / Plotter configuration
2

Selects the printer or plotter.

4.12.105. Paper size
3

Specifies the size of paper by selecting a standard size from the drop-down list. These are sizes supported by the printer.

Nota: The size can be displayed in inches or millimeters by checking one of the two options.

4.12.105. Plot area
4

Specifies which area of the drawing to print.

View
Prints the current view or a named view. Choose a view from the drop-down list.

Extents
Prints the extents of the drawing, which ensures every visible entity is printed.

Nota: 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.

4.12.105. Plot scale
5

Scales the drawing to fit to the paper.

Fit print area to size of paper
Calculates the scale automatically, taking into account the print area of the drawing.

Scale
Specifies the scale factor to use for plotting.

Scale lineweights
Scales lineweights with respect to the plot scale.

4.12.105. Plot style table
6

Specifies the plot style table to use for the printed output, which assigns properties to "pens," colors, and entities.

4.12.105. Drawing orientation
7

Specifies the orientation of the drawing on rectangular paper.

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.

4.12.105. Plot offset 8

Specifies the offset distance for the print.

Nota: To align the print in the center of the page, check the Center on page option.

Center on page
Centers the print on the page, taking into account margins.

4.12.105. Plot options 9

Plot entity lineweights
Toggles the use of lineweights.

Plot with plot styles
Toggles the use of plot styles.

Plot transparencies
Plots entities who have been assigned the transparency property, either by entity or by layer.

Plot paperspace last
Specifies the printing order.

Nota: This option is only available when printing layouts.

Hide paperspace entities
Removes hidden lines from 3D entities in paper space when checked.

Nota: This option is disabled when printing model space.

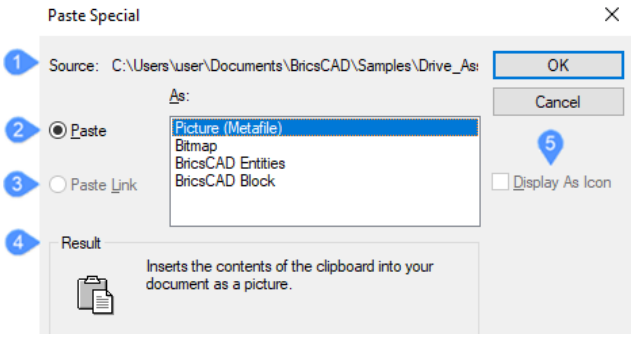
4.12.105. Shaded viewport options 10

Override the visual style of the current view when printing model space.

Shade plot
Choose a shade style to plot the drawing with.

4.12.106 Paste special dialog box

The **Paste Special** dialog box allows you the select the type of object to paste from the Clipboard into the drawing. This command is available on Windows only. Shortcut Keystroke: CTRL+ALT+V.



1 Source

2 Paste As

3 Paste Link

4 Result

5 Display As Icon

4.12.106. Source

1

Displays the path where the drawing/entity/block can be found.

4.12.106. Paste As

2

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 WMF (Windows Metafile) format image and then pasted as an OLE (Object Linking and Embedding) image.
- **Bitmap**: Entities are converted to a BMP format image and then pasted as an OLE image.
- **BricsCAD Entities**: pastes as individual entities or table entries including layers, linetypes, etc.
- **BricsCAD Block**: inserts the contents of the clipboard into your document as a block.

Paste As BricsCAD Entities

Pastes CAD entities into the drawing as BricsCAD entities.

Select insertion point or [Rotate/Scale/Mirror]: - *(enter an option)*

1 **Select insertion point** – *(specifies 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:*

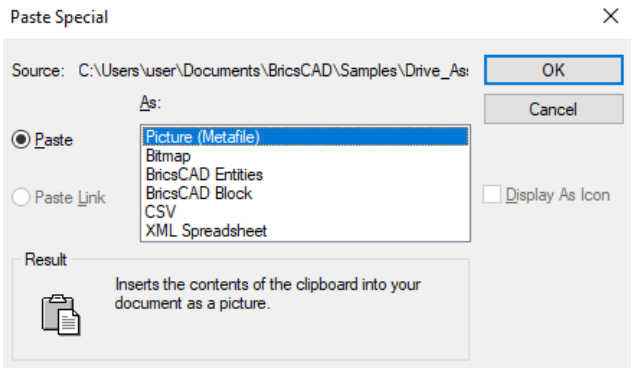
Select mirroring control point – *(Pick a second point for the mirror line)*

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.

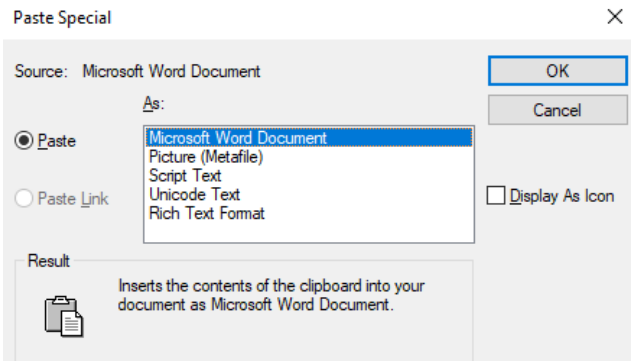
Paste As BricsCAD Block

Opens the **Insert Block** dialog box.

Here is the same dialog box, but it has some new options as the entities are copied from an Excel spreadsheet.



If you copy a text string from outside of BricsCAD, there are some new options as shown in the picture below:



4.12.106. Paste Link

3

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.

4.12.106. Result

4

Displays a short description of the selection option you want to use.

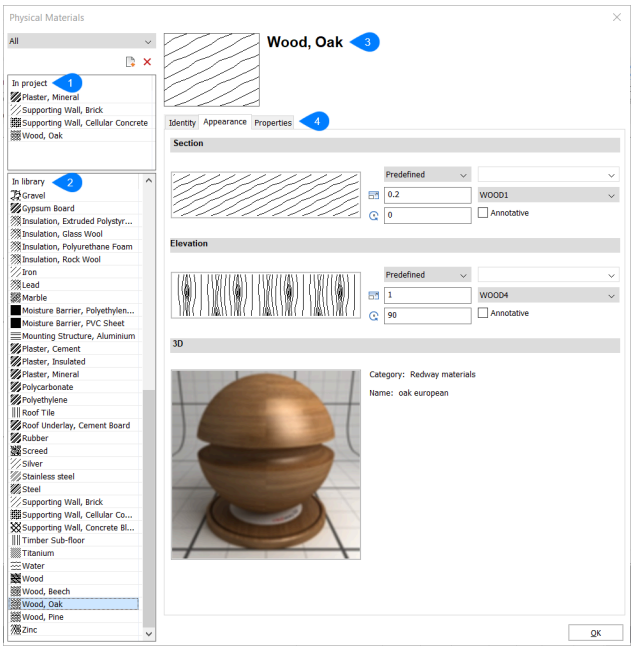
4.12.106. Display As Icon

5

This option is not available for CAD entities. Displays the pasted entities as an icon identifying their source application.

4.12.107 Physical materials dialog box

The Physical Materials dialog box opens the Physical Materials library in the current project and the referenced library.



1 In project

2 In library

3 Material name

4 Material specification tabs

4.12.107. In project

1

Displays all materials which are currently loaded in the project.

4.12.107. In library

2

Displays all materials which are loaded in the library.

4.12.107. Material name

3

Displays the name of the selected material.

4.12.107. Material specification tabs

4

Material specification tabs

Lists the broader specifications of the selected material into three tabs.

Identity

- **Name**: specifies the name of the material.
- **Class**: specifies the material class the material belongs to.
- **Description**: displays a short description of the material.

Appearance

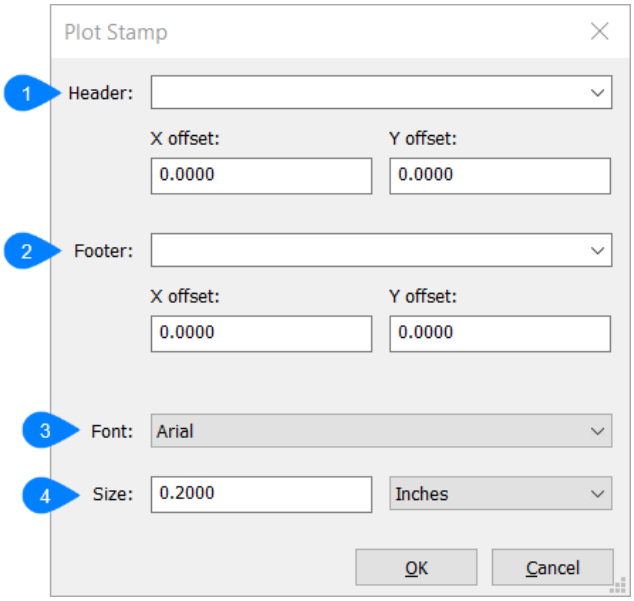
- **Section**: specifies the hatch pattern used to display the material in a section view.
- **Elevation**: specifies the hatch pattern used to display the material in an elevation view.
- **3D**: shows the three dimensional texture of the material that will be used in all 3D views.

Properties

Specifies additional properties of the material such as the density.

4.12.108 Plot stamp dialog box

The Plot Stamp dialog box allows you to add information about drawings and prints to the edge of the plotted drawing.



1 Header

2 Footer

3 Font

4 Size

4.12.108. Header

1

Appears at the top of the plot. A meta-phrase for the header can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

4.12.108. Footer

2

Appears at the bottom of the plot. A meta-phrase for the footer can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

4.12.108. Font

3

Specifies the font used for the plotstamp.

4.12.108. Size

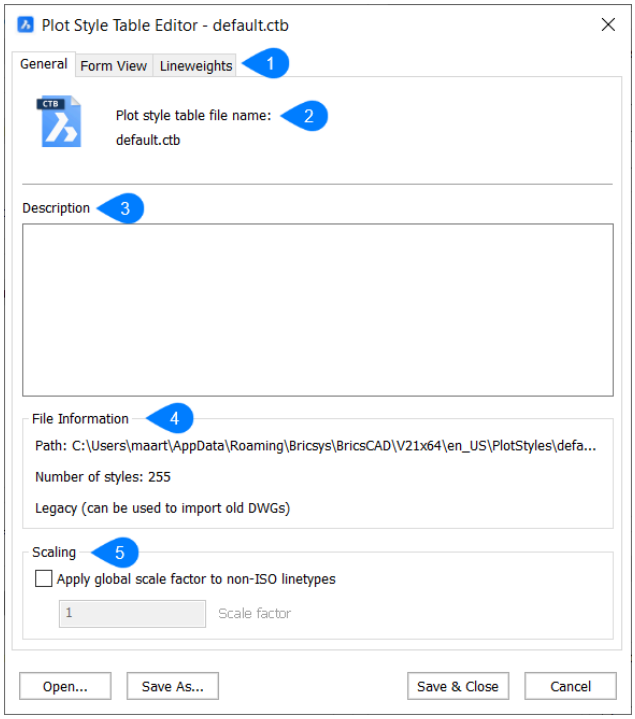
4

Specifies the height of the text.

- Inches: specifies the height of the text in inches.
- Millimeters: specifies the height of the text in millimeters.

4.12.109 Plot style table editor dialog box

The Plot Style Table Editor displays the plot styles contained in the specified plot style table.



1 Editor tabs

2 Plot style table file name

3 Description

4 File Information

5 Scaling

4.12.109. Editor tabs

1

The Plot Style Table Editor includes the following tabs.

General tab

The General tab lists general information about the plot style table.

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.

Lineweights tab

The Lineweights tab lists the available lineweights.

4.12.109. Plot style table file name

2

Displays the name of the plot style table file you are editing.

4.12.109. Description

3

Displays a description you want to include about the plot style table.

4.12.109. File Information

4

Displays information about path, number of plot styles and the version number of the Plot Style Table Editor.

4.12.109. Scaling

5

Apply global scale factor to non-ISO linetypes

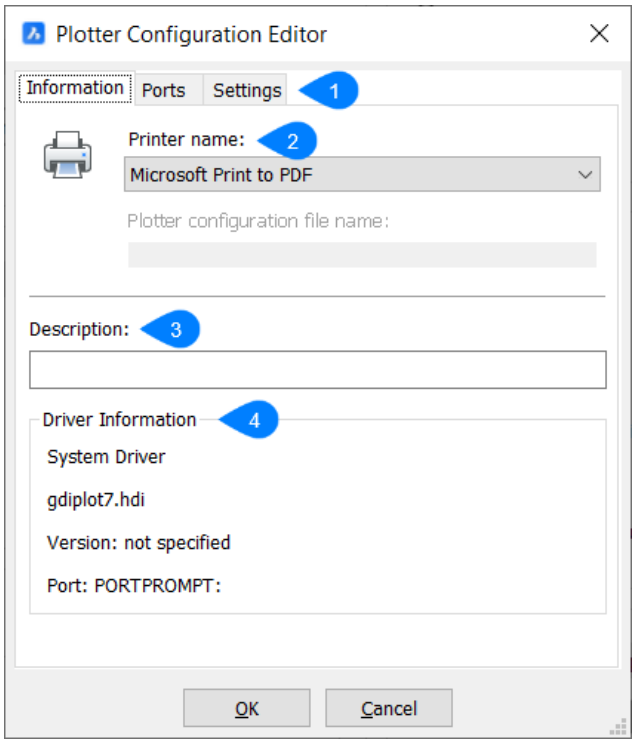
Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.

Scale factor

Specifies the factor by which to scale non-ISO linetypes and fill patterns.

4.12.110 Plotter Configuration Editor dialog box

The Plotter Configuration Editor dialog box allows you to create and edit customized parameter PC3 files for printers and other output devices.



1 Editor tabs

2 Printer name

3 Description

4 Driver Information

4.12.110. Editor tabs

1

Information

Gives basic information about the plotter.

Ports

Gives options on how the file will be printed.

Settings

Provides additional settings to alter. The settings available depend on the capabilities of the printer selected.

4.12.110. Printer name

2

Lists the names of printers supported by the operating system.

4.12.110. Description

3

Describes the plotter configuration. It is displayed by tooltips.

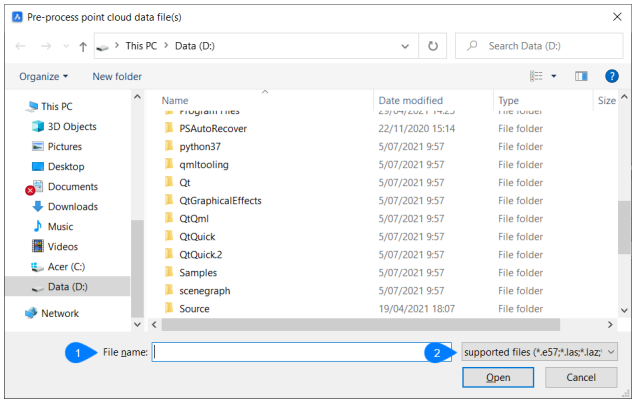
4.12.110. Driver Information

4

Displays basic information about the driver.

4.12.111 Pre-process point cloud data file(s) dialog box

The Pre-process Point Cloud Data File's dialog box allows you to attach point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing.



1 File Name

2 Supported files

4.12.111. File Name

1

Specifies the name of the point cloud data file you want to attach.

Nota: A manual search can be done in the window above.

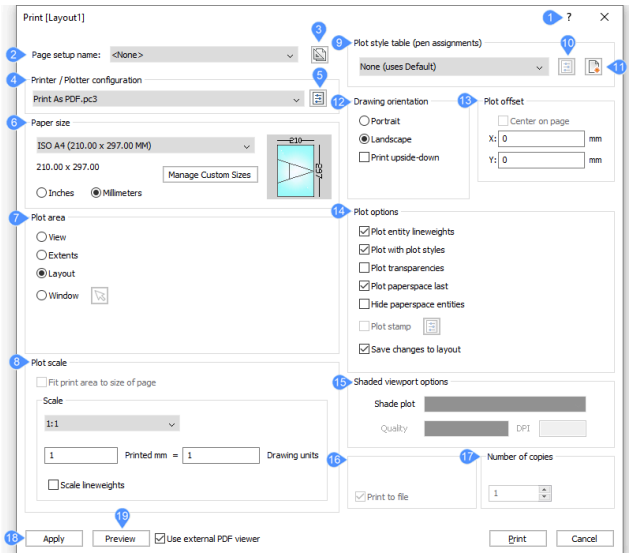
4.12.111. Supported files

2

Specifies the file types you will be able to choose from.

4.12.112 Print dialog box

The **Print** dialog box allows you to print and preview drawings to plotters and files.



1 Command reference

2 Page setup name

3 Create page setup

4 Printer/Plotter configuration

5 Edit plotter configuration

6 Paper size

7 Plot area

8 Plot scale

9 Plot style table

10 Edit Plot Style

11 Create New Plot Style

12 Drawing orientation

13 Plot offset

14 Plot options

15 Shaded viewport options

16 Print to file

17 Number of copies

18 Apply

19 Preview

4.12.112. Command reference

1

Opens the Bricsys Help article about the PRINT command.

4.12.112. Page setup name 2

You can select plot settings used previously from the drop-down list:

- **<None>**: Uses options saved in the current layout or model space.
- **<Previous Plot>**: Uses options that were saved from the last time when this dialog box was used.
- Names of other page setups stored in the drawing.

4.12.112. Create page setup 3

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

4.12.112. Printer/Plotter configuration 4

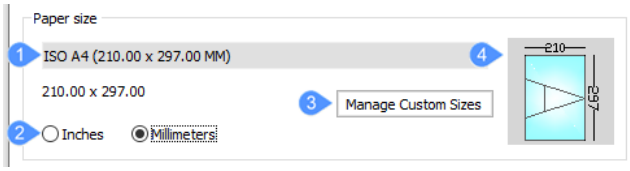
Allows you to select from the drop-down list the printer or plotter.

The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices -- as well as predefined printer parameters stored in PC3 files.

4.12.112. Edit plotter configuration 5

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3files. See the PLOTTERMANAGER command.

4.12.112. Paper size 6



1 Standard Paper Sizes list

2 Units

3 Manage Custom Sizes

4 Preview

Standard Paper Sizes list

Specifies the size of paper. You can select standard sizes listed by the drop-down list; these are sizes supported by the printer.

While printers may appear to support many different sizes, you should select only the size of paper that is in the printer.

Units

- **Inches**: Use imperial units for measurements involving printing.
- **Millimeters**: Use metric units for measurements involving printing.

Manage Custom Sizes

Nota: This option is only available for the PDF output.

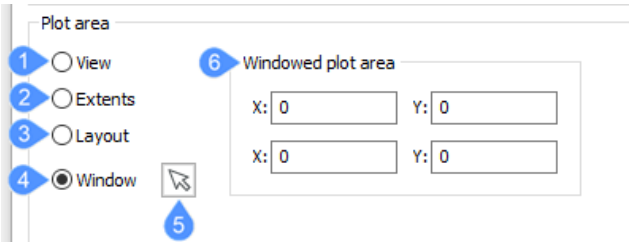
Manages custom paper sizes when Print as PDF.pc3 is selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.

Preview

This section indicates the size, position and orientation of the current plot area on the selected paper size.

4.12.112. Plot area

7



1 View

2 Extents

3 Layout

4 Window

5 Select area to be printed

6 Windowed plot area

View

Prints the current view or a named view. Choose a view from the drop-down list. Use the VIEW 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 considered when calculating the extents.

Layout

Prints the current layout.

Window

Prints a rectangular area of the drawing.

- You define the rectangular area by entering the x, y coordinates or by clicking the **Select Area to be Printed** button.
- The x, y coordinates resulting from the selection or from entering in the Command line are added to the **Windows Plot Area** boxes. You can edit the values in a convenient way.

4.12.112. Plot scale
8

Scales the drawing to fit it to the paper:

- **Fit print area to the size of paper**: Scale is calculated automatically by the program, considering 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 drop-down list or choose the **Custom** option that will allow you to enter your own scale factors in **Printed inches/mm** and **Drawing units** fields. The scale factor shown can be edited with the SCALELISTEDIT command.
  - For scale factors like 1:5, the drawing is printed smaller.
  - For scale factors like 5:1, the drawing is printed larger.
- **Scale lineweights**: When on, lineweights are scaled with respect to the plot scale.

4.12.112. Plot style table
9

Specifies the plot style table to use for the printed output, which assigns properties to pens, colors, and entities.

- 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?

- **Yes**: Assigns the CTB or STB plot style file to all layouts.
- **No**: Assigns the plot style file to the current layout only.

4.12.112. Edit Plot Style
10

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

4.12.112. Create New Plot Style
11

Opens the **Add Plot Style Table** dialog box to create new plot styles. See the STYLESMANAGER 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).

4.12.112. Drawing orientation
12

Specifies the orientation of the drawing on rectangular paper:

- **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.

4.12.112. Plot offset
13

Specifies the offset distance for the print.

- **Center on Page**: Centers the print on the page, considering 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:
  - **Positive values**: Move the drawing up and to the right.
  - **Negative values**: Move 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.

4.12.112. Plot options
14

- **Plot entity lineweights**: Toggles the use of lineweights:
  - On: Mimics the 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 and overrides lineweight settings.
  - Off: Does not use plot styles.
- **Plot Transparencies**: Plots entities who have been assigned the Transparency property, either by-entity or by layer.
- **Plot Paperspace Last**: Specifies the printing order:
  - 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 command. This option is unavailable when the Printer/Plotter configuration is set to *Print As PDF.pc3*.
  - 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 the Use Plot Settings From drop-down list.
  - On: Saves changes made to this dialog box with the layout.
  - Off: Does not save changes.

4.12.112. Shaded viewport options 15

Override the visual style of the current view when printing model space:

Choose from the drop-down list a shade style to plot the drawing with.

This option is disabled when printing a paper space layout. The **Shade Plot** mode of a paper space viewport is defined in the **Shade Plot** property of the viewport. The quality of the **Rendered** option is defined through the current render preset; see the RENDERPRESETS command.

4.12.112. Print to file 16

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

4.12.112. Number of copies 17

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

4.12.112. Apply 18

Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

4.12.112. Preview 19

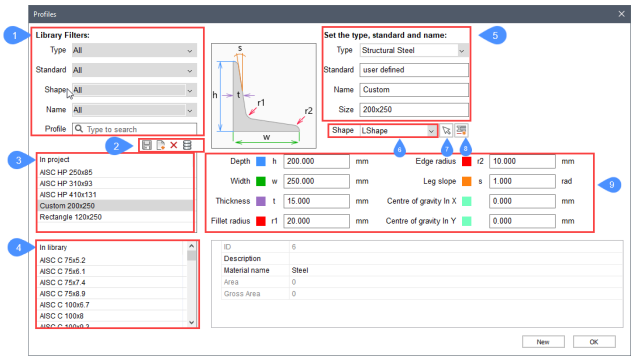
Displays a preview of the print. In the preview window, click the **Print Settings** button to return to this dialog box. See the PREVIEW command.

> ***Nota:*** On Mac or Linux, printing will always print to PDF. So, a PDF document will be generated that then still needs to be sent to the printer for physical printing.

On MAC and Linux there is a relevant PAGESETUP command.

4.12.113 Profiles dialog box

Opens the profiles library in the current project and the referenced library.



1 Library Filters

2 Tools

3 In Project

4 In Library

5 Custom fields

6 Preselect shape

7 Pick profile in model

8 Set profile offset

9 Properties of the profile

4.12.113. Library Filters

1

Allows to filter the profiles list by type, standard, shape or name.

- **Type**: The **Type** options are: **All**, **Generic**, **HVAC**, **Piping** or **Structural Steel**.
- **Standard**: The Localized standard options are: **All**, **AISC** (American Institute of Steel Construction), **BS** (British Steel), **EURO** (European standard steel profiles) and **User Defined**.
- **Shape**: The **Shape** options are: **All**, **Circle Hollow**, **I Shape**, **L Shape**, **Rectangle**, **Rectangle Hollow**, **T Shape** and **U Shape**. Depending on the selected standard, some shape options are not available.
- **Name**: Filters the profiles by a selected character string.
- **Profile**: Select a character string. Only profiles that contain the selected string are listed.

4.12.113. Tools

2

- **Save Profile**: Save the currently edited profile.
- **New Profile**: Create a new profile.
- **Delete Profile**: Delete the selected profile.
- **Project and Library Information**: Opens the BIM **Project Info** dialog box.

4.12.113. In Project

3

Lists the available profiles used in the current project with respect with above filters.

4.12.113. In Library

4

Lists the available profiles with respect with above filters.

4.12.113. Custom fields

5

Lists the characteristics of the chosen profile or in case of creating a new profile, sets the type, standard and name for it.

4.12.113. Preselect shape

6

Select a shape from the drop-down list.

4.12.113. Pick profile in model

7

Select a closed 2D entity or the boundary entities of an enclosed area.

4.12.113. Set profile offset

8

Sets the profile offset.

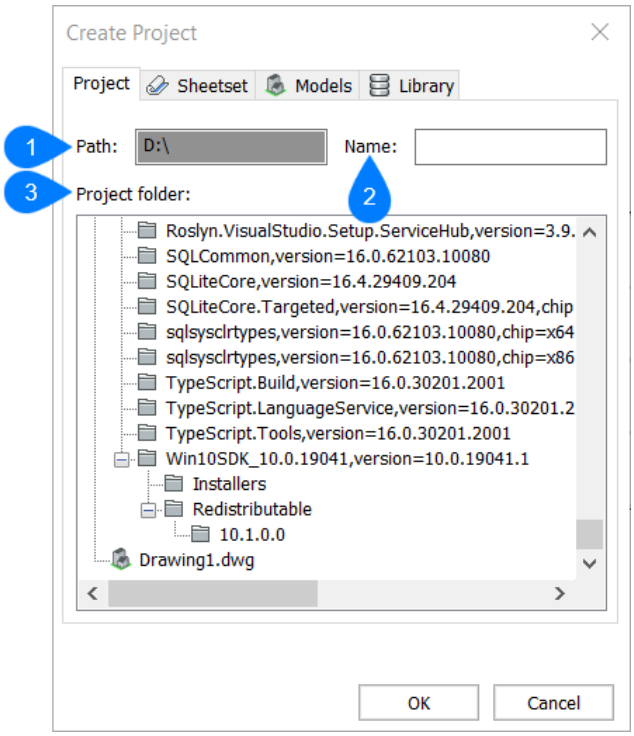
4.12.113. Properties of the profile

9

Shows the properties of the selected profile or sets the properties of a new profile.

4.12.114 Project setup dialog box

The Project Setup dialog box allows you to create a new project and choose the file location.



1 Path

2 Name

3 Project folder

4.12.114. Path

1

Specifies the location of the project by selecting the parental folder.

4.12.114. Name

2

Creates a new project file by entering a new name.

Nota: By default, the name is that of the parental folder.

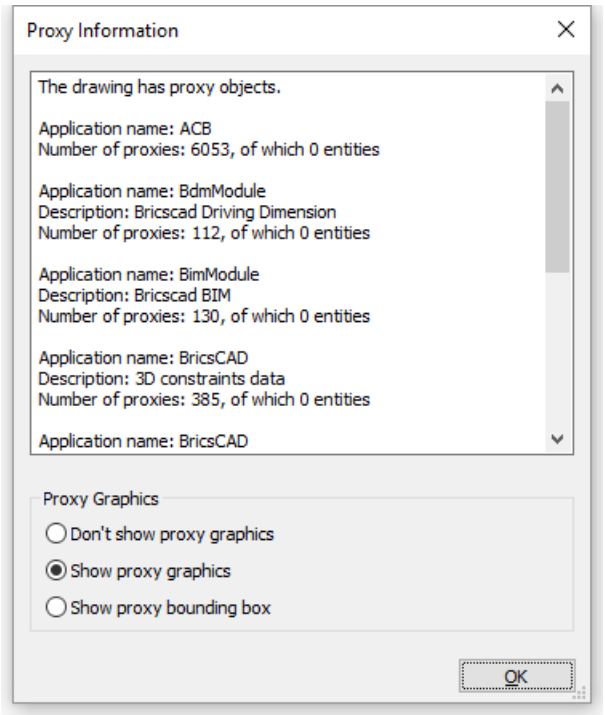
4.12.114. Project folder

3

Specifies the folder in which the project will be saved by selecting the destination folder.

4.12.115 Proxy information dialog box

The **Proxy Information** dialog box displays information about proxy entities and toggles their display in the current drawing.



4.12.115. Proxy Graphics

1

Toggles the display of proxy graphics in the drawing:

- **Don't show proxy graphics**: turns off the display of proxy graphics.
- **Show proxy graphics**: turns on the display of proxy graphics
- **Show proxy bounding box**: displays a rectangle or cube instead of the proxy.

4.12.116 Publish dialog box

The Publish dialog box allows you to send one or more drawings, layouts and sheets to the printer or export them in PDF format. This is useful for batch printing of books of drawings.

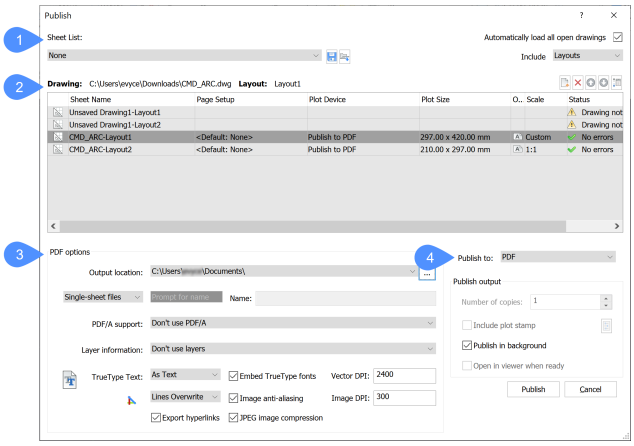
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command

bricscad.exe /pl <"the path of your *.dwg file"> <"the path of your *.dsd file">

For example:

bricscad.exe /pl "C:\Users\user\Desktop\doc.dwg" "C:\Users\user\Desktop\doc.dsd"

Also, in the Page Setups section of the Drawing Explorer dialog box set the printer you use in the Device name.



1 Sheet List

2 Drawing

3 PDF options

4 Publish to

4.12.116. Sheet List

1

Displays a list of sheets. If you have a *.dsd file (drawing sheets for publishing), you can select it and the Load Sheet List dialog box will appear.

Save Sheet List

Saves the current sheet list to a *.dsd (Drawing Set Description) file. Opens the Save Sheet List dialog box.

Load Sheet List

Loads a saved sheet list. Opens the Load Sheet List dialog box. If the current list is not saved yet, you are prompted to save it.

If you are trying to load a file with an existing name, after you press the Open button in the Load Sheet List dialog box you are prompted to replace or append the sheets to the list.

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**: includes only model tabs of open drawings.
- **Layouts**: includes only layout tabs of open drawings.
- **Model & Layouts**: includes both model and layout tabs of open drawings.

4.12.116. Drawing

2

Layout

Page Setup
Specifies the Page Setup name of each sheet. Click the Page Setup name, then click the down arrow to select a different page setup or choose **Import…** to import page setups of another drawing. This will open the Import Pagesetups dialog box.

Plot Device
Displays the name of the plot device, as defined in the Page Setup dialog box of the sheet.

4.12.116. PDF options

3

Output location
Displays the path where the PDF will be saved. Opens the Choose a folder dialog box to change the path.

File
Select an option:

- **Single – sheet files**: each layout is published to a separate *.pdf. The file names are: <drawing>_<layout>.pdf
- **One Multi – sheet file**: layouts are published to a single file. You can choose to specify a name or to be prompted for a name when the publishing procedure starts.

Name options
Select an option:

- **Prompt name**: after you press the Publish button you are prompted to insert the name you want for your *.pdf in the Specify PDF file name dialog box.
- **Specify name**: the field for Name becomes available. Insert here the name you want for your *.pdf.
- **Generate name**: the *.pdf will have the same name as the drawing.

Name
Allows you to type the name you want for your *.pdf. This field is available only for the **Specify name** option.

PDF/A Support
Allows you to select from the drop-down list one of the options.

Layer information

- **Don't use layers**: no layers are exported.
- **Use all layers with visible entities**: only layers which are current ON and Thawed are exported.
- **Use layers with entities, including OFF and FROZEN layers**: all layers containing entities are exported.

TrueType Text
Select an option:

- **As text**: ensures that the text in the PDF looks exactly as in the drawing.
- **As geometry**: converting to geometry increases the file size and pixelation might occur when viewed at a high zoom level.

Embed TrueType fonts
If the font is not embedded, a substitute font might be used by the PDF viewer.

Nota: Licensed fonts cannot be embedded.

Vector DPI
Controls the resolution of vector graphics and gradients.

Merge control
Determine whether overlapping lines are merged or lie on top of each other.
Select an option:

- **Lines Overwrite**: the lines on top overwrite the lines below.
- **Lines Merge**: the colors of crossing lines are blended into a new color.

Image anti-aliasing
If the resolution of an image is too low to match the Image DPI setting, then the raster image is anti-aliased (generate pixels artificially) to make the raster image look more smoothly. Without that setting an image with a too low resolution is embedded "as-is".

Image DPI
Defines the resolution of the resulting image in the PDF for raster images. This setting allows you to lower the resolution of the original raster image to avoid including a gigabyte raster image in the PDF. For printing the resolution should match the output resolution of the printer. For on-screen viewing high resolutions (2400 dpi +) are required.

Export hyperlinks
If checked, it allows you to export hyperlinks in the *.pdf.

JPEG image compression
This setting applies to all included raster images in the PDF, both raster images and rendered views. It applies a JPEG compression to these images to reduce the PDF file size. When switched off, the images are embedded as raw bitmaps. The JPEG compression is lossy compared to the raw bitmap introducing small imperfections.

4.12.116. Publish to

4

Nota: When **Publish to** is set to **Plotter named in page setup**, the PDF options from this dialog box disappear.

Number of copies
Sets the number of copies for each sheet to be printed.

Include plot stamp
If this option is checked, it adds a plot stamp.

Plot Stamp
Opens the Plot Stamp dialog box.

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.

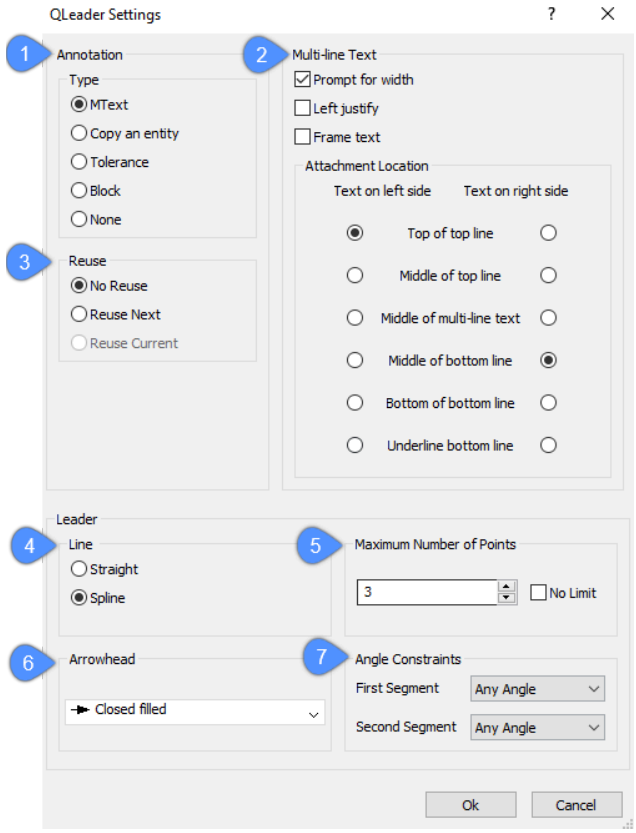
Open in viewer when ready

Determines if the sheet set will be opened in the viewer when it is ready.

Nota: This option is not available when **Publish to** is set to **Plotter named in page setup**, only when **Publish to** is set to **PDF**.

4.12.117 QLeader Settings dialog box

The **Qleader Settings** dialog box allows you to specify the properties of leaders.



1 Annotation

2 Multi-line Text

3 Reuse

4 Leader Line

5 Maximum Number of Points

6 Arrowhead

7 Angle Constraints

4.12.117. Annotation

1

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.

  Nota: 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 (than the leader itself) in the drawing as the annotation.
- **Tolerance**: Uses tolerance notation for the annotation; displays the **Geometric Tolerance** dialog box. See the TOLERANCE command.
- **Block**: Uses a block from the drawing or an external block as the annotation. The prompt displayed is identical to that of the -INSERT command.
- **None**: No annotation is used.

4.12.117. Multi-line Text

2

Nota: This option is only available when the annotation type is MText.

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 to the 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.

4.12.117. Reuse

3

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.

4.12.117. Leader Line 4

Specifies the style of leader line:

- **Straight**: Draws the leader lines as straight segments.
- **Spline**: Draws the leader line as a smooth continuous spline.

4.12.117. Maximum Number of Points 5

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.

4.12.117. Arrowhead 6

Specifies the type of arrow at the start of the leader; choose from the drop-down list:

- Choose an arrowhead type.
- **None**: No arrow is drawn.
- **Select arrow**: Choose a block from those in the 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 command to turn it into a block.

4.12.117. Angle Constraints 7

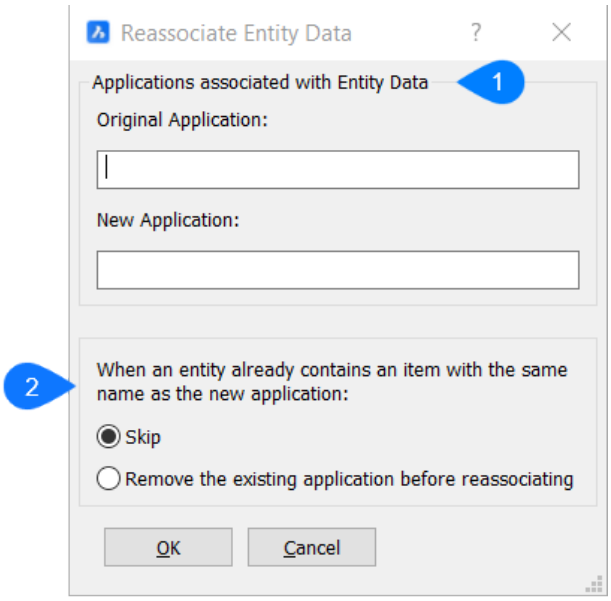
Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees.

Click the drop-down list, 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.

4.12.118 Reassociate entity data dialog box

The Reassociate Entity Data dialog box allows you to associate extended entity data with a specific application.



1 Applications associated with Entity Data

2 Options

4.12.118. Applications associated with Entity Data

1

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.

4.12.118. Options

2

Skip

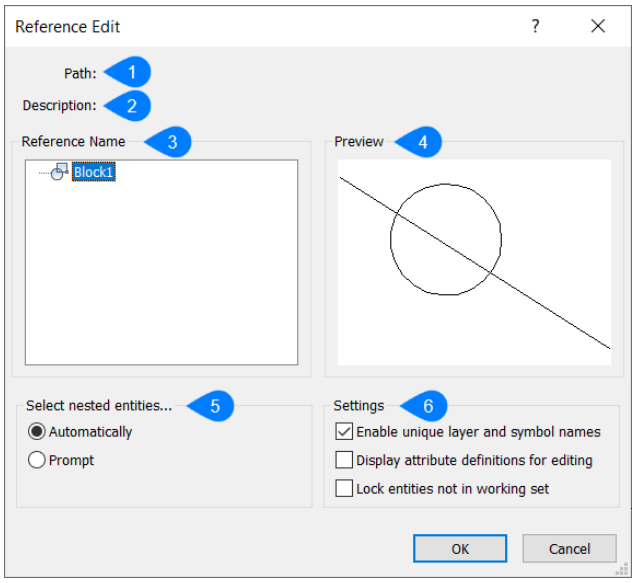
Skips entities that already contains an item with the same name as the new application.

Remove the existing application before reassociating

Removes the existing application before reassociating.

4.12.119 Reference Edit dialog box

The Reference Edit Dialog Box allows you to edit block references and externally-referenced drawings.



1 Path

2 Description

3 Reference Name

4 Preview

5 Select nested entities

6 Settings

4.12.119. Path

1

Displays the name of the path.

4.12.119. Description

2

Displays the description of the path.

4.12.119. Reference Name

3

Lists the name of the block and nested blocks, if any.

4.12.119. Preview

4

Displays a preview of the current reference.

4.12.119. Select nested entities

5

Determines how nested blocks/xrefs are handled.

Automatically

Includes nested entities automatically.

Prompt
Includes nested entities by selecting them after the dialog box is closed by pressing OK.

4.12.119. Settings

6
Determines how entities are treated during editing.

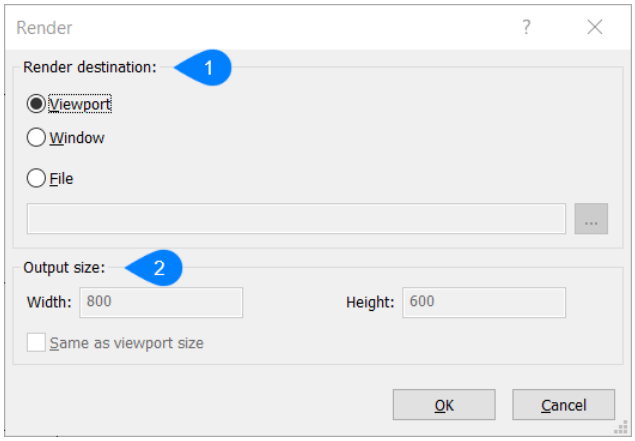
Enable unique layer and symbol names
Toggles the use of 0 in layer and other names.

Display attribute definitions for editing
Toggles the inclusion of attributes.

Lock objects not in working set
Toggles locking of entities not being edited.

4.12.120 Render dialog box

The Render dialog box allows you to generate a photo-realistic rendering using materials and lights.



1 Render destination

2 Output size

4.12.120. Render destination

1
Specifies where the render will appear.

Viewport
Generates the render in the current viewport.

Window
Generates the render in a new window.

File
Saves the render to a file. Selecting the three dots opens the render output file dialog box.

4.12.120. Output size

2

Specifies the size of the rendering, in pixels. Larger sizes provide more detail but take longer to render.

Width

Specifies the width of the rendering, in pixels.

Height

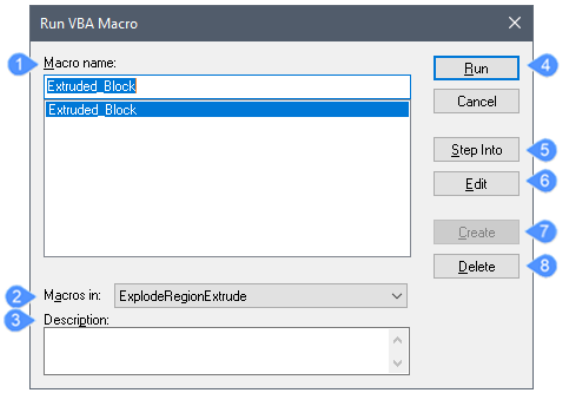
Specifies the height of the rendering, in pixels.

Same as viewport size

Matches the output size or the size of the output window to the current viewport size.

4.12.121 Run BricsCAD VBA macro dialog box

The **Run VBA Macro** dialog box runs, creates, edits, and deletes VBA macros.



1 Macro name

2 Macros in

3 Description

4 Run

5 Step Into

6 Edit

7 Create

8 Delete

4.12.121. Macro name

1

Lists the names of macros found in the project file. Choose one to run.

4.12.121. Macros in

2

Lists the names of projects loaded into the current drawing. Choose one from the drop-down list.

4.12.121. Description
3

Describes the macro. You can edit this text and save it pressing the **Close** button.

4.12.121. Run
4

Runs the selected macro.

4.12.121. Step Into
5

Opens the integrated development environment window. The macro is in debug mode, which means that it runs one line of code at a time.

See the VBAIDE command.

4.12.121. Edit
6

Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code.

See the VBAIDE command.

4.12.121. Create
7

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 the VBA code.

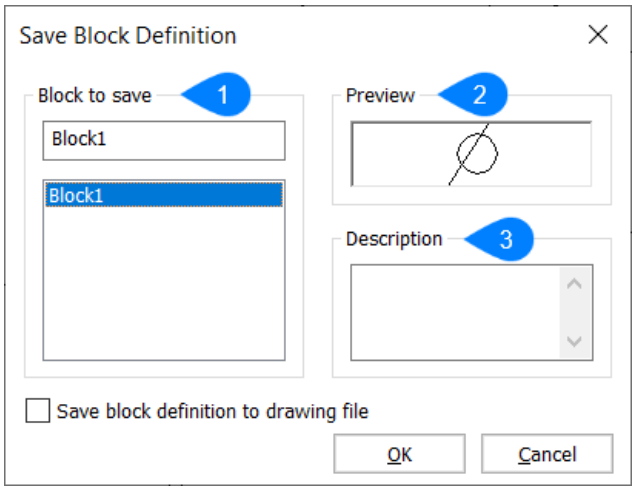
See the VBAIDE command.

4.12.121. Delete
8

Deletes the macro. A dialog asks to confirm the action.

4.12.122 Save block definition dialog box

The Save Block Definition dialog box allows you to copy the current block definition to a new name.



1 Block to save

2 Preview

3 Description

4.12.122. Block to save

1

Specifies a new name for a selected block to make a copy of.

4.12.122. Preview

2

Displays a preview of the selected block.

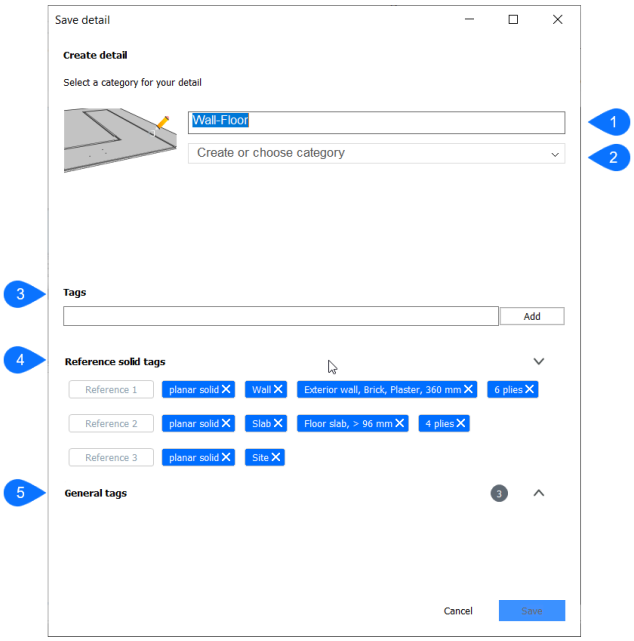
4.12.122. Description

3

Displays the description of the selected block, if it has one.

4.12.123 Save detail dialog box

The Save Detail dialog box allows you to save the selected elements as a detail.



1 Name

2 Category

3 Tags

4 Reference solid tags

5 General tags

4.12.123. Name

1

Sets the name of the new BIM detail.

4.12.123. Category

2

Sets the category of the new BIM detail.

4.12.123. Tags

3

Adds tags to the detail so it can be found easily when searching for this specific detail.

4.12.123. Reference solid tags

4

Displays tags that specify the detail reference entities.

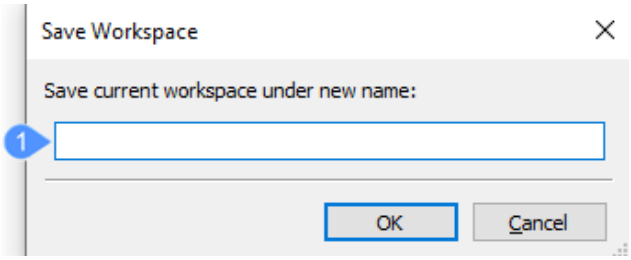
4.12.123. General tags

5

Displays tags that specify the detail in general.

4.12.124 Save workspace dialog box

The **Save Workspace** dialog box allows you save the current workspace under a different name, creating a new workspace.



1 Save current workspace under new name

4.12.124. Save current workspace under new name

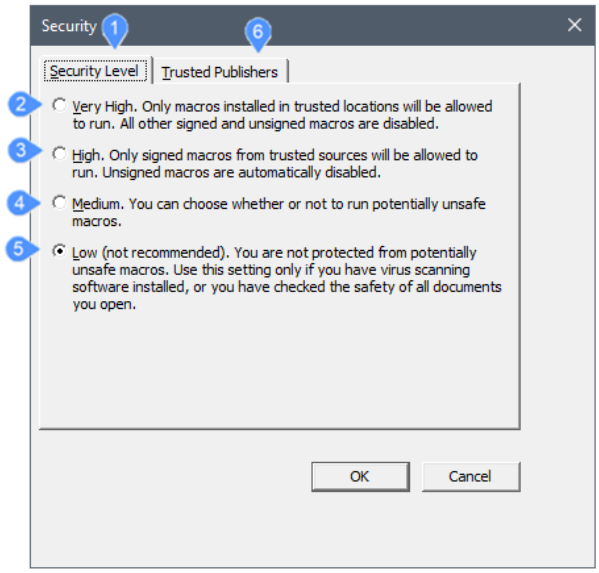
1

Specifies the name for the new workspace. Press **OK** to apply the changes.

You can access it later through the **Workspaces** toolbar or from the status bar.

4.12.125 Security dialog box

The Security dialog box sets the security level to Low, Medium, High or Very High to determine whether VBA macros are allowed to run automatically.



1 Security Level

2 Very High

3 High

4 Medium

5 Low

6 Trusted Publisher

4.12.125. Very High

1

Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.

4.12.125. High

2

Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.

4.12.125. Medium

3

BricsCAD displays a dialog box when a VBA macro runs.

The dialog box allows you to choose whether or not to run the macro.

4.12.125. Low

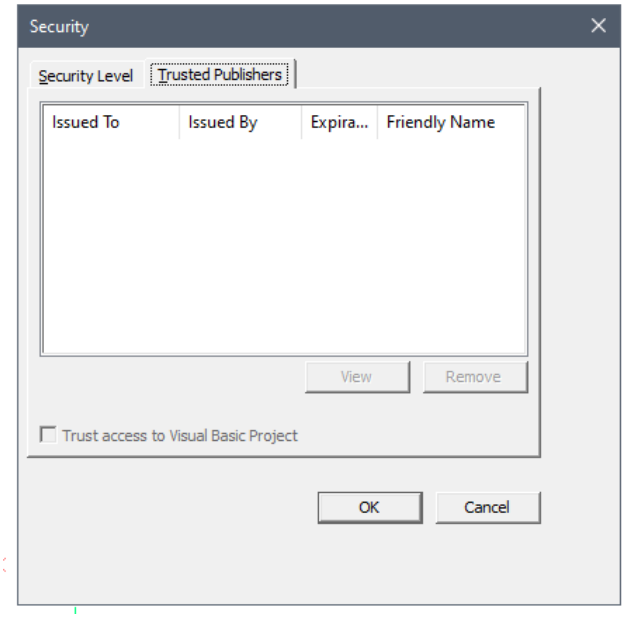
4

All VBA macros are run.

4.12.125. Trusted Sources Option

5

Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High.



Remove

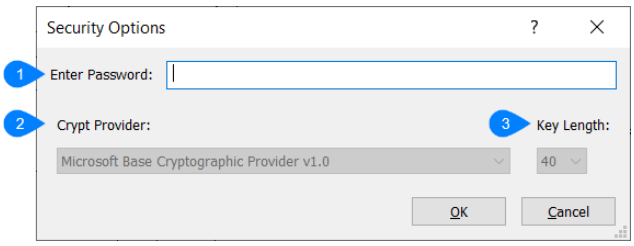
Removes the selected VBA macro from the list.

Trust access to Visual Basic Project

Allows access to the VBA Project.

4.12.126 Security options dialog box

The Security Options dialog box allows you to set a password to protect your drawing from unauthorized access.



1 Enter Password

2 Crypt Provider

3 Key Length

4.12.126. Enter Password

1

Specify a password by entering one or more characters.

4.12.126. Crypt Provider

2

Specifies the encryption level for drawing properties. Choose an encryption provider supplied by the operating system.

4.12.126. Key Length

3

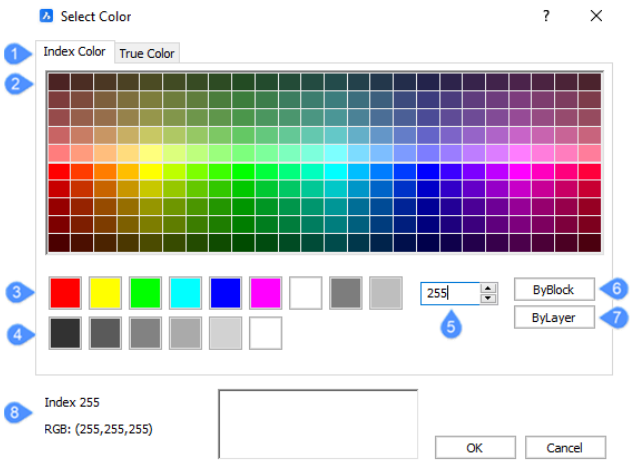
Select a length for the encryption key. The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.

4.12.127 Select color dialog box

The Select Color dialog box allows you to select a color for layers, grids, dimensions, background etc. It is composed from two tabs: **Index Color** tab and **True Color** tab.

4.12.127. Index Color tab

1



1 Index Color tab

2 Colors

3 Basic colors

4 Shades of gray

5 Color index

6 ByBlock

7 ByLayer

8 Color sample

Colors
Displays the other 240 colors.

Basic colors
Displays the basic colors.

Shades of gray
Displays the shades of grays.

Color index
Displays the index of the selected color. You can insert an index and see the preview of the color at field number 8.

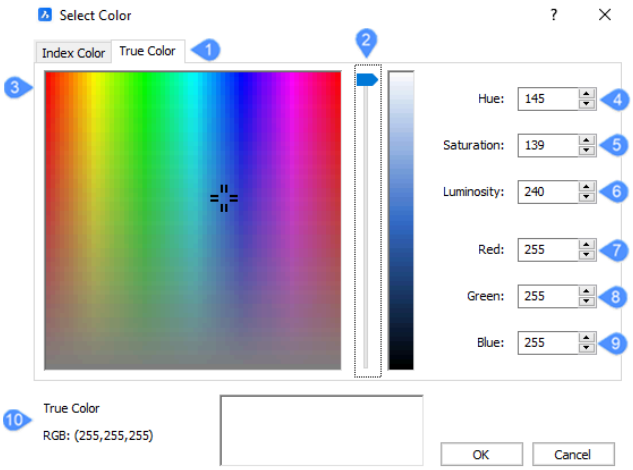
ByBlock
Sets the color to ByBlock in which entities take on the color defined by their block.

ByLayer
Sets the color to ByLayer in which entities take on the color defined by their layer's property.

Color sample
Displays a preview for the selected color and the RGB color value.

4.12.127. True Color Tab

2



1 True Color Tab

2 Luminosity

3 Color

4 Hue

5 Saturation

6 Luminosity

7 Red

8 Green

9 Blue

10 Color sample

Luminosity
Chooses the luminosity of the color.

Color
Chooses hue (left and right) and saturation (up and down).

Hue
Specifies the color's hue. It is ranging from red through green, blue, and pink.
It ranges from 0 to 240.

Saturation
Specifies the intensity of the color, ranges 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, ranges 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, ranges from 0 (no red) to 255 (full red).

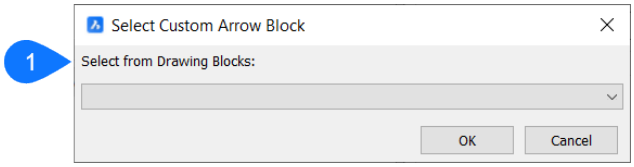
Green
Specifies the amount of green in the color, ranges from 0 (no green) to 255 (full green).

Blue
Specifies the amount of blue in the color, ranges from 0 (no blue) to 255 (full blue).

Color sample
Displays a preview for the selected color and the RGB color value.

4.12.128 Select Custom Arrow Block dialog box

The Select Custom Arrow Block dialog box allows you to save the selected elements as a detail.



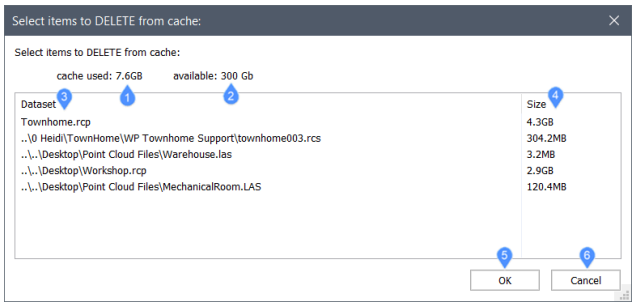
1 Select from Drawing Blocks

4.12.128. Select from Drawing Blocks

1
Specifies one of the blocks in the drawing to use as arrowhead.

4.12.129 Select items to delete from cache dialog box

TheSelect Items To Delete from Cache dialog box enables you to delete point cloud files that are stored in the cache.



1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

4.12.129. Cache used

1
Specifies the total amount of used cache.

4.12.129. Available
2

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

4.12.129. Dataset
3

Displays a list of cached point cloud files including their names and locations. You can select point cloud files from the list to delete them from the cache.

4.12.129. Size
4

Displays the amount of cache used for each point cloud file.

4.12.129. OK
5

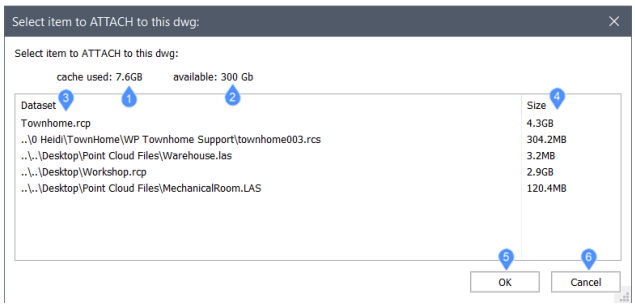
Displays the Confirm deleting selected cache items dialog box where you can review the list of point clouds to be deleted.

4.12.129. Cancel
6

Closes the dialog box without deleting point cloud files from the cache.

4.12.130 Select item to attach to this DWG

TheSelect item to attach to this DWG dialog box enables you to attach point cloud files that have already been processed and stored in the cache.



1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

4.12.130. Cache used

1

Specifies the total amount of used cache.

4.12.130. Available

2

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

4.12.130. Dataset

3

Displays a list of cached point cloud files including their names and locations. You can select a cached file from the list to attach to the current drawing.

4.12.130. Size

4

Displays the amount of cache used for each point cloud file.

4.12.130. OK

5

Displays the Attach point cloud dialog box where you can specify how to attach the selected point cloud file to the current drawing.

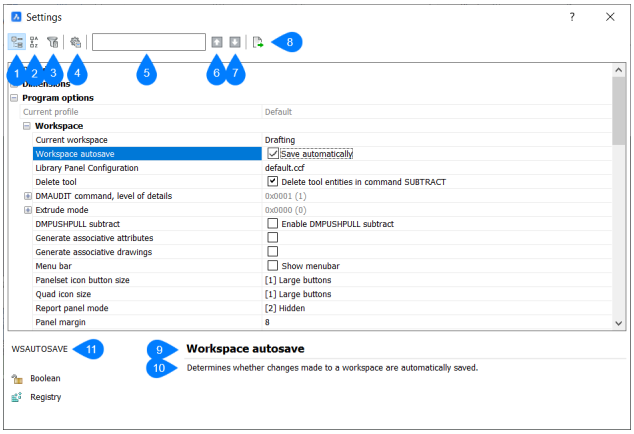
4.12.130. Cancel

6

Closes the dialog box without attaching a point cloud file.

4.12.131 Settings dialog box

The Settings dialog box allows you to change the values of system variables and user preferences.



1 Categorized

2 Alphabetic

3 Show differences

4 Dialog configurations

5 Search bar

6 Next

7 Previous

8 Export

9 Name

10 Description

11 Properties

4.12.131. Categorized

1

Lists variables in related groups.

4.12.131. Alphabetic

2

Lists variables in related groups.

4.12.131. Show differences

3

Lists variables whose values are changed from the initial (default) values.

4.12.131. Dialog configurations

4

Searches for variables, configures how the search is carried out, and specifies how differences are handled. The Configure Settings dialog box opens when clicking the icon.

4.12.131. Search bar

5

Lists variables trough a real-time search field when entering the first few letters of their names or descriptions.

4.12.131. Next

6

Jumps to the next candidate that matches the search phrase.

4.12.131. Previous

7

Jumps to the previous candidate that matches the search phrase.

4.12.131. Export

8

Opens the Export Settings dialog box to allow you to save all settings and values to a CSV file.

4.12.131. Name

9

Displays the name of the selected system variable.

4.12.131. Description

10

Displays a brief description of the selected system variable.

4.12.131. Properties

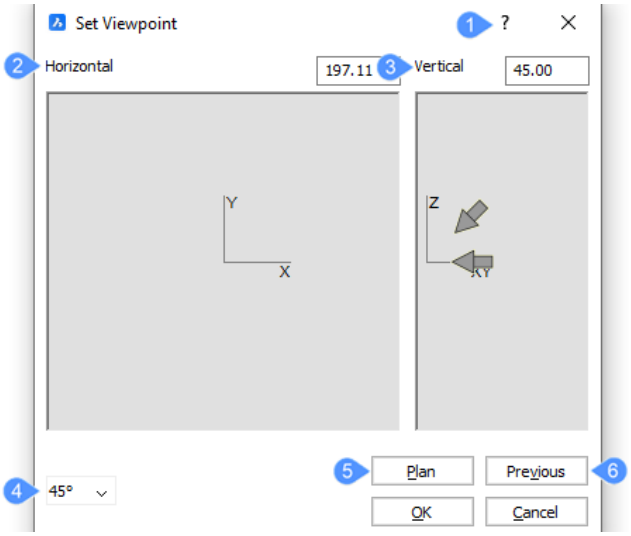
11

Lists several basic properties/characteristics of the selected system variable.

4.12.132 Set viewpoint dialog box

The Set Viewpoint dialog box allows you to set the current 3D viewport or plan view.

Nota: Turn off the perspective mode (view PERSPECTIVE command), because this command is not available in the Perspective mode.



1 Command reference

2 Horizontal

3 Vertical

4 Angle Preset

5 Plan

6 Previous

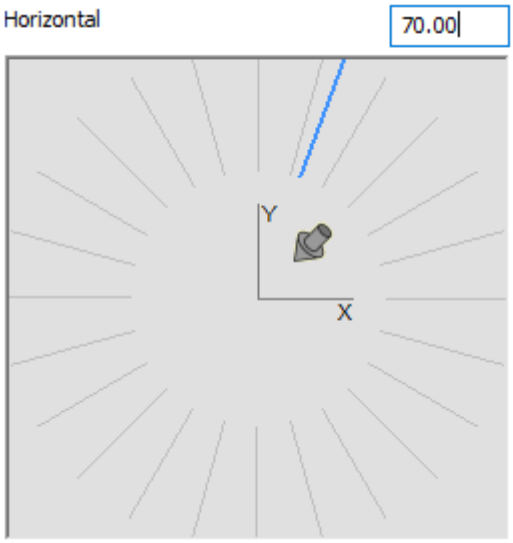
4.12.132. Command reference

1

Opens the Bricsys Help article about the DDVPOINT command.

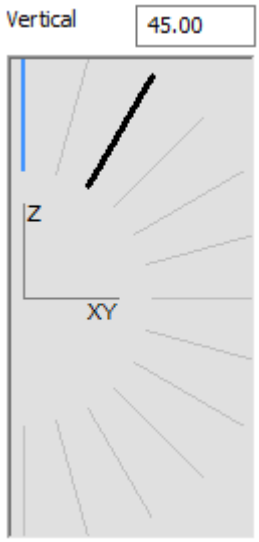
4.12.132. Horizontal
2

Specifies the horizontal angle from the X - axis. Click a preset angle or type a value in the **Horizontal** field. 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.

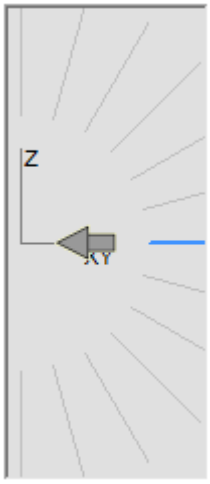


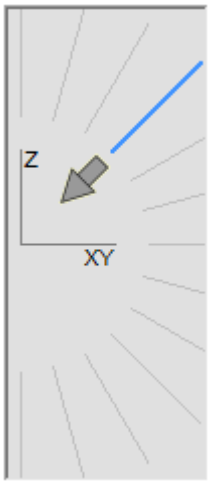
4.12.132. Vertical
3

Specifies the vertical angle from the XY plane. Click a preset angle, or else type a value in the Vertical field.

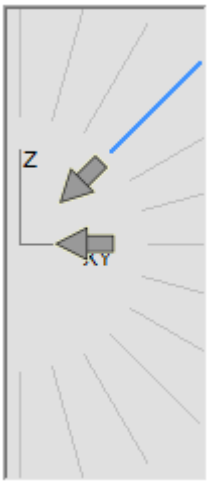


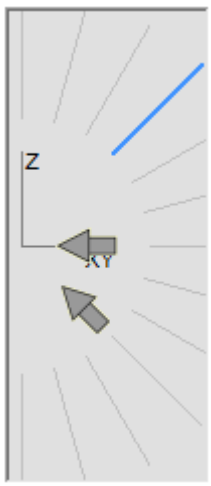
Click in the inner half circle. (1) View direction is horizontal. (2) View direction is 45° down (not available if PresetAngle = 45°).

(1) 

(2) 

View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. (3) First select the horizontal arrow, then (4) click the 45° down arrow.

(3) 

(4) 

4.12.132. Angle Preset

4

Sets markers at 5, 15, or 45 degrees; default = 45.

4.12.132. Plan

5

Reverts the 3D viewpoint to 2D plan view.

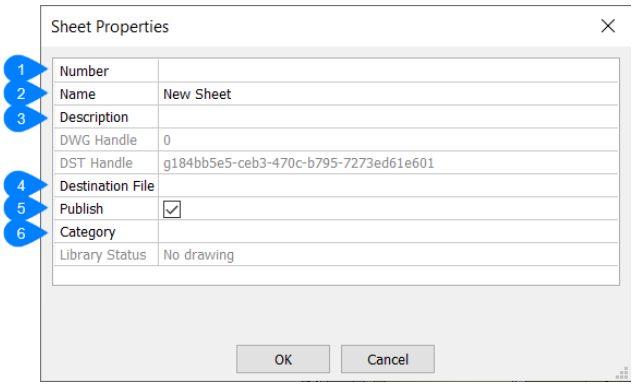
4.12.132. Previous

6

Reverts to the previous viewpoint.

4.12.133 Sheet Properties dialog box

The Sheet Properties dialog box allows you to view and edit properties of a sheet.



1 Number

2 Name

3 Description

4 Destination File

5 Publish

6 Category

4.12.133. Number

1

Specifies the number of the selected sheet.

4.12.133. Name

2

Specifies the name of the selected sheet.

4.12.133. Description

3

Specifies a short description of the selected sheet.

4.12.133. Destination File

4

Specifies the file in which the sheet will be saved.

4.12.133. Publish

5

Toggles the option to publish the currently selected sheet.

4.12.133. Category

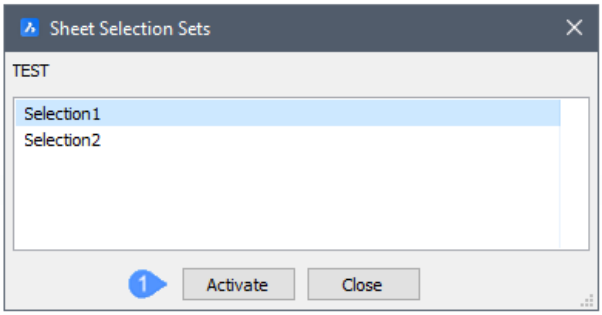
6

Specifies the category of the selected sheet.

4.12.134 Sheet selection sets dialog box

The Sheet Selection Sets dialog box allows you to manage the sheet selections.

To create a sheet selection, right click when multiple sheets are selected to open the **Sheet Selection** context menu and **Save Sheet** selection with a name.



1 Activate

4.12.134. Activate

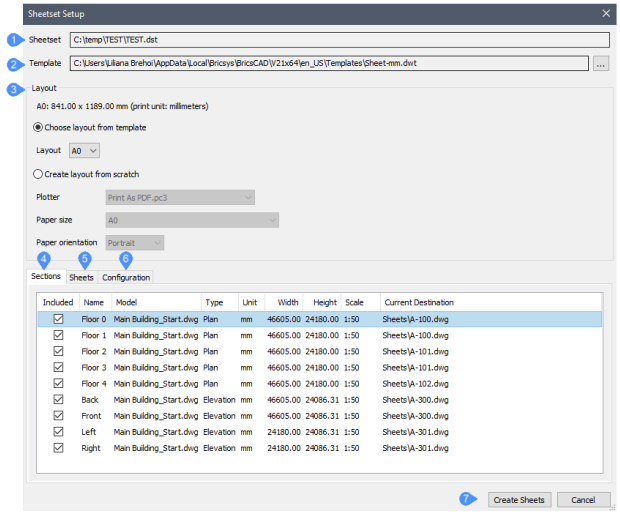
1

Restores the selected selection sheet set.

Right click a selection sheet set to display a context menu.

4.12.135 Sheetset setup dialog box

The **Sheetset Setup** dialog box provides the settings that are used for plotting and publishing.



1 Sheetset

2 Template

3 Layout

4 Sections Tab

5 Sheets Tab

6 Configuration Tab

7 Create Sheets

4.12.135. Sheetset

1

Shows the path of the sheetset data file (.dst) that stores the associations and information that define a sheet set.

4.12.135. Template

2

Shows the path of the current template that contains the page setups for all new sheets.

Click **Browse** () to change the template.

4.12.135. Layout

3

Selects which layout (different paper sizes) the section results will be generated.

- **Choose layout from template**: Allows to select a predefined layout from a drop-down list.
- **Create layout from scratch**: Allows to set:
  - **Plotter**: Choose from the drop-down list with the available plotters.
  - **Paper size**: Choose the paper size from the drop-down list.
  - **Paper orientation**: Choose the paper orientation from the drop-down list that opened.

4.12.135. Sections Tab

4

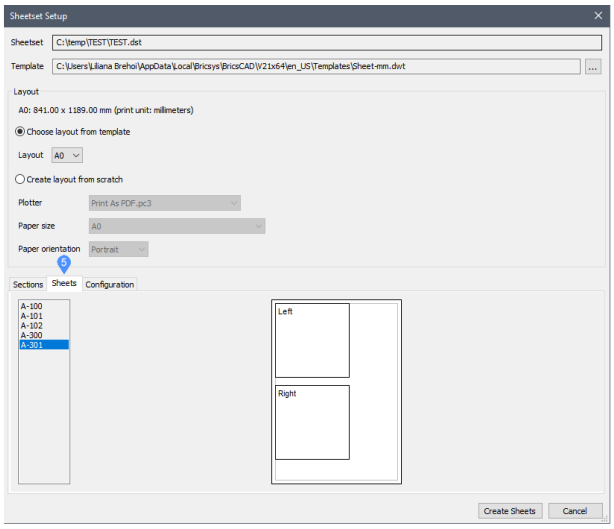
Lists all sections. For each section with an empty **Current Destination** field, a viewport and a sheet view is generated.

- **Included**: Specifies which sections you want to be generated.
- **Name**: Displays the name of the sections.
- **Model**: Displays the name of the model from the sections area.
- **Type**: Displays the section type.
- **Unit**: Displays the drawings unit.
- **Width**: Displays the maximum model width dimension.
- **Height**: Displays the maximum model height dimension.
- **Scale**: Displays the scale for the section sheet. It may be changed pressing right click and selecting a new scale form the drop-down list.
- **Current Destination**: Displays the destination .dwg file of the current section.

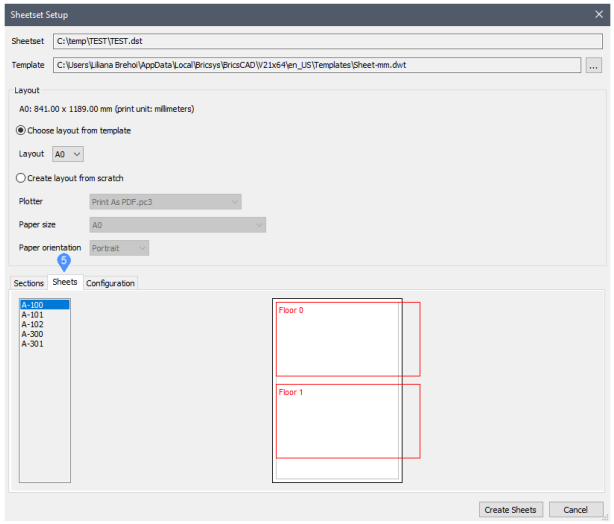
4.12.135. Sheets Tab

5

Shows how the viewports will be displayed in the layout.

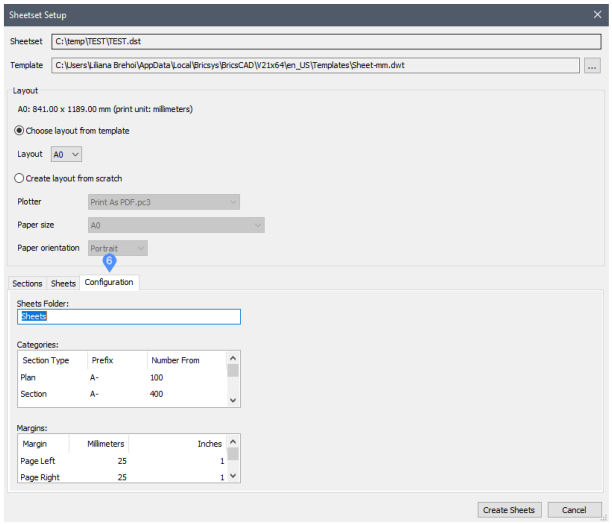


Nota: When the viewport is larger than the layout, it displays in red.



4.12.135. Configuration Tab

6



Slowly double-click (or click and hit Space/Enter) to change the properties:

- Under **Sheets Folder** you may change the name of the folder where the sheets will be stored.
- Under **Categories** you can change the **Prefix** and **Number From** for each Section Type.
- Under **Margins** you can set the margins of the page.

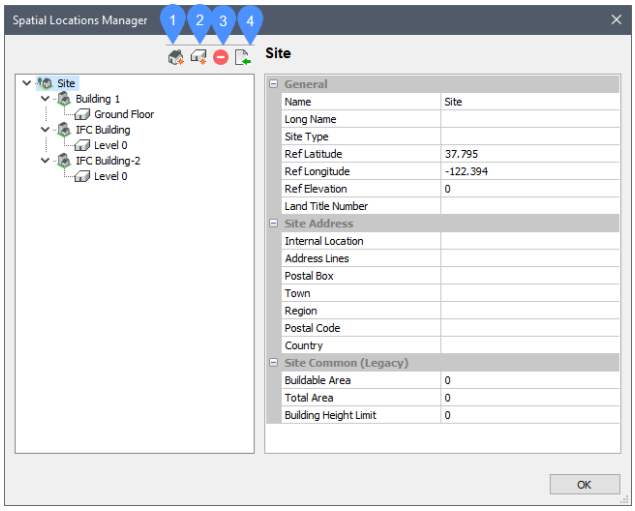
4.12.135. Create Sheets

7

The sheets are generated.

4.12.136 Spatial Locations Manager dialog box

The Spatial Locations Manager dialog box allows you to create and edit sites, buildings and stories.



1 New Building

2 New Story

3 Delete

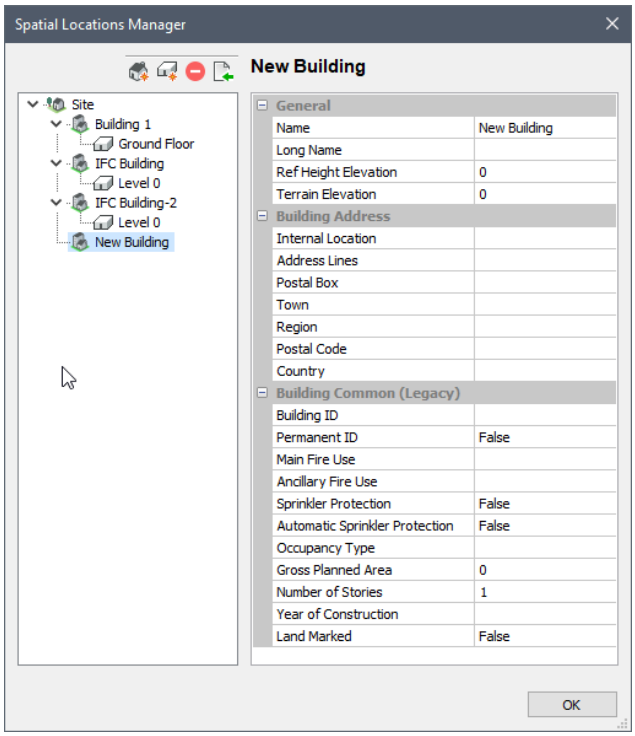
4 Import

4.12.136. New Building

1

Adds a new building to the model.

Fill out the properties grid:

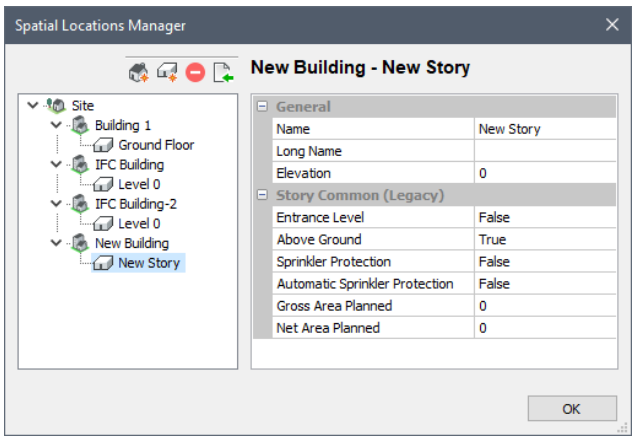


4.12.136. New Story

2

Adds a new story in the selected building.

The properties grid is completed.



4.12.136. Delete

3

Deletes the selected building and story.

4.12.136. Import

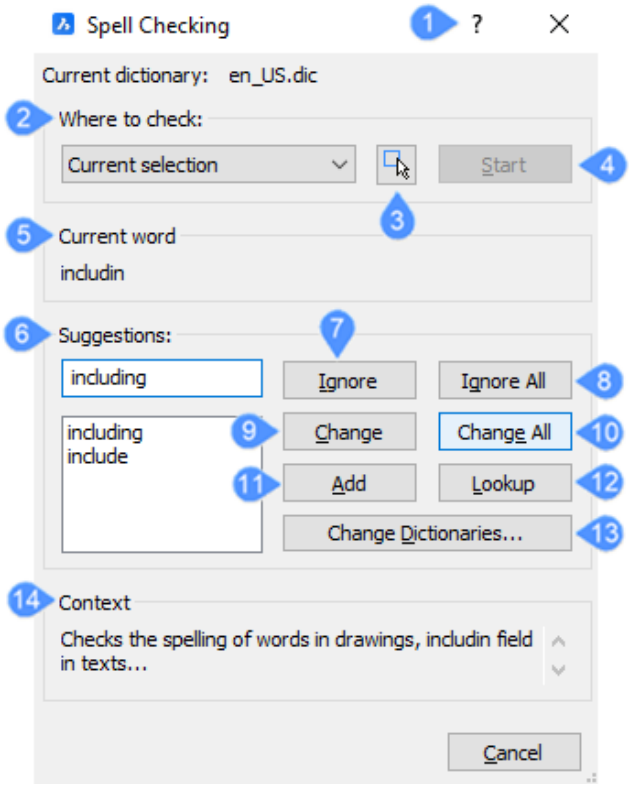
4

Imports spatial locations from a text file.

Displays a Select File to Insert dialog box from which will open the *.txt spatial location file.

4.12.137 Spell checking

The **Spell Checking** dialog box allows you to check the spelling of words in drawings, including fields in texts, multiline texts, leader, multileader, table and block attributes.



1 Command reference

2 Where to check

3 Select entities

4 Start

5 Current word

6 Suggestions

7 Ignore

8 Ignore All

9 Change

10 Change All

11 Add

12 Lookup

13 Change Dictionaries…

14 Context

4.12.137. Command reference

1

Opens the Bricsys Help article about the SPELL command.

4.12.137. Where to check

2

Checks text in:

- **Entire drawing**: Checks all text in the drawing.
- **Selected entities**: Checks text in selected entities. Click the **Select Entities** button to choose text.

4.12.137. Select entities

3

Select the text to be checked. Choose one or more pieces of text and press Enter to return to the dialog box.

4.12.137. Start

4

Starts checking the spelling of words.

4.12.137. Current word

5

Displays the misspelled word.

4.12.137. Suggestions

6

Lists the suggested replacements for misspelled words. Choose a replacement word and then click **Change** or **Change All**.

4.12.137. Ignore

7

Ignores the spelling of the word.

4.12.137. Ignore All

8

Ignores all instances of the word.

4.12.137. Change
9

Changes the word to the one selected from the **Suggestion** list.

4.12.137. Change All
10

Changes all instances of the word to the one selected from the **Suggestion** list.

4.12.137. Add
11

Adds the word to the word list.

4.12.137. Lookup
12

Provides additional alternative spellings for the word selected in the **Suggestions** list.

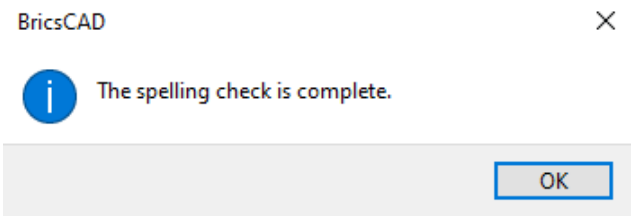
4.12.137. Change Dictionaries...
13

Changes the dictionary used for spell checking. Displays the **Change Dictionaries** dialog box.

4.12.137. Context
14

Displays the context in which the misspelled word appears.

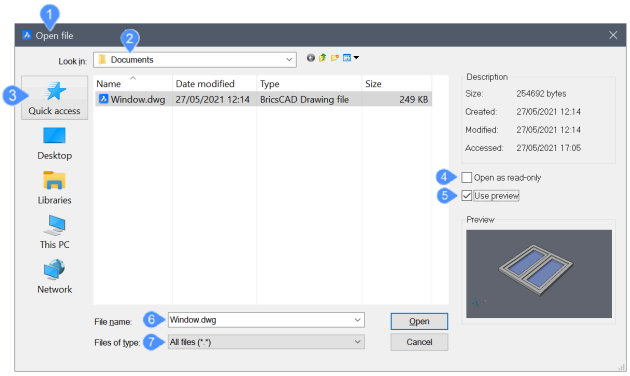
When the spelling check is completed, the following message box appears:



4.12.138 Standard open file dialog box

The Standard open file dialog box allows you to open or import a file.

This dialog box can be accessed using multiple commands.



1 Dialog name

2 Look in

3 Quick access

4 Open as read-only

5 Use preview

6 File name

7 Files of type

4.12.138. Dialog name

1

Displays the dialog name depending on the command used.

4.12.138. Look in

2

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

4.12.138. Quick access

3

The Quick access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate the Quick access with folders that you want to be always available, not only with folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

The Quick access folders are available during both open and save operations.

4.12.138. Open as read-only

4

Tick the checkbox to open the file as read-only. Consequently, you do not have permission to write new information or edit the data in the file.

4.12.138. Use preview

5

Tick the checkbox to view a part of the file before opening.

4.12.138. File name

6

Specifies the name of the file.

4.12.138. Files of type

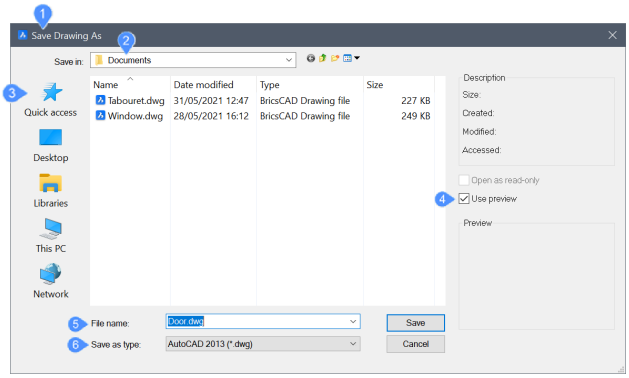
7

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

4.12.139 Standard save file dialog box

The Standard Save File dialog box allows you to save or export a file.

This dialog box can be accessed using multiple commands.



1 Dialog name

2 Save in

3 Quick access

4 Use preview

5 File name

6 Files of type

4.12.139. Dialog name

1

Displays the dialog name depending on the command used.

4.12.139. Save in

2

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

4.12.139. Quick access

3

The Quick Access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate Quick access with folders that you want to be always available, not only for folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

Quick Access folders are available during both open and save operations.

4.12.139. Use preview

4

Tick the checkbox to see a part of the file before saving.

Nota: A preview is available when selecting an existing file to replace it.

4.12.139. File name

5

Specifies the name of the file. You can accept the predefined name or you can specify a different name.

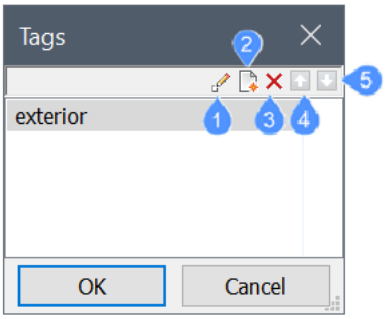
4.12.139. Files of type

6

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

4.12.140 Tags dialog box

The Tags dialog box allows you to add or edit compositions and physical materials tags.



1 Edit

2 New

3 Delete

4 Move up

5 Move down

4.12.140. Edit

1

Edits the name of the selected tag.

4.12.140. New

2

Adds a new tag at the bottom of the list.

4.12.140. Delete

3

Deletes the selected tag.

4.12.140. Move up

4

Moves up the selected tag.

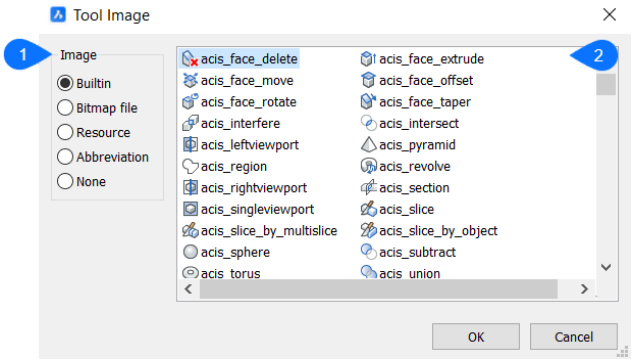
4.12.140. Move down

5

Moves down the selected tag.

4.12.141 Tool image dialog box

The Tool Image dialog box allows you to specify the image of tools.



1 Image

2 List of tool images

4.12.141. Image

1

Specifies the way to set the tool images.

Builtin

Specifies the tool image from one of the builtin tool images from the list.

Bitmap File

Specifies the tool image from a bitmap file.

Resource

Specifies the resource of the tool image.

Abbreviation

Specifies an abbreviation for the tool instead of an image.

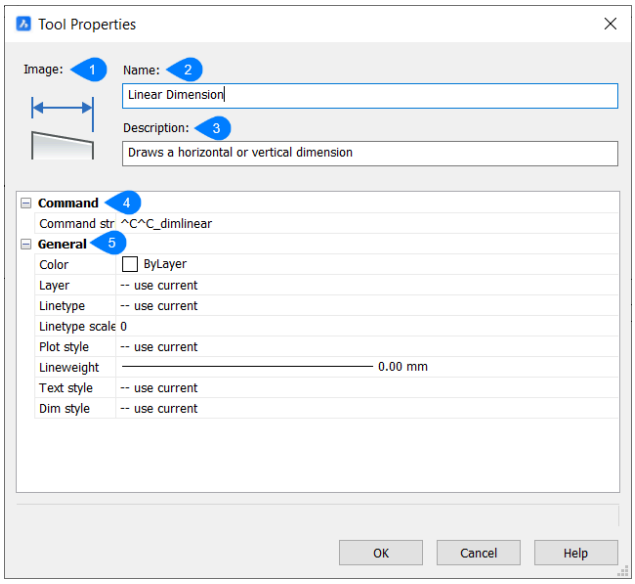
None
No image or abbreviation is used for the tool.

4.12.141. List of tools

2

Displays a list of built in tool images when the **Builtin** option is selected.

4.12.142 Tool Properties dialog box



1 Image

2 Name

3 Description

4 Command String

5 General

4.12.142. Image

1

Displays an image of the selected tool.

4.12.142. Name

2

Specifies the name of the selected tool.

4.12.142. Description

3

Specifies the description of the selected tool.

4.12.142. Command String

4

Specifies the macro to run when you click the icon; you can use any macro characters used for menus and toolbar buttons.

4.12.142. General

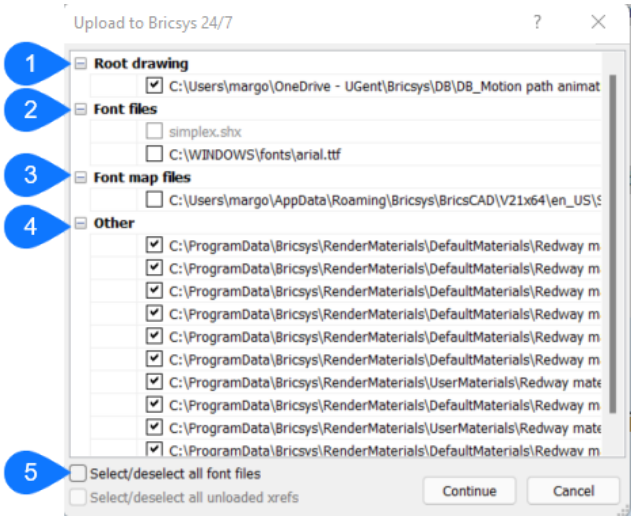
5

Specifies the properties of the entity; the properties vary, depending on the tool.

4.12.143 Upload to Bricsys 24/7 dialog box

The Upload to Bricsys 24/7 dialog box allows you to upload packages of drawing files and it dependencies to Bricsys 24/7

Nota: The project needs to be saved before uploading it to Bricsys 24/7.



1 Root drawing

2 Font files

3 Font map files

4 Other

5 Select/deselect all font files

4.12.143. Root drawing

1

Displays a list of the current drawing files.

4.12.143. Font files

2

Displays a list of the currently used font files.

4.12.143. Font map files

3

Displays a list of the currently used font map files.

4.12.143. Other

4

Displays a list of other used files such as render material images.

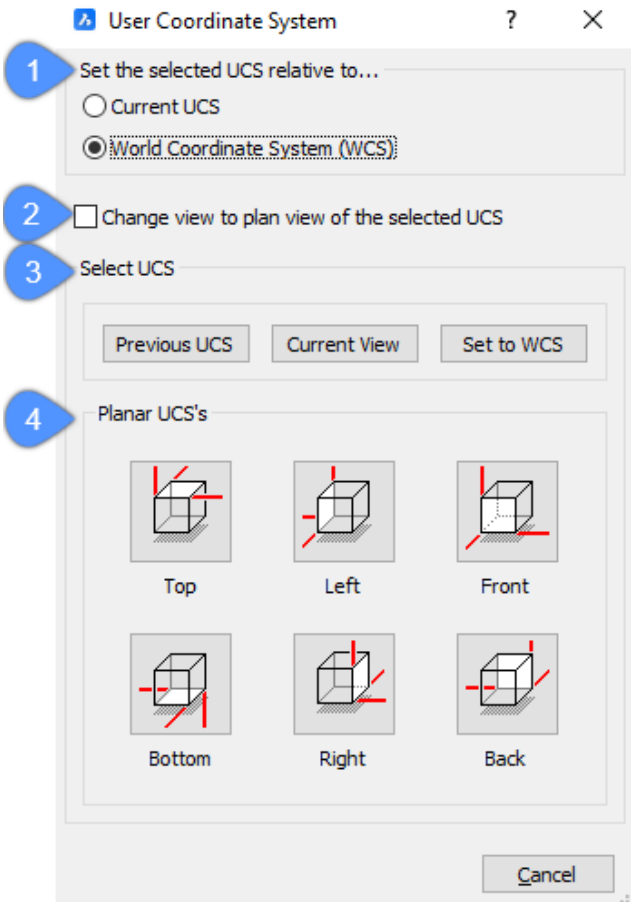
4.12.143. Select/deselect all font files

5

When checked, all font files and font map files are checked if this was not the case before. When unchecked, all font files are unchecked if there were any checked before. Pressing the **Continue** button will ask you to sign in Bricsys 24/7 to complete the uploading process.

4.12.144 User coordinate system dialog box

The **User Coordinate System** dialog box allows you to set the UCS to a specified viewpoint.



1 Set the selected UCS relative to...

2 Change view to plan view of the selected UCS

3 Select UCS

4.12.144. Set the selected UCS relative to...

1

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. World Coordinate System is the default coordinate system when no UCS is set and it cannot be changed.

4.12.144. Change view to plan view of the selected UCS

2

Toggles whether the plan view is displayed when switching UCSES:

- **Yes**: plan view of the new UCS is displayed.
- **No**: the viewpoint does not change.

4.12.144. Select UCS

3

Sets the UCS to one of the following coordinate system. 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 UCS**: Changes the UCS to match the WCS.

4.12.144. Planar UCS's

4

Sets the UCS to one of the following standard orthographic views. After you click one of the options, the program sets the UCS and then exits the dialog box.

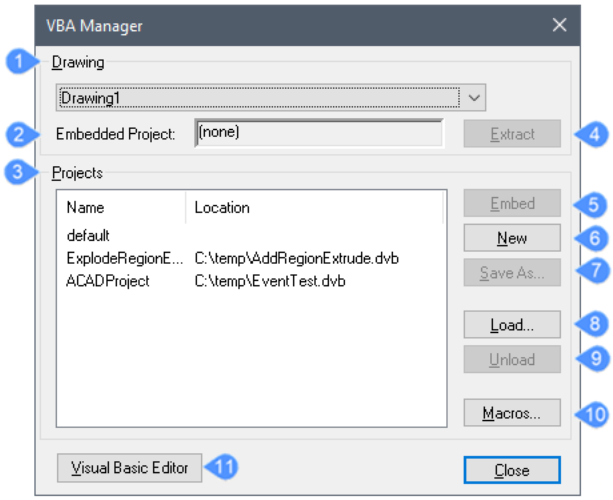
- **Top**: Changes the UCS to match the top view.
- **Left**: Changes the UCS to match the left view.
- **Front**: Changes the UCS to match the front view.
- **Bottom**: Changes the UCS to match the bottom view.
- **Right**: Changes the UCS to match the right view.
- **Back**: Changes the UCS to match the back view.

4.12.145 User profile manger dialog box

The User Profile Manager dialog box allows you to set the current user profile, to create, copy, delete, import, and export profiles.

4.12.146 VBA manager dialog box

The VBA Manager dialog box allows you to manage VBA projects.



1 Drawing

2 Embedded Project

3 Projects

4 Extract

5 Embed

6 New

7 Save As

8 Load

9 Unload

10 Macros

11 Visual Basic Editor

4.12.146. Drawing

1

Displays the names of drawings currently open in BricsCAD:

Choose a drawing name from the drop-down list; the embedded project is listed in the Embedded Project text box, if any.

4.12.146. Embedded Project

2

Displays the name of the embedded project, if any.

4.12.146. Projects

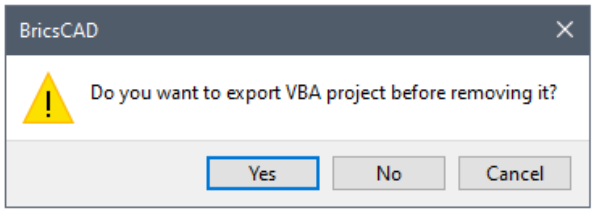
3

Name

Displays the names of projects loaded into BricsCAD at this time.

Location
Displays the drive and folder names of DVB and VBI files.

4.12.146. Extract
4
Removes the project from the current drawing; displays dialog box:



Yes: displays the Save As dialog box:

Name the project and 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.

4.12.146. Embed
5
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.

4.12.146. New
6
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.

4.12.146. Save As
7
Saves the selected project by another name. Displays the *Save As* dialog box.

4.12.146. Load
8
Loads DVB and VBI project files; displays the *Open* dialog box.

4.12.146. Unload
9
Unloads the selected project without a warning.

4.12.146. Macros

10

Displays the *Run BricsCAD VBA Macro* dialog box.

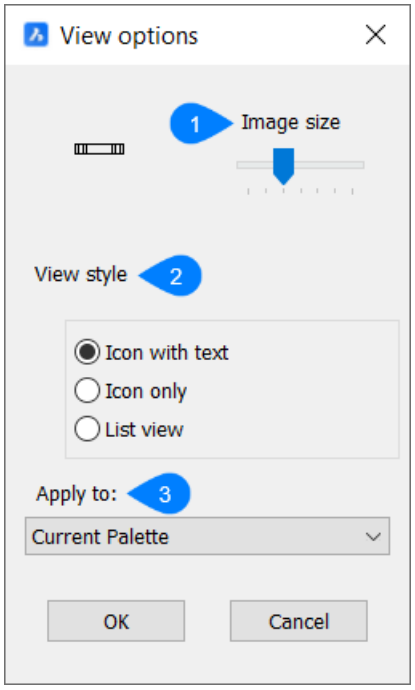
4.12.146. Visual Basic Editor

11

Opens the VBA integrated development environment.

4.12.147 View Options dialog box

The View Options Dialog Box allows you to customize how the tools in the tool palettes panel are shown.



1 Image size

2 View style

3 Apply to

4.12.147. Image size

1

Sets the size of the displayed icons by sliding the indication arrow.

4.12.147. View style

2

Specifies how the tools are shown in the palette.

Icon with text

Displays the tools in a grid view with their icon and name.

Icon only

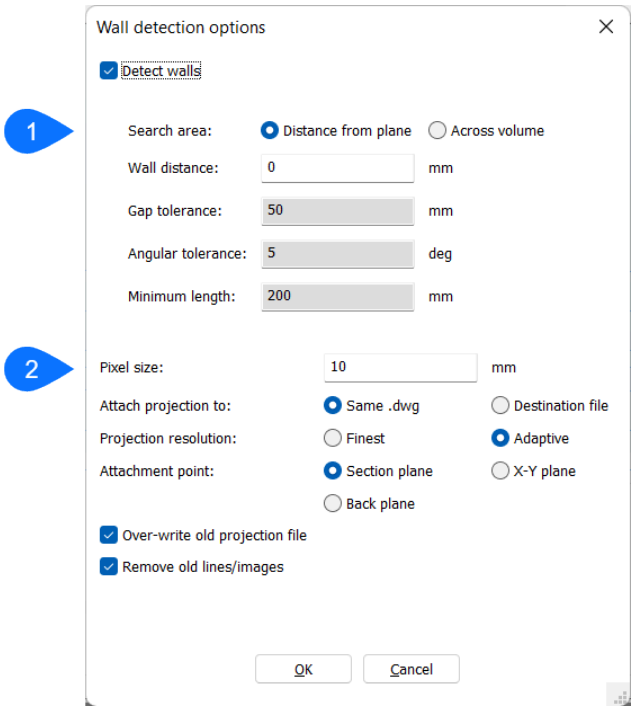
Displays the tools in a grid view with their icon only.

List view
Displays the tools in a list view with their icon and name next to it.

4.12.147. Apply to
3
The changes made can either be applied to the current palette only, or to all palettes.

4.12.148 Visual settings dialog box

4.12.149 Wall detection options dialog box



1 Parameters that control 2D line drawing

2 Parameters that control the generated raster image

4.12.149. Parameters that control 2D line drawing
1
Detect walls
When checked, lines representing walls will be generated with the raster image. When unchecked, only a raster image is created.

Search area

- **Distance from plane**: detects walls in a given radius from a plane. This option is often used to generate vertical sections.
- **Across volume**: detects all walls in the point clouds. This option is often used to generate floor plans.

Wall distance
Controls the search area for wall detection.

Nota: This option is only available when 'Distance from plane' is selected.

Gap tolerance
Controls the search area for the algorithm to draw lines. Gaps smaller than the specified value will be closed.

Angular tolerance
Controls the angular tolerance of the generated lines. A high tolerance will result in pronounced right angles. This is useful for perpendicular plans.
Generated lines with a smaller angle than the specified value will be merged.

Minimum length
Controls the minimum length of a generated line. Lines with a smaller length than the specified value will not be generated.

4.12.149. Parameters that control the generated raster image 2

Pixel size
Controls the pixel size of the generated image.

Nota: the pixel size is in model units. Higher pixel size results in a lower resolution, resulting in a faster process.

Attach projection to
Determines where the resulting image is generated.

Same.dwg
The projection is made in the same file

New destination
Creates a new .dwg file to make the projection.

Nota: You can overwrite a previous file or create a new one.

Projection resolution

Finest
Produces a sharp and detailed background picture.

Adaptive
Produces a less detailed and softer background image.

Attach point
Determines the insertion point for the generated image.

- In-place: places the lines and image in the same location and orientation as the sectionplane.
- X-Y plane: aligns the section with the X_Y axis.

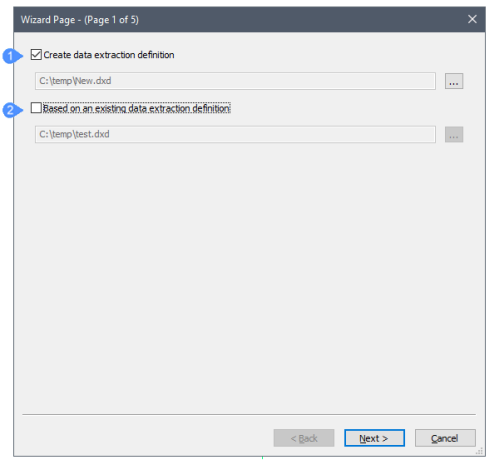
  Nota: The section can be edited in Top view as a regular 2D drawing. This is practical for vertical sections.

Over-write old projection file
When checked, the existing projection file will be overwritten.

4.12.150 Wizard page dialog box

The Wizard Page dialog box guides you to export entity properties, block attributes, and drawing information to a table or to an external data file.

4.12.150. Wizard Page 1 Dialog Box 1



1 Create data extraction definition

2 Based on an existing data extraction definition

Create data extraction definition

Creates a new .dxd (Data eXtraction Definition) file:

1 Tick the checkbox.

2 Click the Browse button.

3 Select a folder in the Save Data Extraction file dialog box.

4 Enter a name in the File name field.

5 Click the Save button.

Based on an existing data extraction definition

Uses an existing .dxd file as the template for the new one to perform the same type of extraction in a different drawing or to edit the data extraction definition:

1 Tick the checkbox.

2 Click the Browse button.

3 In the Open Data Extraction Definition file dialog box, select a .dxd file.

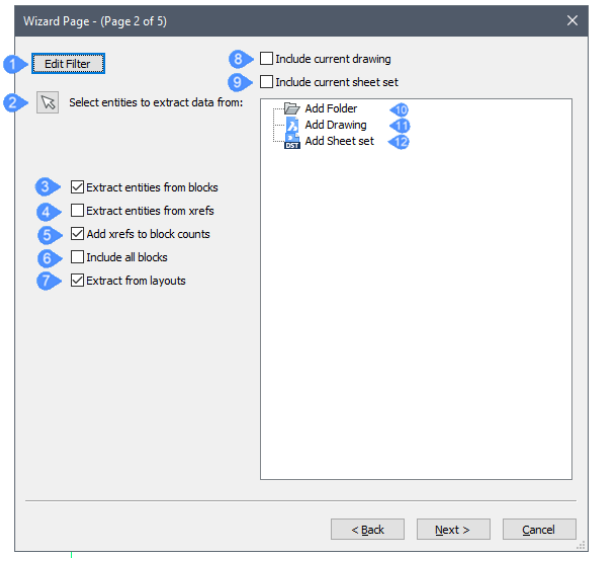
4 Click the Open button.

Newly created custom *.dxd files can be used to create schedules in the Project Browser.

.dxd files can be edited in a text editor such as MS Notepad.

Click **Next** for the next page.

4.12.150. Wizard Page 2 Dialog Box 2



1 Edit filter

2 Select entities to extract data from

3 Extract entities from blocks

4 Extract entities from Xrefs

5 Add Xrefs to block counts

6 Include all blocks

7 Extract from layouts

8 Include current drawing

9 Include current sheet set

10 Add Folder

11 Add Drawing

12 Add Sheet Set

Guides to select entities for Data Extraction and apply filters for them before it is extracted in order to have less information to sort later.

The DXEVAL system variable controls when the update notification occurs in the drawings where changes affect the extracted data.

Edit filter
Creates or edits a Filter.

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.

Extract entities from blocks
Includes entities nested in blocks.

Extract entities from Xrefs
Includes entities in externally referenced drawings (Xref files).

Add Xrefs to block counts
Counts selected Xrefs as blocks.

Include all blocks
Includes all block entities.

Extract from layouts
Includes entities from paper space of the drawing.

Include current drawing
Adds the current drawing from which to extract data.

Include current sheet set
Adds the current sheet set from which to extract data.

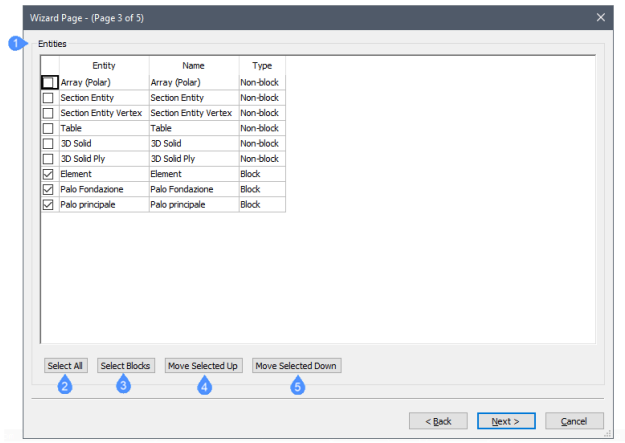
Add Folder
Adds a folder of drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **Folder**: click the **Browse** button, then select a folder in *Choose a Folder* dialog box.
- **Options**
  - **Include subfolders**: tick the checkbox to include subfolders.
  - **Use a wildcard to filter drawings to be included**: tick the checkbox, then type a text string in the filter field. For example, House*.dwg includes all files starting with 'House'.

Add Drawing
Adds a drawing from which to extract data with a double click.

Add Sheet Set
Adds a sheet set from which to extract data with a double click.
Click **Next** for the next page.

4.12.150. Wizard Page 3 Dialog Box 3



1 Entities

2 Select All

3 Select Blocks

4 Move Selected Up

5 Move Selected Down

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 clicking 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.

Entities

- **Entity**: lists the entity types and blocks in the selection, according to the options set in the *Data Extraction* dialog box.
- **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.

Move Selected Up

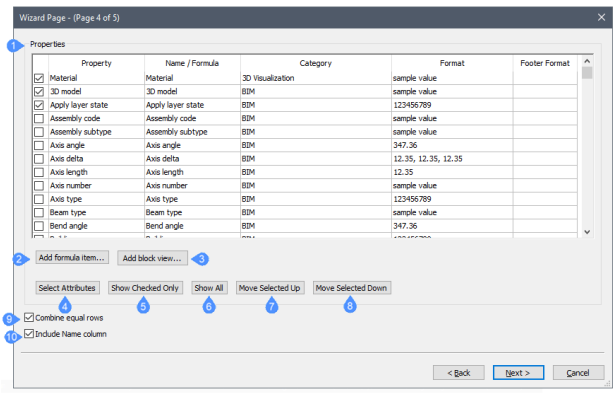
Move the selected item(s) up the list.

Move Selected Down

Move the selected item(s) down the list.

Click **Next** for the next page.

4.12.150. Wizard Page 4 Dialog Box 4



1 Properties

2 Add formula item

3 Add block view

4 Select Attributes

5 Show checked only

6 Show All

7 Move Selected Up

8 Move Selected Down

9 Combine equal rows

10 Include Name column

Select the properties to be included in the data extraction file: tick the checkbox to check/uncheck a property. Use the same techniques to select more than one item as described in Wizard Page 3 Dialog Box.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

Properties

- **Property**: lists all properties of the selected entities.
- **Name /Formula**: lists the name of the selected entities or the formula. Accept the default names or type a customized name. The property names display in the column headings in the data extraction export.
- **Category**: reports the category of the selected entities.
- **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, just like the Format section of the Field dialog.
- **Footer Format**: selects from the drop-down list opened with right mouse button the Footer settings:

Add formula item

Adds a formula item.

Add block view
Adds BlockViewProperty as Block view category.

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.

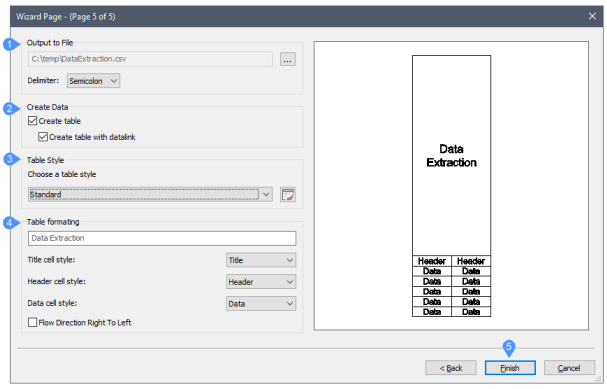
Move Selected Down
Move the selected item(s) down the list.

Combine equal rows
Combine rows that contain the same entities.

Include Name column
Toggles the display of the Name column in the exported data.

Click **Next** for the next page.

4.12.150. Wizard Page 5 Dialog Box 5



1 Output to File

2 Create Data

3 Table Style

4 Table formatting

5 Finish

Output to File
Exports the data in a delimiter separated format (.csv):
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.

Delimiter: specifies the separator between fields of data:

- Semicolon
- Comma

  Nota: Multiply composition names contain a comma, therefore do not select comma as the delimiter character in case multiply compositions are processed.
- Space
- Tab

Create Data
Exports the data in a Table:

- Create table: creates a static table.
- Create table with datalink: creates a dynamic table. Executing the DataLinkUpdate command synchronizes the data in the table with the model. Save the model first. You will be prompted to save the *.dxd definition file with the table.

Table Style
Select a table style from the drop-down list.
Click the **Table Style** icon; edit an existing table style or create a new one.

Table formatting
Type a title in the Title field.

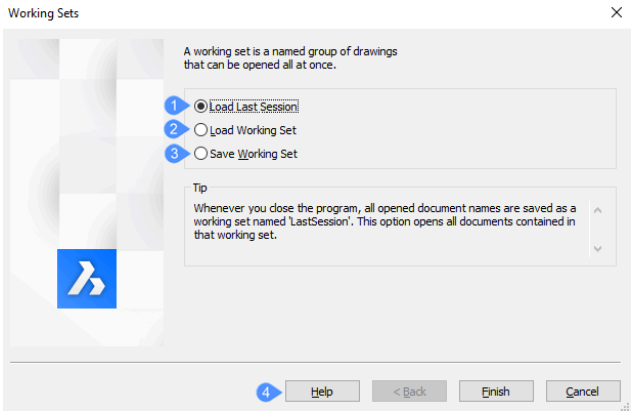
- Title cell style: sets the title cell style.
- Header cell style: sets the Header cell style.
- Data cell style: sets the Data cell style.
- Flow Direction Right to Left: indicates the flow direction from right to left.

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).

4.12.151 Working sets dialog box

The **Working Sets** dialog box allows you to create and load sets of drawing files, making it convenient to load and entire group of drawings at once.



1 Load Last Session

2 Load Working Set

3 Save Working Set

4 Help

4.12.151. Load Last Session

1

Opens all drawings that where open when the last BricsCAD session was closed.

After you click the **Finish** button, BricsCAD opens all the drawings that were open the last time you closed the program.

4.12.151. Load Working Set

2

Click the **Next** button and choose the name of a workset from the **Select a Workset** list. Click the **Finish** button. The program opens all the drawings that are part of the workset.

4.12.151. Save Working Set

3

Saves drawings currently open by name as a working set. Click the **Next** button.

When you click on the **Save** button, the **Save workset** dialog box opens. Click the **Finish** button to close the dialog box.

Nota: Working sets are saved in the registry and so they are defined differently for each user profile. See the PROFILEMANAGER command to learn more about user profiles.

Nota: Delete removes a named working set. Redefine (overwrites) an existing named working set.

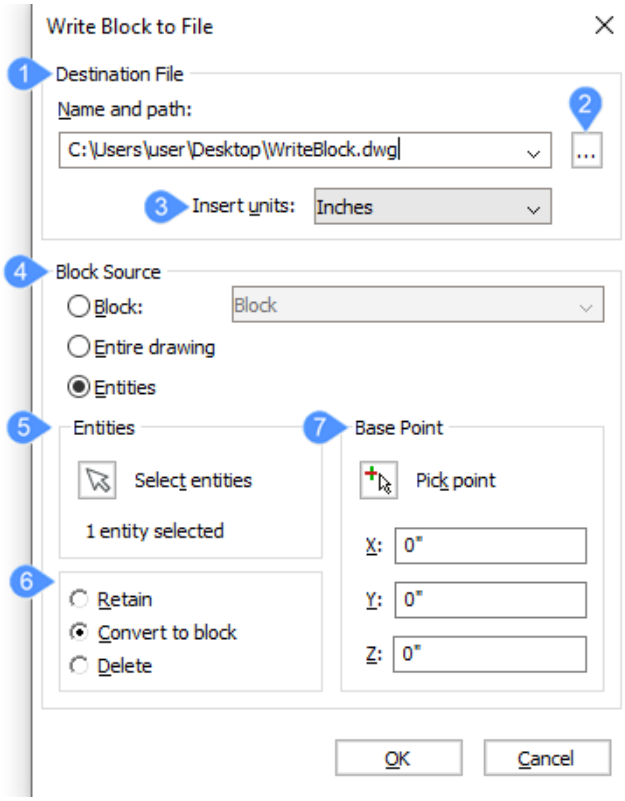
4.12.151. Help

4

Opens the Bricsys Help article about WORKSETS command.

4.12.152 Write block to file dialog box

The **Write Block to File** dialog box allows you to write blocks and other drawing parts to a separate drawing file.



1 Destination file

2 Save Block

3 Insert units

4 Block Source

5 Entities

6 Options

7 Base Point

4.12.152. Destination file

1

Specifies what the file should be named and where it should be saved.

- **Name and path**: Specify the default path and file name (new block.dwg).

4.12.152. Save Block

2

Changes the path and file name. Opens the **Save Block** dialog box.

4.12.152. Insert units
3

Specifies the units to be used by the exported entities when they are later inserted into other drawings.

You can choose the measurement unit from the drop-down list.

Units are saved in the INSUNITS variable.

4.12.152. Block Source
4

Selects the part of the drawing to save to a new block:

- **Block**: Saves a block from the drawing. This option is unavailable when no blocks exist in the drawing. When this option is selected, the other options become unavailable. You can choose from the drop-down list the name of a block you want to save.
- **Entire drawing**: Saves the entire drawing to the file. When this option is selected, all other options become unavailable.
- **Entities**: Saves entities that you select to the file.

4.12.152. Entities
5

When the **Block Source** is set to **Entities**, you are prompted at the Command line to choose one or more entities using any selection method.

4.12.152. Options
6

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 selected in the drawing; this is like moving the selection to a new drawing.

  Nota: If the **Delete** option was set, the OOPS command brings back the erased block entities.

4.12.152. Base Point
7

Specifies the origin point of the saved entities in the new file. You can pick a point in the drawing or specify the X, Y, Z coordinates.

5. Gerenciar desenhos

5.1 Creating a drawing

A new drawing is always created as a copy of a template drawing. The default template is saved through the BASEFILE system variable.

In BricsCAD you can create new drawings:

- In the **Start** tab.
- Using the QNEW command.
- Using the NEW command.
- Using the NEWWIZ command.
- Double-clicking a *.dwt file in a Windows Explorer window.

5.1.1 Specifying the template folder

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select **Template Path** (TEMPLATEPATH system variable).

3 Click the **Browse** button.

4 On the **Browse for folder** dialog box, select a folder, then click the **OK** button.

5.1.2 Setting the default template drawing

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select the BASEFILE system variable.

3 Click the **Browse** button.

4 On the **Choose a File** dialog box, select a drawing file (*.dwg) or drawing template (*.dwt), then click the **Open** button.

5.1.3 Using the Start tab

1 Click the **Start** tab.

Launch the GOTOSTART command to open the **Start** tab if necessary.

The name of the current drawing template is displayed beneath the **New Drawing** button.

2 Do the following:

- (option) Click the template list* button, to choose a different template drawing.
- Click the **New Drawing** button.

* All template drawings in the **Template** folder matching the current workspace (Drafting, Modeling, Mechanical or BIM), are listed here. Choose **More...** to see the full content of the **Template** folder.

5.1.4 Creating a drawing using the default template drawing

First method:

Click the + button at the right hand side of the document tabs.

Second method:

Launch the QNEW command.

Third method:

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog, choose **Start from Default template**, then click the **Next** button.

Nota: If the default template drawing is not specified, the **Create New Drawing** wizard is launched depending on the current value of the STARTUP system variable, otherwise the **Select Template** dialog box appears.

5.1.5 Creating a drawing using the New command

1 Launch the NEW command.

2 On the **Create New Drawing** dialog box, choose **Start from Scratch**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings.

Click the **Finish** button to create the drawing.

Nota: 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.

BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

5.1.6 Setting the Plot style policy

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Plot and publish**, select **Plot style policy** (PSTYLEPOLICY system variable).

3 Do one of the following:

- Choose **Associate an entity's plot style with its color** for Color dependent (CTB) plot styles.
- Choose **No association between color and plot style** for Named (STB) plot styles.

5.1.7 Creating a drawing using a template drawing

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box choose **Start from template**, then click the **Next** button.

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.

- Click the **Browse...** button to select a template drawing.

  You can choose between ***.dwg** and ***.dwt** file types.

5.1.8 Creating a drawing using a wizard

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box, choose **Use a Wizard**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings, then click the **Next** button.

4 Select the **Unit of measurement** and the display **Precision**, then click the **Next** button.

5 Select the **Angle of measurement** and the display **Precision**, then click the **Next** button.

6 Choose between **Color Dependent** plot styles (CTB) or **Named** plot styles (STB), then click the **Next** button.

7 Select the default **Color** and **Linetype**.

8 Define the **Grid**, **Snap**, **UCS icon** and **Marker Blips** settings.

9 Click the **Finish** button to create the drawing.

(option) Choose **Save As..** in the **File** menu to save the drawing as a new template in the **Templates** folder.

5.2 Working in the Drawing Explorer dialog box

5.2.1 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** dialog box appears.

2 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.

Nota:

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder are greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

5.2.2 Opening a drawing

To open a drawing in one of your favorite folders:

1 In the **Drawings** > **Folders** sub-window, open the folder of the drawing.

2 Select a drawing.

3 A preview is shown in the **Preview** sub-window.

4 Do one of the following:
   - Double-click the drawing.
   - Right-click and choose **Open** in the context menu.

5.2.3 Logging on to Bricsys 24/7

1 In the **Drawings** pane, select **Folders**.

2 Click **Logon to Bricsys 24/7**.

3 Fill out your username and password.

4 When logged in, all of your projects on Bricsys 24/7 are listed.

5.2.4 Opening a drawing from a Bricsys 24/7 project

1 Expand the project, then select the drawing folder.

2 Select the drawing.

3 A preview is shown in the **Preview** sub-window.

4 Right-click the drawing, then choose **Download** in the context menu.

A **Save File As** dialog box appears.

5 Select a folder and optionally rename the drawing, then click the **Save** button.

The drawing is downloaded and opened in the current BricsCAD session.

5.2.5 Copying definitions between drawings

1 Open both the source and the target drawing.

2 Launch the **Drawing Explorer** dialog box.

3 In the source drawing, select the category you want to copy items from, e.g. **Blocks**.

4 (option) Adjust the view option of the **Details** pane, e.g. **Icon View** button ( ).

5 Select the definition(s) you want to copy, e.g. a series of blocks.

6 Press and hold the Ctrl key to select multiple items.

7 Do one of the following:
   - Press and hold the left mouse button to drag the selection set to the **Open Drawings** section of the Drawing Explorer.

     Release the left mouse button when on the target drawing name.

     The selected items are copied to the target drawing.
   - Right-click and choose **Copy** in the context menu.

     Select the same definition category in the target drawing, then right-click and choose **Paste** in the context menu.

Nota:

- You will be prompted to overwrite items of the same name.
- Drag-and-drop is not supported for some definition categories, e.g. MultiLeader Styles, Visual styles, Render Presets and Page Setups.

5.3 Drawing properties

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored within a drawing.

The **Drawing Properties** dialog box includes:

- General information: drawing data
- Summary: editable default properties
- Statistics: history data
- Sheetset: info about the associated sheetset
- User defined custom properties

Drawing properties can be referenced in fields.

5.3.1 Opening the Drawing Properties dialog

Do one of the following:

- Choose **Drawing Properties...** in the **File** menu.
- Execute the DWGPROPS command.

5.3.2 Defining default properties

1 On the **Drawing Properties** dialog box, click the **Summary** tab page.

2 Fill out the property fields.

3 Click the **OK** button.

5.3.3 Creating custom properties

1 On the **Drawing Properties** dialog box, click the **Custom** tab page.

2 Click the **Add** button.

The **Custom Property** dialog appears.

3 Type a name in the **Name** field.

The custom property name is listed under **Document** in the **Field** dialog.

4 Type a value in the **Value** field.

The value is displayed where the custom property is inserted as a field.

5.4 Drawing security options

The SECURITYOPTIONS command allows you to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

5.4.1 Setting the password

1 Do one of the following:

- Choose **Security Options...** in the **File** | **Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

The **Security options** dialog box opens.

2 Type the password in the **Enter Password** field.

3 Choose a crypt provider from the **Crypt Provider** list.

4 Set the **Key Length** to **40**, **48** or **56**.

5 Click the **OK** button or press Enter.

The **Security options** dialog box 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.

5.4.2 Removing the password

1 Do one of the following:

- Choose **Security Options...** in the **File** | **Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

2 The **Security options** dialog box opens.

3 Empty the **Enter Password** field.

4 Click the **OK** button or press Enter.

A message box appears to confirm that the password is removed.

5 Click the **OK** button or press Enter to close the message box.

5.5 Opening and saving drawings

The way BricsCAD opens and saves drawings is controlled through a series of system variables.

Name	Title	Description
	Display full path in title	Controls whether the full path of a drawing displays in the title bar, or the file name only.
	Drawings path	Specifies an additional folder in the file dialog for theOPENandSAVEAScommands. On the Windows platform: the fifth folder in the**Open File** dialog's **Places** bar.
-	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).
	Thumbnail preview image size	Sets the size of the thumbnail image in the Preview on file dialogs.

Name	Title	Description
	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 handling Xrefs. Therefore create indexes only for drawings that will be used as an external reference. 0 = No indexes.
	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. 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, entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded.
	Multi-Threading flags	Bitcode 1024: Delayed XREF loading. Delays the loading of Xrefs to idle time.
	Incremental save backup	Controls the creation of a backup files (BAK). Improves the speed of incremental saves if switched off, especially for large drawings.

Name	Title	Description
	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 at the end of the drawing data base. A full save takes longer to be executed than a partial save.
	Recent File List	Specifies the number of files shown in the**Recent Files**flyout of the**File**menu and on the **Start** tab.
	Proxy graphics	Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.
	Proxy notice	Displays a notice when you open a drawing containing proxy entities (1) created by an application that is not present.
	Proxy show	Controls how proxy entities (1) display in a drawing.
	Proxy web search	Determines whether BricsCAD checks for object enablers.
	Save format	Sets the default dwg version to save a drawing.
	Save file path	Specifies the path and folder where automatic saves (.SV$) and temporary files are stored. TheSAVEFILEFOLDERcommand opens the SAVEFILEPATH location in the system file browser.
	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a.SV$extension in the folder defined by the SAVEFILEPATH system variable.
	Enable macros	Defines whether macros are enabled when loading a VBA-project.

Name	Title	Description
	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no auto-save file will be created for files that are viewed only.
	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.

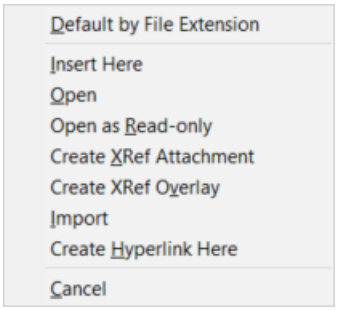
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(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.

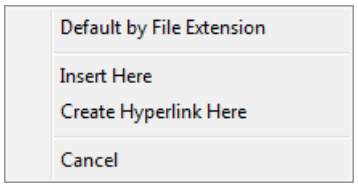
5.5.1 Using drag-and-drop to open a drawing

You can open a drawing by dragging it from a File dialog box onto the BricsCAD application window. Select the file, then hold down the left mouse button and move the cursor to the BricsCAD application window and release the left mouse button to load the drawing. When you drag a file holding down the right mouse button, a context menu displays:

When a drawing file is selected:



When another file type is selected:



Choose**Default by File Extension**to:

- Open a drawing file.
- 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 as a block in the current drawing.
- Attach a PDF.
- Attach a raster image file.
- Insert the file as an OLE object.

Choose**Open**to open a drawing file .

Choose **Open as Read-only** to open a drawing file and preserve the original from being overwritten. You can only save such drawing under a different name.

Choose**Create Xref**or **Create Xref Overlay** to attach a drawing file as an external reference to the current drawing.

- **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.

Choose **Import** to import the geometry in the current drawing.

Choose**Create Hyperlink Here**to create an hyperlink to the selected file.

Nota:

- Drawing file extensions are: .DWG, .DXF or .DWT
- When multiple files are selected, the files are processed one after the other.

5.6 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.

5.6.1 General drawing information

The DWGPROPS command opens the **Drawing Properties** 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.

5.6.2 Information about entities

The STATUS command reports the total number of entities in the drawing.

Using **Quick Select**, 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.

5.7 Comparing drawings

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 you to merge the compared drawings visually.

5.7.1 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 entities 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.

5.7.2 Executing a comparison session

1 Launch the DWGCOMPARE command.

1 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 a **File** dialog box.

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 are displayed using the **Compare Display Colors**.

The **Compare** toolbar appears.

5 To check the **Diff Type** of an entity, do one of the following:
- If **Rollover Tips** (RT) are on, hover over the entity.
- Select the entity and check the **Diff type** field in the **Properties** panel.

6 Click the **Endcompare** tool button on the **Compare** toolbar to conclude the comparison session.

5.7.3 Analyzing the comparison in the Structure panel

1 Load the **Default.cst** configuration in the **Structure** panel.

2 Expand the **Comparison** node.

Four subnodes are shown: **Missing**, **Modified (Original)**, **New** and **Modified (Compared)**.

The number of entities in each subnode is indicated between brackets.

3 Do the following:
- Click a subnode to select the corresponding entities in the drawing.
- 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.
- Right-click, then choose **Zoom** in the context menu to zoom onto the selected entity or entities.

Nota: When selecting entities in the **Structure** panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. (See the **Structure** panel).

5.7.4 Merging the compared drawings

1 **Missing entities** only exist in the current drawing.
- 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.
- 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.

3 **New entities** exist in the compared drawing only.
- 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.

When the merge is complete, the **Comparison** node in the **Structure** panel disappears. A message appears in the Command line: No more differences to show. Session is closed.

5.8 Repairing a drawing

TheAUDIT command analyzes the integrity of the current drawing and optionally tries to fix errors.

TheRECOVER command repairs a damaged drawing, then opens the drawing.

5.8.1 Analyzing the integrity of the current drawing

1 Launch the AUDIT command.

Prompts you: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 #.

Nota:

- Set theAUDITCTLsystem variable to 1, to letAUDIT create an ASCII file describing the detected problems and the action taken. This report, with the .adt file extension, is placed in the same directory as the current drawing .
- Use theRECOVER command to repair the errors thatAUDIT cannot fix.

5.8.2 Repairing a drawing

1 Do one of the following:

- Choose**Recover**in the**File** > **Drawing Utilities**menu.
- Launch the RECOVER command.

The**Open Drawing**dialog window displays.

2 Select the drawing, then press the**Open**button on the**Open Drawing**dialog window.

The drawing is processed and - if the repair succeeds - opened.

3 (option) Press F2 to open the**Prompt History**window in order to read the recover report.

5.9 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.

5.9.1 Accessing the Project settings

1 Launch the SETTINGS command.

1 The **Settings** dialog box opens.

2 Go to **Program Options** > **Files** > **Projects**.

5.9.2 Creating projects

1 Edit the PROJECTSEARCHPATHS system variable.

2 Click the **Browse** button at the right hand side of the **Project search paths** settings field.

The **Project Settings** dialog box appears.

3 To create the first project:

- Click the **Add Project or Path** button on the **Project Settings** dialog.

  A new project is added.

- Type a name to replace the **NewProject1** default name.

4 To create additional projects:

- Click **Projects** (on top of the projects tree) or select an existing project.
- Click the **Add Project or Path** button.

  A new project is added.

- Type a name to replace the **NewProject1** default name.

5.9.3 Adding search paths to a project

1 Edit the PROJECTSEARCHPATHS system variable.

2 Select the project in the **Projects** tree.

3 Click the **Add Project or Path** button.

A path is added to the project.

4 Type a name for the new path.

5 Click the **Browse** button.

The **Choose a Path** dialog box appears.

6 Browse to a folder, then click the **Select Folder** button.

7 Repeat the previous steps to add more search paths to a project.

5.9.4 Changing the order of the search paths

1 Edit the PROJECTSEARCHPATHS system variable.

2 Expand the project in the **Projects** tree.

3 Select a search path, then click the up or down buttons to change the order of the search paths.

5.9.5 Assigning a project to the current drawing

1 Edit the PROJECTNAME system variable.

2 Type a project name in the settings field.

6. Desenhar com precisão

6.1 Orthogonal mode

When creating or manipulating entities, Orthogonal mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the currentcoordinate system.

Lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. When drawing, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor.

Nota:

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the Orthogonal Mode axes using the SNAPANG system variable.
- Holding down the Shift key reverses the Orthogonal Mode setting.
- Orthogonal Mode is overruled by entity snaps.
- Turning on Ortho Mode automatically disables Polar Tracking and vice versa.

6.1.1 Toggling Orthogonal mode

Do one of the following:

- Click theORTHOfield in theStatus Bar.
- Press theF8function key.
- Hold down the Shift key.
- Launch the ORTHO command.

6.2 Unit settings

The UNITS command sets units of linear and angular measurements through the **Settings** dialog box. The -UNITS command sets units of linear and angular measurements in the Command line.

The INSUNITS (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

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.

Nota:

- 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].

6.2.1 System variables

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.
LUNITS	Linear Units Type	Sets the linear unit type: **Scientific**, **Decimal**, **Engineering**, **Architectural** or **Fractional**.
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 are displayed 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 are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.

Name	Title	Description
VOLUMEUNITS	Volume Units	Defines a list of units used for displaying volumes when the **Volume** bit of PROPUNITS is ON. If empty, all lengths are displayed 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 are displayed.

6.2.2 Entering angles

The AUNITS system variable defines the angular unit type.

The ANGDIR system variable sets the positive angle direction, counter clockwise (default) or clockwise. Angles are measured with respect to direction of the positive X-axis of the current coordinate system.

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).

6.3 Coordinate reference system

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 parameters, Molodenksy...)

Geographic datasets are defined in **geodatabase.xml**that is placed in *<User folder>\AppData\Roaming \Bricsys\BricsCAD\<RELEASE>\en_US\Support for Windows*. The content of this file is composed of the 5 sections:

1 CoordinateReferenceSystems

2 Datums

3 Ellipsoids

4 Transformations

5 ProjectionCode and Methods

6.3.1 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.

CRSnode 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" • "Northing" Directions MUST be defined.	String
xAxis, yAxis	Orientation of geographic direction. Possible values: • "Latitude" • "Longitude"	String
Greenwich	Greenwich Meridian relation.	Degree

Attribute name	Description	Units
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

Attribute name	Description	Units
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" Greenwich="0" proj="TMerc" Lon="12" Lat="0" SF="1"
 FE="4500000" FN="0" zone="4" units="Meter">
```

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

Attribute name	Description	Units
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 example, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc.	Integer

Example:there are unique CRS definitions according to Datums insideCRSnode:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN/2" id="DHDN/2.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN" id="DHDN.Gauss3d-4" pjcode="3" />
```

6.3.2 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 least 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 is 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 optional.	String

Example:

```
<Datums>
<Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz ">
<Datum id="DHDN/3">
<Transformation epsg="1777" alias="DHDN/3_to_WGS84" />
</Datum>
```

```
<Datum id="DHDN/2" name="Deprecated - Replaced by DHDN/3">
<Transformation epsg="1777" alias="DHDN/2_to_WGS84" />
</Datum>
<Datum id="DHDN" name="Deprecated - Replaced by DHDN/2">
<Transformation epsg="1673" alias="DHDN_to_WGS84" />
</Datum>
<Ellipsoid epsg="7004" alias="BESSEL" />
</Datum>
â€¦
</Datums>
```

6.3.3 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 optional.	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="6356583.7999999998" f="294.9786982139"
 e="0.0822718542" />
```

6.3.4 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 optional.	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 • 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

Attribute name	Description	Units
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" method="CFR" tx="-40.595" ty="-18.55" tz="-69.339"
 ds="-4.299" rx="-2.508" ry="-1.832" rz="2.611" accuracy="9" />
```

6.3.5 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.

6.4 Restrições 2D

Restrições paramétricas são usadas para manter relações e controlar dimensões da geometria 2D.

Existem dois tipos de restrições:

- **Geométricas**restrições controlam as relações entre as entidades.

- **Dimensionais**restrições são usadas para controlar as dimensões de uma entidade como comprimento, ângulo, raio ou diâmetro.

Uma restrição dimensional pode usar um valor ou uma fórmula, com outras restrições de dimensão ou parâmetros. Restrições dimensionais podem ser editadas no painel Parâmetros e Restrições e no painel Navegador de Mecânica.

Nota:

- Recomenda-se primeiro aplicar restrições geométricas, e então as restrições dimensionais.
- A variável de sistema PARAMETERCOPYMODE controla copia de restrições e parâmetros relacionados quando entidades são copiadas.
- O comando PARAMETRIZE2D cria automaticamente geométrico e dimensional parâmetros.

Quando se usam restrições um desenho pode estar em um de três estados:

- **Sub-restrito**: algumas entidades são restritas. Pode ser impossível aplicar algumas alterações usando comandos e procedimentos de modificação.
- **Totalmente restrito**: todos os geométricos e dimensionais possíveis e relevantes restrições são aplicadas à geometria do projeto. O desenho só pode ser modificado por alterando os valores das restrições dimensionais.
- **Superrestrito**: uma ou mais restrições violam ou contradiz outras restrições.

Restrições paramétricas permitem:

- Manter a intenção geométrica do projeto.
- Fazer múltiplas versões de um projeto aplicando diferentes valores às restrições dimensionais.

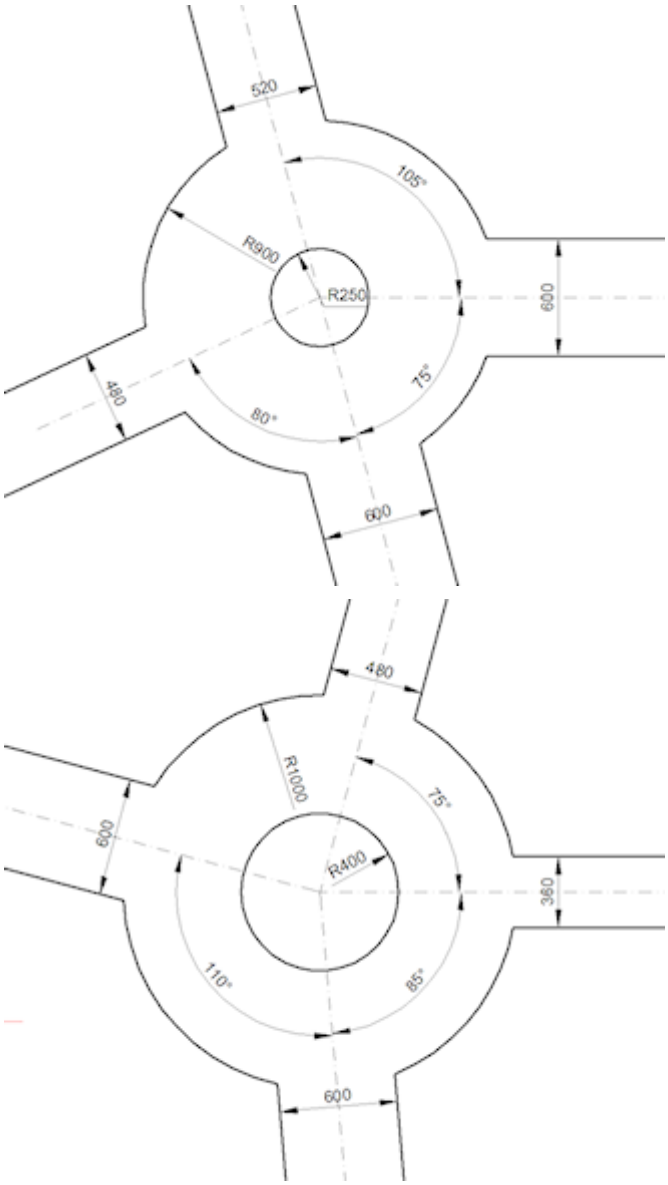


Figura 1: Duas versões do mesmo desenho totalmente parametrizado representando uma rótula

6.4.1 Barras de ferramentas e comandos

As ferramentas de restrições paramétricas 2D estão localizadas no **2D Barra de ferramentas Restrições** e a guia Faixa **paramétrica**.

Ferramentas para controlar a exibição de barras de restrição e restrições geométricas estão localizadas em duas barras de ferramentas flyout: **Restrição Dimensional** e **Geométrico Restrição**.

Comando	Descrição
	Permite definir restrições dimensionais para ser exibidas ou ocultas.

Comando	Descrição
	Aplica relacões geométricas entre entidades e/ou pontos de restrição válidos em entidades.
	Restringe pontos e entidades a uma posição fixa.
	Aplica uma restrição geométrica de coincidência entre dois pontos ou restringe um ponto a uma entidade.
	Restringe os pontos centrais de círculos, arcos, elipses ou arcos elípticos para coincidirem.
	Força linhas ou segmentos de polilinha para serem co-lineares.
	Força duas linhas ou segmentos de polilinha linear para que fiquem paralelas entre si.
	Restringe duas linhas ou segmentos de polilinha a encontrarem-se perpendicularmente um ao outro.
	Restringe duas entidades para manterem um ponto de tangência entre si, ou em suas extensões. As seguintes entidades são aceitas: linhas, segmentos de polilinha, círculos, arcos, elipses ou arcos elípticos.
	Restringe linhas ou pares de pontos a ficar em paralelo ao eixo-X do sistema de coordenadas atual.
	Restringe linhas ou pares de pontos a ficar em paralelo ao eixo-Y do sistema de coordenadas atual.
	Força uma spline a manter continuidade geométrica fluída com outra spline, linha, arco ou polilinha.

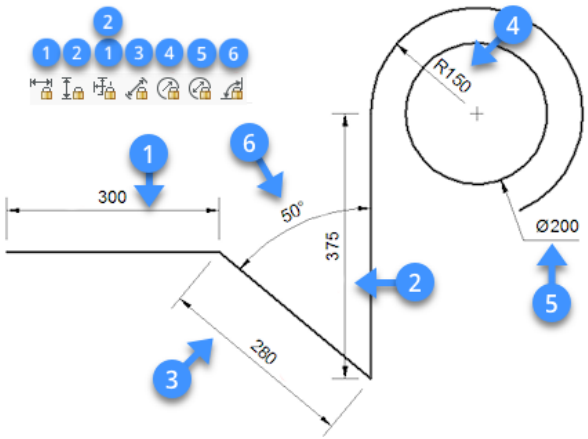
Comando	Descrição
	Restringe duas entidades ou pontos para que respeitem a simetria em relação a uma linha selecionada.
	Restringe arcos e círculos ao mesmo raio, ou linhas e segmentos de polilinha ao mesmo comprimento.
	Controla a exibição da barra de restrição das entidades. Barras de restrições estão sempre ocultas ao abrir um desenho.
	Aplica uma restrição dimensional a uma entidade ou entre pontos restritos em entidades. Converte as cotas associativas em dimensões dinâmicas.
	- restringe a distância horizontal (distância- X) entre dois pontos em relação ao sistema de coordenadas atual.
	- restringe a distância horizontal (distância- X) entre dois pontos em relação ao sistema de coordenadas atual.
	Restringe a distância horizontal (distância X) ou a distância vertical (distância Y) entre dois pontos em relação ao sistema de coordenadas atual.
	Restringe a distância entre dois pontos ou o comprimento de um segmento de linha ou polilinha.
	Restringe o raio de um círculo de um arco ou um segmento de arco de polilinha.
	Restringe o diâmetro de um círculo de um arco ou um segmento de arco de polilinha.

Comando	Descrição
	- restringe o ângulo entre duas linhas ou segmentos lineares de polilinha; o ângulo total de um arco ou um segmento de arco de polilinha; ou o ângulo entre três pontos em entidades.
	Converte uma dimensão associativa de uma restrição de dimensão.
	Remove todas as restrições geométricas e dimensionais de uma entidade.

6.5 Using dimensional constraints

Dimensional constraints control the dimensions of 2D entities.

- Horizontal distance (1) (DCLINEAR and DCHORIZONTAL commands).
- Vertical distance (2) (DCLINEAR and DCVERTICAL commands).
- Distance between points or the length of lines and polyline segments (3) (DCALIGNED command).
- Radius of arcs, circles and arc polyline segments (4) (DCRADIUS command).
- Diameter of arcs, circles and arc polyline segments (5) (DCDIAMETER command).
- Angle between entities or between points and entities (6) (DCANGULAR command).



Nota: Dimensional constraints are created using the current dimension style.

6.5.1 Editing dimensional constraints

Dimensional constraints can be edited:

- In the Properties panel.
- In theMechanical browser panel.
- In the Parameters and constraints panel.

- Using theDDEDITcommand.

Editing dimensional constraints in the Properties panel

1 Select the dimensional constraint.

The properties of the entity display in the Properties panel.

2 Type a new value orexpressionin the**Expression**field, then press Enter.

The readout of the**Value**field changes accordingly.

3 (option) Edit the**Name**field.

4 (option) Edit the**Description**field.

5 (option) Edit the**Text rotation**field.

6 Press the Esc key to unselect.

Nota: Select multiple dimensional constraints simultaneously to set their value equally.

Editing dimensional constraints in the Mechanical browser panel

1 If not already open, open theMechanical Browser panel.

2 Select a name of the dimensional constraint in the Parameters list.

The properties the selected parameter display in bottom part of theMechanical browser panel.

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 usingthe BMINSERT command.

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: linear, planar or volume.

Select the field, then click the down-arrow button and choose an option.

Using the DDEDIT command

1 Do one of the following:

- Double click the dimensional constraint.
- Launch the DDEDIT command.

You are prompted: _DDEDIT

The **Edit Text** dialog box displays.

2 Type a new value in the**Text**field, then press Enter or click the**OK** button.

3 (option) Click another dimensional constraint and repeat the previous step.

4 Do one of the following:
   - Keep editing more dimensional constraints.
   - Press Enter or right click to stop.

     All edited dimensional constraints are updated simultaneously.

6.5.2 Setting the visibility of dimensional constraints

Dimensional constraints are hidden when a drawing is closed and reopened.

Tools to control the display of dimensional constraints are located on:

- The **Dimensional Constraint** toolbar.
- the **Parametric ribbon** tab.
- the **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the DCDISPLAY command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.
- Hide All: hide all dimensional constraints in the drawing.

Nota:

- TheDYNCONSTRAINTMODEsystem 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.
- When hovering over an entity with a dimensional constraint, the blue constraint glyph () displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

6.5.3 Using expressions to define dimensional constraints

Dimensional constraints are named automatically upon creation.

The naming rules are as follows:

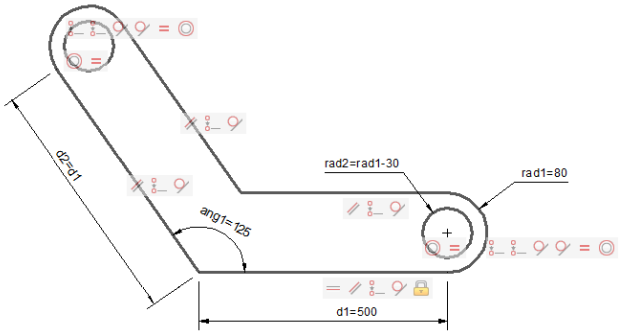
- distance: d1, d2, d3, ...
- radius: rad1, rad2, rad3, ...
- diameter: dia1, dia2, dia3, ..
- angular: ang1, ang2, ang3, ..

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.



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)

Function	Syntax
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)*
Smallest element in array	min(expression1;expression2)*

Function	Syntax
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

* Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Creating a new parameter

1 Right click **Parameters** in the Mechanical Browser panel.

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.

See the **Editing dimensional constraints in the Mechanical browser panel** procedure above.

Nota:

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

6.6 Using geometric constraints

Geometric constraints:

- Specify a geometric relation between 2 entities (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 in a constraint bar.

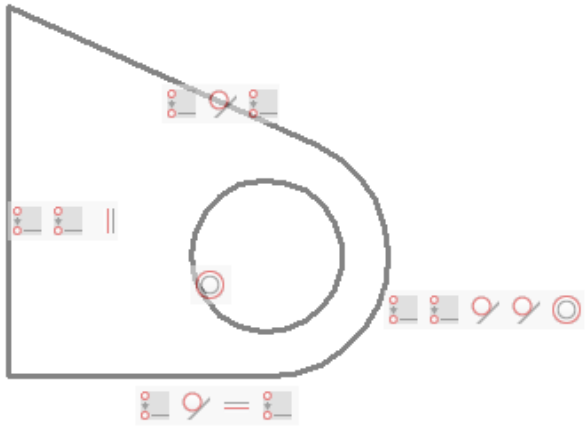
6.6.1 Creating geometric constraints

Tools to create dimensional constraints are located in:

- The **2D Constraints** toolbar.
- The **Parametric ribbon** tab.
- The **2D Constraints** fly-out menu of the **Parametric** menu.

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).



6.6.2 Displaying constraint bars

Constraint bars are hidden when a drawing is closed and reopened.

Tools to control the display of constraint bars are located on:

- The **Geometric Constraint** toolbar.
- The **Parametric** ribbon tab.
- The **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the CONSTRAINTBAR command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.

- Hide All hide all dimensional constraints in the drawing.

Nota:

- The value of theCONSTRAINTBARDISPLAYsystem 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.
- When hovering over an entity with a geometric constraint, the blue constraint glyph () displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

6.6.3 Controlling 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 theCONSTRAINTBARcommand restores the default position of the constraint bar.

Relocating 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.

Restoring the default position of constraint bars

1 Do one of the following:

- Click the**Show/Hide Geometric Constraint**tool.
- Launch the CONSTRAINTBAR command.

Prompts you:Select Entities.

2 Select the entities, then right click or press Enter.

BricsCAD® reports the number of selected entities.

Prompts you: Select option for constraints [Show/Hide/Reset] <Show>:

3 Do one of the following:

- Choose**Reset**in the context menu.
- Type**R**in the Command line, then press Enter.

6.6.4 Working with constraint bars

Controlling a constraint

1 Move the cursor over a constraint icon.

2 A tooltip displays, indicating the constraint type.

3 The associated entity (or entities) highlights.

4 The corresponding icon on the constraint bar of the associated entity highlights.

Deleting a constraint

1 Move the cursor over the constraint icon in the constraint bar.

2 Right click, then click**Delete**.

3 The constraint is deleted and the icon is removed from the constraint bar and from the constraint bar of the associated entity.

Hiding the constraint bar of an entity

Move the cursor of the constraint bar, then click the**Close**(x) button.

6.6.5 Deleting constraints

To delete all constraints from a selection set:

1 Do one of the following:

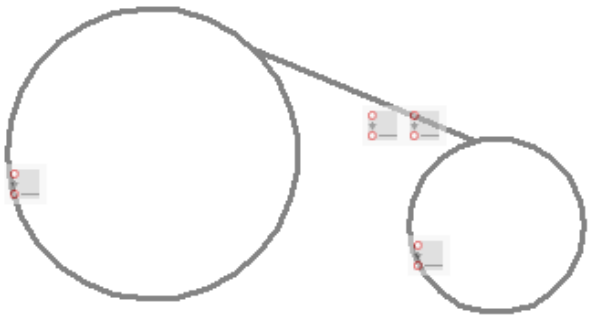
- Click the**Delete 2D Constraints**tool.
- Launch the DELCONSTRAINT command.

You are prompted: Select entities to delete all constraints:

2 Selectthe entities you want to delete the constraints from.

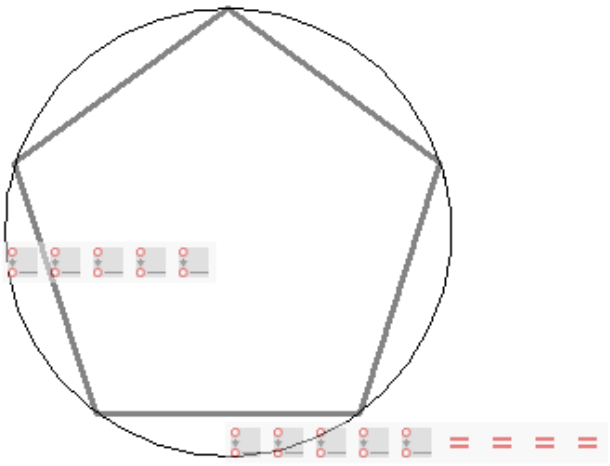
3 Right click to stop selecting entities and delete all constraints from the selection set.

6.6.6 Some examples

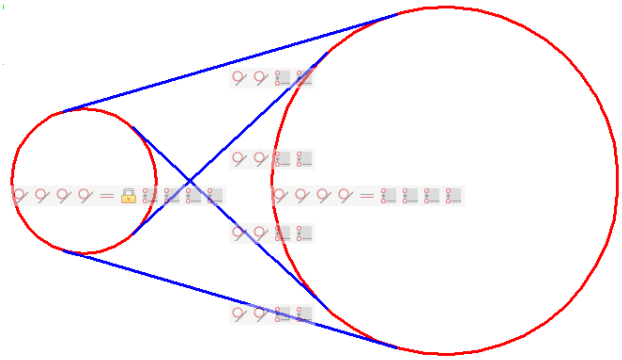


Coincident constraints between the endpoint of a line and two circles.

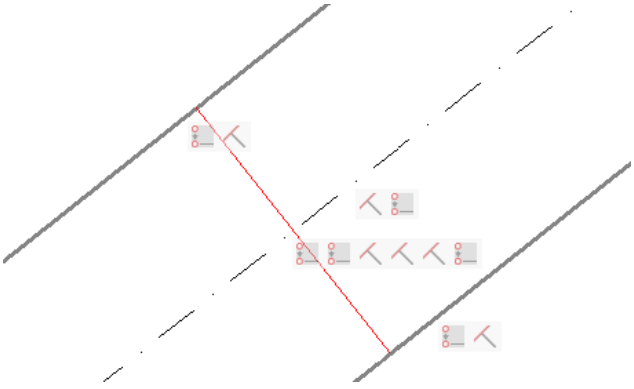
If the endpoints or center points of entities already coincide, the**Autoconstraint**option of theGCOINCIDENTcommand automatically applies coincident constraints to such points.



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.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



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 always lies centered between the two bold lines.

6.7 Dynamic dimensions

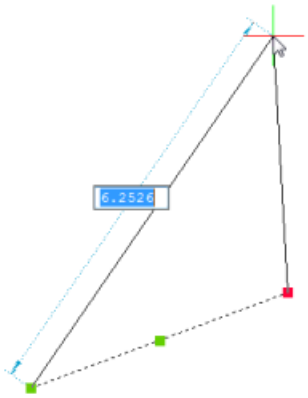
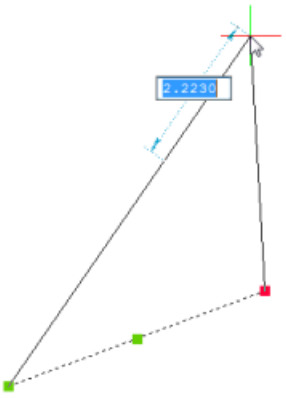
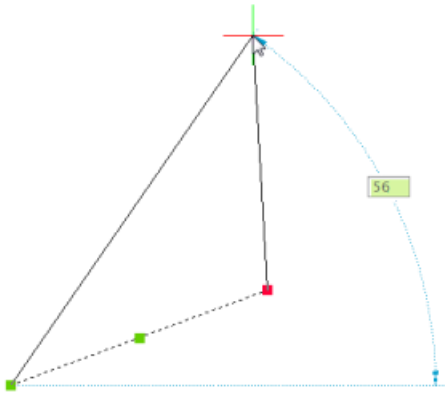
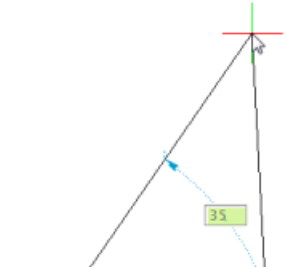
Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such, dynamic dimensions help you to focus on the drawing area.

The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

6.7.1 Defining the Dynamic Dimensions Settings

Right click the**DYN**field in theStatus Bar, then choose**Settings**in the context menu. The**Dynamic input**settings in the**Settings**dialog display.

Name	Title	Description
DYNMODE	Dynamic input mode	Controls whether and when dynamic dimensions display: • Editable dynamic dimensions: when creating or manipulating entities. • Tracking dynamic dimensions: when using entity snap tracking.

Name	Title	Description
DYNDIGRIP	Show dynamic dimensions	Defines which dynamic dimensions display when creating or grip editing entities, depending on the value of the DYNDIVIS system variable Resulting length: total length of the line or polyline segment  Extended length: incremental length of the line  Absolute angle: angle relative to the X-axis of the current UCS  Relative angle: angle relative to the original angle of the linear entity 

Name	Title	Description
DYNDIVIS	Dynamic dimension visibility	Defines the number of dynamic dimensions that display when creating or grip editing entities.
DYNPICOORDS	Default mode for dynamic coordinates input	Specifies the default mode in which coordinates are entered during dynamic input: Relative or Absolute.

Nota:

- The appearance of dynamic dimensions (color, linetype, position and transparencey) is controlled by a set of user preferences: DynDimColorHover, DynDimColorHot, DynDimDistance, DynDimLinetype, DynImputTransparence.
- If bitcode 16 (= format dynamic dimensions) of thePROPUNITSsystem variable is set, dynamic dimensions are formatted using a dedicated unit.
- Automatic unit conversion during length input can be used when bitcode 1 ofPROPUNITS(Format length properties) is set. For example, whenINSUNITSis set to **mm**, entering **2m** in a length property field will result in entering [2000.0000 mm]. Entering **1ft** gives [304.8000 mm]. If no unit is specified in the entry field, the unit as specified byINSUNITSis used.

6.7.2 Using dynamic dimensions when creating entities

1 Launch the command to create a 2D entity (e.g. LINE andDYNDIVIS= 1 or 2).

2 Specify the first point, then move the cursor to display the dynamic dimensions.

The**Length**dynamic dimension highlights.

3 Do one of the following:

- Type the desired length of the line, then press 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.
- Press he TAB key to jump to the**Angle**dynamic dimension.

4 Do one of the following:

- Type the desired angle, then press the Enter key to create the line.
- Type the desired angle, then press the TAB key to return to the**Length**dynamic dimension.
- Press the Enter key to create the line at the angle defined by the cursor position.
- Press the TAB key to return to the**Length**dynamic dimension.

Nota:

- IfDYNDIVIS= 0, only one dynamic dimension displays. Press 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 usingrelative coordinateswith respect to the previous point.

6.7.3 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 theDYNDIVISsystem 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 press 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.
- Press the TAB key to jump to the next dynamic dimension.

4 Repeat step 3 or press Enter to stop.

6.7.4 Using dynamic dimensions to measure entities

The following procedure applies to the 2DWireframe visual style only.

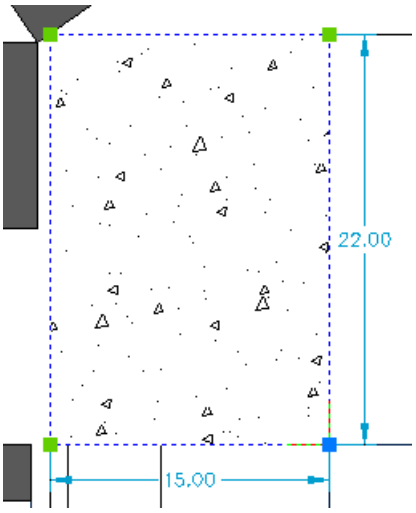
1 Select the entity or entities.

All grips on the selected entities display.

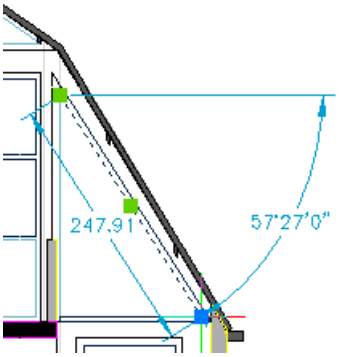
2 Hover the cursor over a grip.

The grip highlights and all dimensions related to the selected grip display.

Dynamic dimensions used to measure a rectangle



Dynamic dimensions used to measure a line



3 (option) Click to start editing the selected grip.

6.7.5 Using dynamic dimensions with tracking points

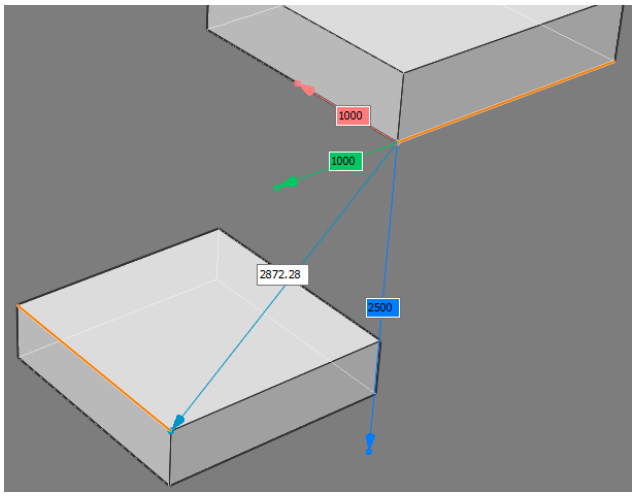
If the**Tracking dynamic dimensions**option of theDYNMODEsystem variable is set, dynamic dimensions display whenentity snap tracking pointsortemporary tracking pointsare acquired.

6.8 Nearest distance

The Nearest Distance feature allows to dynamically view and modify the distance between two selected entities or subentities. In addition, the aligned nearest distances along the X-, Y- and Z-axes of the WCS display.

Double clicking a Nearest Distance field:

- Toggles the display of the X-, Y- and Z-axis aligned distance fields.
- Zooms out to fully display the selected entities if necessary.



TheNEARESTDISTANCEsystem variable controls the display of the Nearest Distance feature.

6.8.1 Using Nearest Distance

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 a distance field to zoom out to fully display the selected (sub)entities and/or toggle the display of the aligned nearest distances along the X-, Y- and Z-axes.

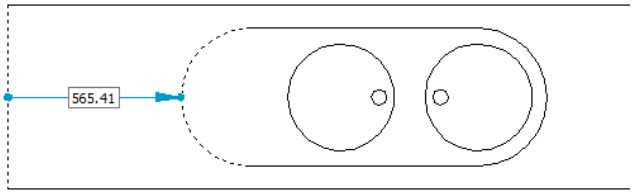
4 (option) Click a distance field and type a value.

The second entity is moved.

* Press and hold the Ctrl-key to select a subentity.

You can use the Nearest Distance feature to:

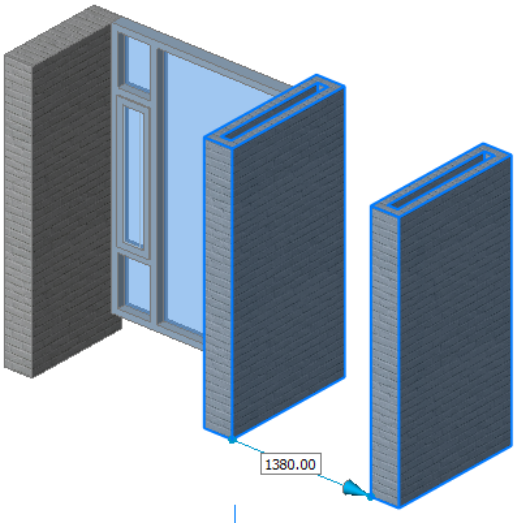
- Measure the distance between two entities or subentities.
- Adjust the distance between a (sub)entity and a block.



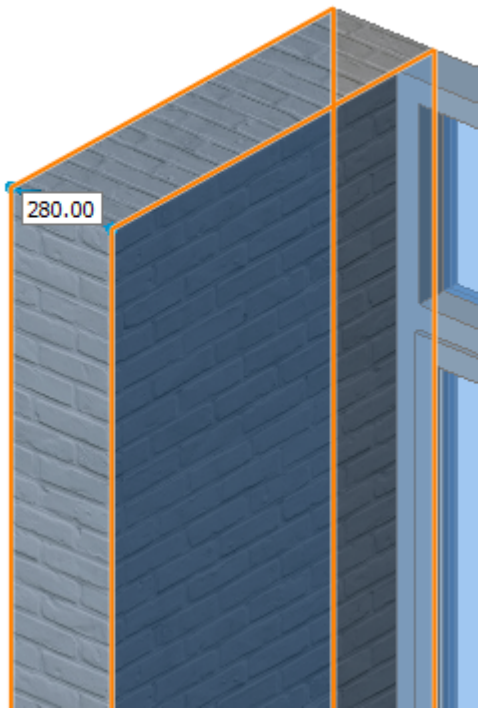
- Modify the dimensions of a rectangle.



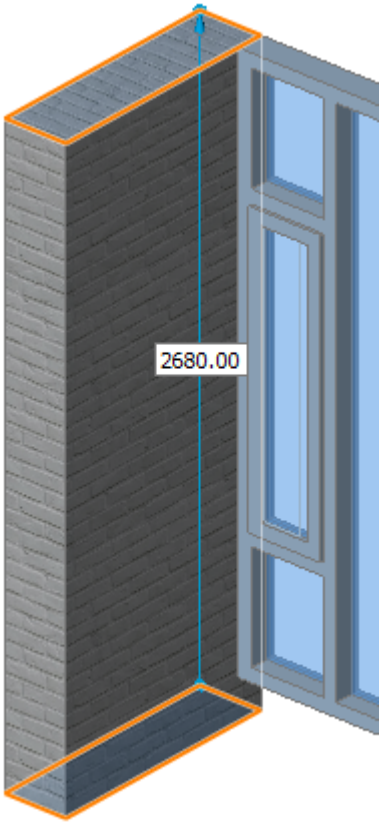
- Adjust the distance between 3D solids.



- Edit the width of a 3D solid.



- Measure and edit the height of 3D solid.



6.9 Coordinate input

When you create entities in a drawing, they are located in relation to the underlying Cartesian coordinate system of the drawing. A drawing has a fixed coordinate system called the**World Coordinate System**(WCS).

You can also define arbitrary coordinate systems which are calleduser coordinate systems(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

6.9.1 Relative coordinates

When you type the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called**Relative Coordinates**.

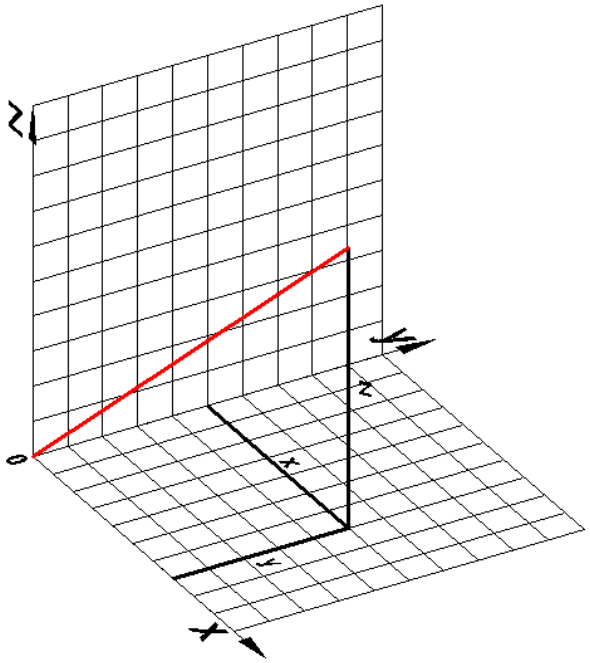
WhenDynamic Dimensionsare active and you type a value in the**Length**field and add a comma (,) the content of the**Length**field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the previous point.

6.9.2 Absolute coordinates

When a user defined coordinate system is active you can enter absolute coordinates (World coordinates) if you place an asterisk (*) in front. For example, *0,0 refers to the origin of the WCS (World Coordinate System).

6.9.3 Working with Cartesian coordinates

The Cartesian coordinate system uses 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.



- Type the x-, y- and z- coordinates separated by commas to enter the absolute Cartesian coordinates of a point.
- When the z-coordinate is omitted, the point is placed in the xy-plane (Z = 0).
- Type the @-character in front of the entry (@x,y), to specify the coordinates with respect to the previous point. This technique is called**Relative Cartesian coordinates**.

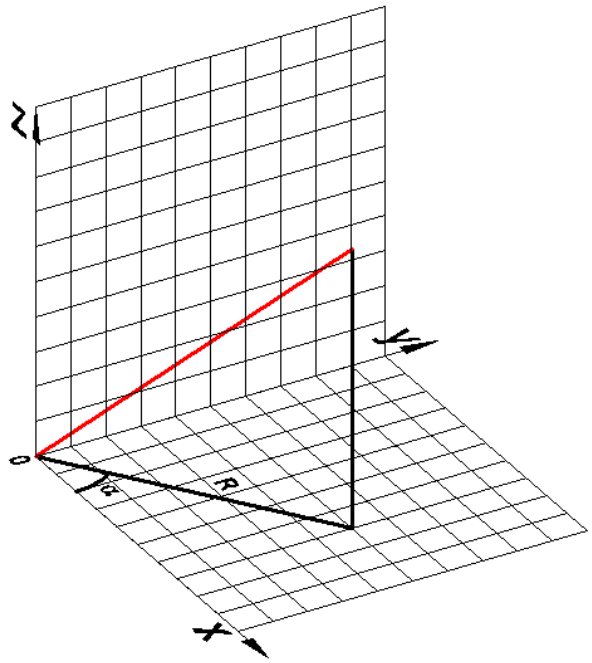
6.9.4 Working with cylindrical coordinates

A cylindrical coordinate system uses 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.
- When the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.
- Type the @-character in front of the entry to calculate the coordinates with respect to the previous point.

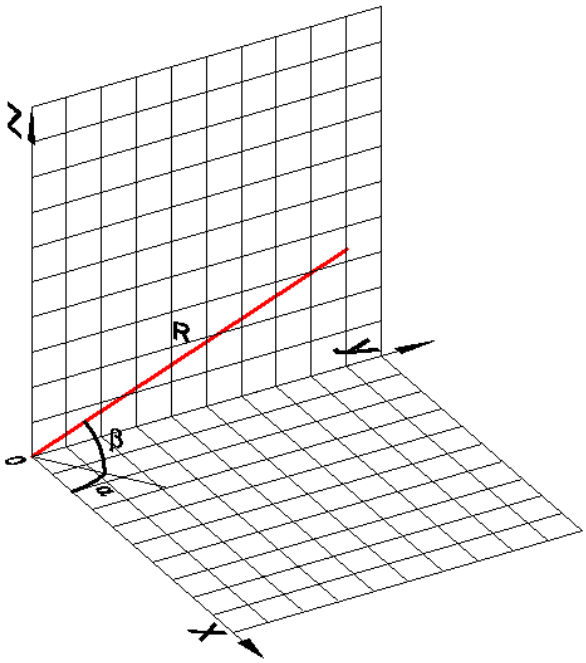


6.9.5 Working with spherical coordinates

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)

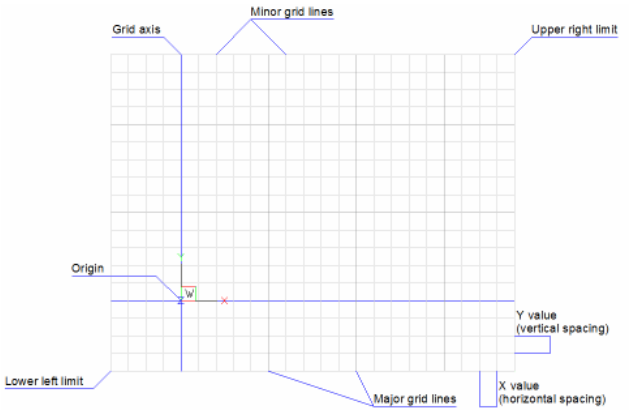


6.10 Snap and grid

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.

TheGRIDSTYLEsystem variable controls whethergrid lines or dotsdisplay. TheGRIDDISPLAYsystem variable controls thedisplay of the grid. The grid can also indicate how far the drawing limits extend.



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 theGRIDtool ( ) on the Settings toolbar or theGRID field in theStatus Bar, or pressCtrl+Gto toggle grid display on/off.

- Click theSNAPtool () on the Settings toolbar or theSNAP field in theStatus Bar, or pressCtrl+Bto toggle snap on/off. The setting is saved through the SNAPMODE system variable.

  If SNAPTYPE = 2,adaptive grid snapis used and SNAPMODE is denied.
- BothSNAPandGRIDcan be set differently in eachviewport.
- TheSnapoption of theGRIDcommand equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

6.10.1 Displaying the Snap and grid settings

Do one of the following:

- In theStatus Bar, right click on theSNAPor GRID field, then chooseSettingsin the context menu.
- In theSettingsdialog go to: **Drawing** > **Drafting** > **Coordinate Input** > **Snap** > **Grid**.

6.10.2 Synchronizing snap and grid spacing

1 Launch the GRID command.

  Prompts you:

  Grid is on. Enter grid spacing or [grid OFf/Snap/Aspect] <10>:

2 Type**s**, then press Enter.

  The grid spacing matches the snap spacing.

3 (Option) Use the Aspect option of the SNAP command to adjust the spacing.

Nota: The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using theGRID command or in the Settings dialog box.

6.10.3 Using drawing limits

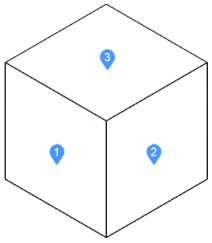
- Execute the LIMITScommand to set the drawing limits.
- Click the**Drawing Limits**item () in the Settings menu to toggle theLIMCHECKsystem variable on/off.

  If LIMCHECK = ON, it is not possible to draw outside the drawing limits.
- The**Display beyond LIMITS area**option of theGRIDDISPLAYsystem variable defines whether or not the grid displays beyond the drawing limits.

6.10.4 Using isometric snap

Isometric grid allows to create two-dimensional isometric drawings, Which simulate a three-dimensional view on a two-dimensional plane. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as**Left** (1),**Right** (2)and**Top** (3). You cannot alter the arrangement of these planes. The three isometric axes are 30 degrees, 90 degrees, and 150 degrees.



Do the following:

- Set SNAPSTYL = 1 (isometric Snap).
- Set GRIDSTYLE = 1 (dotted grid).
- Set SNAPISOPAIR: 0 = Left, 1 = Top, 2 = Right.

  Press the F5 function key to cycle the isometric planes.
- Set ORTHOMODE = On to constrain the movement of the crosshairs to the current isometric plane.
- Make sure SNAPANG = 0.

6.10.5 Setting the snap angle

You can rotate a dotted grid using the SNAPANG system variable (ifSNAPANG is different from 0, grid lines do not display).

1 Display the**Snap/Grid**settings in theSettingsdialog.

2 Select SNAPANG system variable.

3 Type a new value in the setting field.

4 Close the Settings dialog.

Nota: You can also set theSnap Angleby typing**snapang**in the Command line. When in a command, type**'snapang**(with an apostrophe in front), to set theSnap Anglevariable transparently (= without interrupting the running command).

6.11 Adaptive grid snap

The adaptive grid snap feature lets you create, edit,nudgeandmanipulateentities using concise values without keyboard entry.

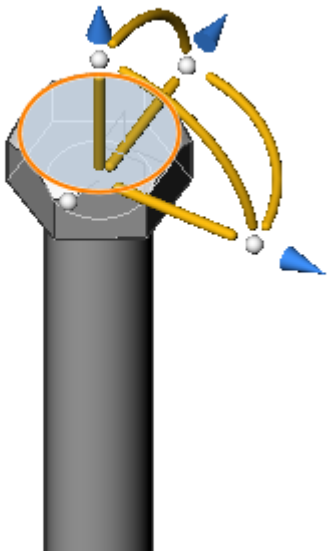
The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active ifSNAPTYPE = 2. This system variable overrides theSNAPMODE system variable.

The AdaptiveGridStepSize user preference controls the smallest available step size; default = 4 pixels.

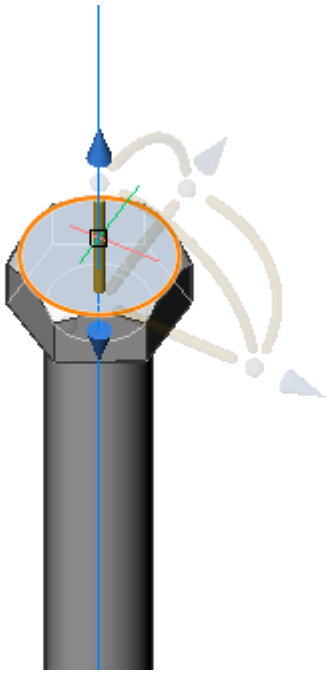
6.11.1 Using adaptive grid snap with the Manipulator

1 Move the cursor over an entity or subentity, then hold down the left mouse button a little bit longer to select the entity (= long left click).

The Manipulator widget displays:



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 Manipulator are faded.



3 Click to select the axis, then 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.

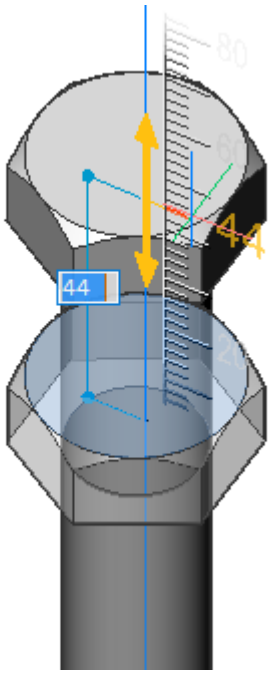
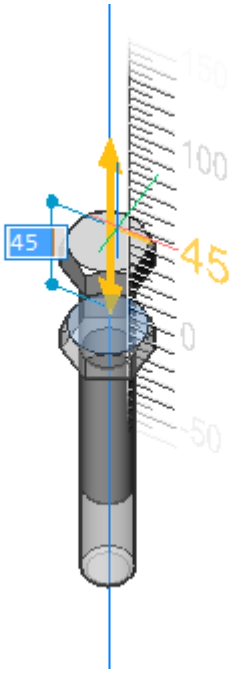


Figura 2: Ruler divisions at 2 mm

Figura 3: Ruler divisions at 5 mm



4 (option) Zoom in or out to adjust the dynamic grid.

5 Do one of the following:

- Click to accept the current value in the Ruler.
- Type a value in the dynamic field, then press Enter.

6.12 Nudge

The Nudge drawing aid allows you to move selected entities or subentities in small increments along the X, Y, or Z axis using the keyboard.

6.12.1 Using Nudge

1 Select one or more entities or subentities (for example polyline segments or 3D solid faces).

2 Do the following to move the selection:

- Along the X axis of the current coordinate system:

  Hold down the Ctrl key, then use the left/right arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Y axis of the current coordinate system:

  Hold down the Ctrl key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Z-axis of the current coordinate system:

  Hold down the Shift key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the Z axis of the UCS: xx*

* The step size used by the NUDGE functionality is based on theadaptive grid snapresolution. Zoom in/out to decrease/increase the step size.

6.13 2D entity snaps

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 theAPBOXsystem 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.

TheDRAGSNAPsystem variable controls the snap behavior while dragging, providing an increased WYSIWIG experience. DRAGSNAP controls whether rubber band 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.

TheOSOPTIONSsystem 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.

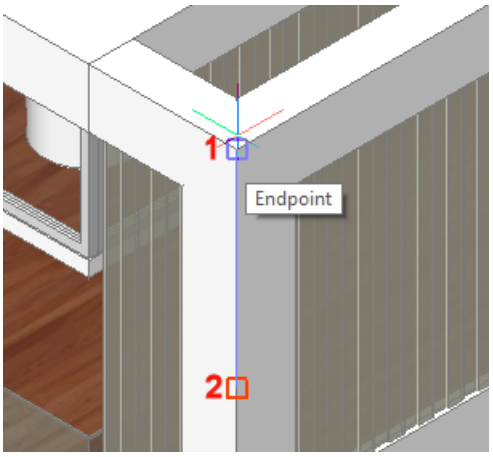
- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

Nota: Press theTABkey, without moving the mouse, to cycle through all possible entity snaps available at the current cursor position.

6.13.1 Defining the 2D Entity Snap system variables

Right click the ESNAP field in the Status bar then choose settings in the context menu. The Settings dialog box displays with the 2D and 3D entity snaps settings expanded.

6.13.2 2D Entity Snap options

Icon	Description
	Sets the OSNAPZ system variable to Ignore entity snap elevation, replacing the Z coordinate of the snapped point with the current value of the ELEVATION system variable.  The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height.
	Disables the Ignore hatches option of the OSOPTIONS system variable, thus allowing to snap to hatches.
	Disables the Entity snap to negative-z option of the OSOPTIONS system variable, thus allowing entity snaps with a negative z-value when aDynamic UCSis active.
	Disables the Ignore endpoints of dimension extension lines option of the OSOPTIONS system variable, thus allowing entity snaps to dimension extension lines.

6.13.3 Entity Snap Modes

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.
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.

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
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.
NODE			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.

* Type this letter to quickly select the entity snap mode in the right click context menu.

** Type the transparent shortcut in the Command line when a command prompts for a point.

Other running entity snaps are temporarily disabled.

6.13.4 Setting the Entity snaps

Do one of the following:

- 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.
- Click the buttons on the Entity Snaps toolbar.

  The buttons of the currently active entity snap modes are pressed.

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.

To toggle theEntity Snapson/off:

- Click theESNAPfield in theStatus Bar.
- Press the**F3**function key.

6.13.5 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 and 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.

6.13.6 Snapping to the extension of two entities

1 If not already on, turn on the**Extension**Entity Snap mode.

2 Launch an entity creation command, for example LINE.

3 Move the cursor over the endpoint of the first entity (1).

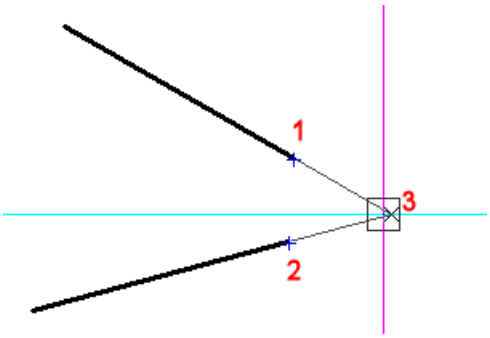
A small red cross indicates the entity is marked for extension.

4 Move the cursor over the endpoint of the second entity (2).

A small red 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).



6 Click to accept the snap point.

Nota: You can snap to the extension of lines, polylines, arcs and elliptical arcs.

6.13.7 Using the Middle of 2 points snap mode

1 When prompted to specify a point, do one of the following:

- Activate the**Snap to the middle of 2 points**snap mode ( ).
- Type**mtp**or**m2p**in the Command line and press Enter.

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.

6.13.8 Using the Parallel entity snap mode

1 Make sureEntity Snap Trackingis active.

2 Launch an entity create command to create a linear entity (for example Line).

3 Specify the first point of the entity.

The line displays dynamically.

4 Press and hold the Shift key, then right click and choose**Snap to parallel**( ) in the context menu.

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.

6.14 3D entity snaps

In addition to the2D entity snaps, 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 bitcodes of the selected snap modes.

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.
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 ofsplines.
Point Cloud			Snaps to point cloud points.

6.14.1 Setting 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.
- 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, temporarily disabling all other selected snap modes.
- Run the3DSONAPor-3DOSNAPthe commands.
- Shift + Right click, hover over**3d Entity Snap**in the context menu and click the 3D entity snap option on the flyout menu.

6.15 Polar tracking

Polar tracking (POLAR) allows you to draw at exact angles:

- At fixed intervals, starting from 3 o'clock (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 theSnap 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 thecolor defined by the SnapMarkerColoruser preference.

- 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).
- Turning onPOLARautomatically disablesORTHO (Orthogonal Mode) and vice versa.
- The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

6.15.1 Defining the tracking settings

1 Do one of the following:
   - Right click on thePOLARor the STRACK field in theStatus Bar, then right click and select**Settings**in the context menu.
   - Open the**Settings**dialog and expand the**Coordinate input**subcategory under**Drafting**in the**Drawing**settings class.

     Expand the**Snap Tracking**settings group.

2 Adjust the settings:
   - AUTOSNAP system variable: controls the behavior of Polar Tracking and whether polar tracking and/or entity snap tracking is active.
   - POLARANG system variable: sets the polar angle increment in degrees; default = 90.
   - POLARADDANG system variable: sets additional angles; separate angles by semicolons.
   - POLARMODE system variable: controls the behavior of polar and entity snap tracking.
   - Relative: the angles are measured with respect to the direction of the snapped entity; otherwise angles are measured according to the current coordinate system (WCS or UCS).
   - Use polar angle settings for entity snap tracking also.
   - Whether additional angles (POLARADDANG) are used or not.
   - Use the Shift key to acquire entity snapping points.
   - TRACKPATH system variable: controls whether and how the track paths display: full screen or partially.

6.15.2 Locking a tracking line

Press the Shift key while a snap track marker line displays to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Press the Shift key again to unlock.

A locked tracking path changes to the color defined by theSnapMarkerColoruser preference.

6.15.3 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. It is recommended to havedynamic dimensions(DYN) active as well.

1 Whenever a command prompts for a point, do one of the following:
   - Click the middle mouse button (mouse wheel).
   - Activate the**Temporary tracking points**entity snap mode ( ).
   - Type**TK**in the Command line and press Enter.

     The command is suspended temporarily.

     You are prompted: Specify first temporary tracking point:

2 Enter a point, for example 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.

6.16 Entity snap tracking

Entity Snap Tracking(STRACK) assists you in drawing entities with respect to other entities.

WhenEntity Snap Trackingis active, BricsCAD® displays temporary tracking lines from entity snap points.At least oneEntity Snapmode must be on if you want to useEntity Snap Tracking.

Click theSTRACKfield in theStatus barto enable/disableEntity Snap Tracking.

Entity Snap Trackinglines are parallel to the X- and Y-axis of the current UCS.

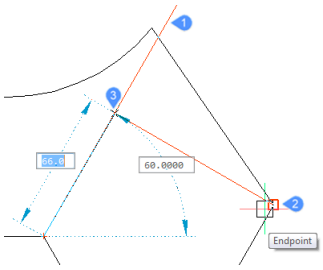
If bitcode 2 of the POLARMODE system variable is checked, the polar angles used inPolar Tracking(interval and additional angles) are also used inEntity Snap Tracking.

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 bitcode 8 of the POLARMODE is checked, 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.

Nota: The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

6.16.1 Locking a tracking line

Press the Shift key when a tracking line displays to lock the tracking line. Locked tracking lines display in the color defined by theSnapMarkerColoruser preference (1). Entity snaps (2) or specified points are projected perpendicularly onto a locked tracking line (3). Press the Shift key again to unlock a tracking line.



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.

6.16.2 Using Entity Snap Tracking

1 Launch a drawing command, for example LINE.

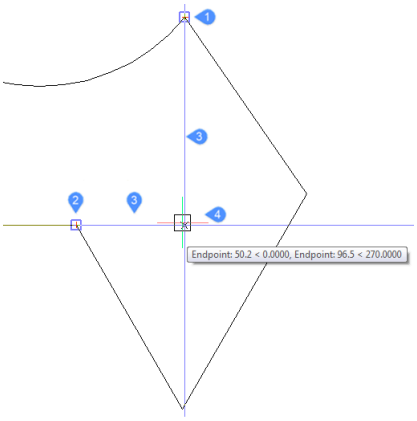
Prompts you: Start of line or [Follow] <Last point>:

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.

5 Click to accept theEntity Snap Trackingpoint.

6.17 User coordinate systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. TheELEVATION system variable specifies the distance from the XY-plane of the current coordinate system. IfELEVATION= 0 (default), 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 define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS). You can create as many UCS as you want, saving or redefining them to help you construct entities in the three-dimensional space.

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 the new viewpoint.

Nota:

- You cannot delete or modify the WCS.
- When theUCSORTHOsystem variable is On, an orthographic UCS is restored automatically when the related orthographic view is restored by the-VIEWcommand or selected on theLook Fromcontrol.

  An orthographic UCS has its origin at the WCS origin or the UCS referred to by the UCSBASE system variable.

  Orthographic views are: top, bottom, left, right, front, back.
- When multiple viewports exist, theUCSVP(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).
- When the UCSFOLLOW system variable is On, the plan view, zoomed to extents, is generated automatically whenever the coordinate system changes.
- ThePLANcommand restores the Plan view (Top view) of the current coordinate system (UCS or WCS).

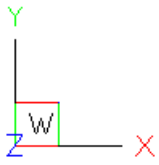
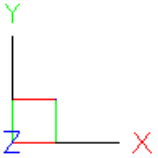
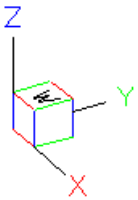
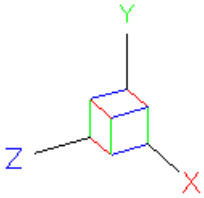
6.17.1 Controlling the display of the coordinate system icon

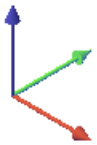
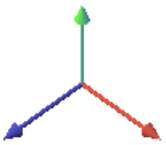
TheUCSICON and UCSICONPOS system variables control the display and location of the UCS icon:

- UCSICON:
  - **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 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 theUCSICONPOS

system variable. When the**At Origin**option is not checked, the icon always displays in the corner of the viewport defined byUCSICONPOS.

- UCSICONPOS : Defines the corner of the viewport for the non-origin location of the UCS icon.

2D Wireframe visual style			
			
WCS Top View	UCS Top View	WCS Isometric View	UCS Isometric View

Rendered Visual Styles			
			

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.

The color of the three axes indicate the orientation of the three-dimensional space:

- X-axis: red
- Y-axis: green
- Z-axis: blue

The color of the axes is controlled by theCOLORX,COLORYandCOLORZsystem variables.

6.17.2 Defining a UCS

1 Type **ucs** at the command prompt.

You are prompted:

Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:

2 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

3 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

4 Specify a point to define the positive Y-axis.

The UCS is defined.

6.17.3 Defining and saving a User Coordinate System

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

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

3 Press Enter to accept the default option (define the UCS using 3 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 Drawing Explorer dialog reopens.

7 Click in the UCS Name field of the newly defined UCS to replace the <NewUCS> default name.

8 Close the Drawing Explorer dialog.

6.17.4 Restoring a saved UCS

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

2 Click the blank tile in front of the UCS name to make it current.

3 Close the Drawing Explorer dialog box.

6.17.5 Restoring the WCS

Type **ucs** in the Command line, then press Enter twice (= choosing the default option of the UCS command).

6.18 Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with the entity under the cursor.

TheUCSDETECTsystem variable controls the supported entity types:

- 1: faces of 3d Solids (the default)

- 2 :other entity types: point, line, polyline, 2D polyline, 3D polyline, ray, xline, arc, circle, ellipse, spline, text, mtext, solid, 3Dface, trace, block insert, viewport, mline, leader, mleader, hatch, helix, camera, light, section, shape, pdf underlay and image.

Right click the DUCS field in the status bar to select the supported entity types.

This feature is controlled by user preferences.

- RedHilite_DUCSLocked_Face_Colorcontrols the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key.
- RedHilite_DUCSLocked_Face_Alphacontrols the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

Nota: It is recommended to set UCSFOLLOW = Off when using the dynamic UCS feature.

6.18.1 Toggling the dynamic UCS

Do one of the following:

- Press the**F6**function key.
- Click the**DUCS**field in thestatus bar.

6.18.2 Using the Dynamic UCS

1 With the Dynamic UCS feature active, launch an entity creation command, for example Draw.

2 Move the cursor over an entity or 3D solid face.

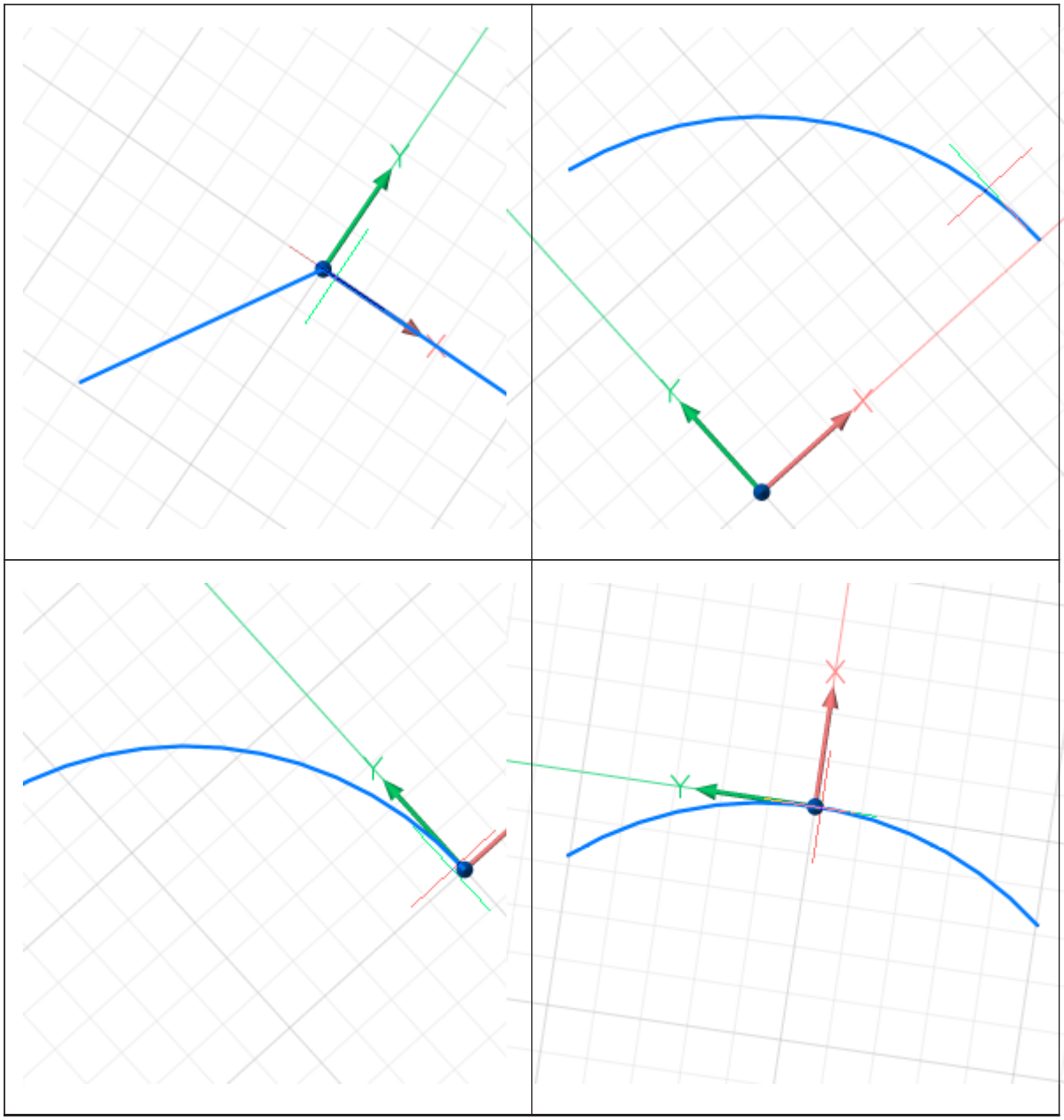
The entity or face highlights.

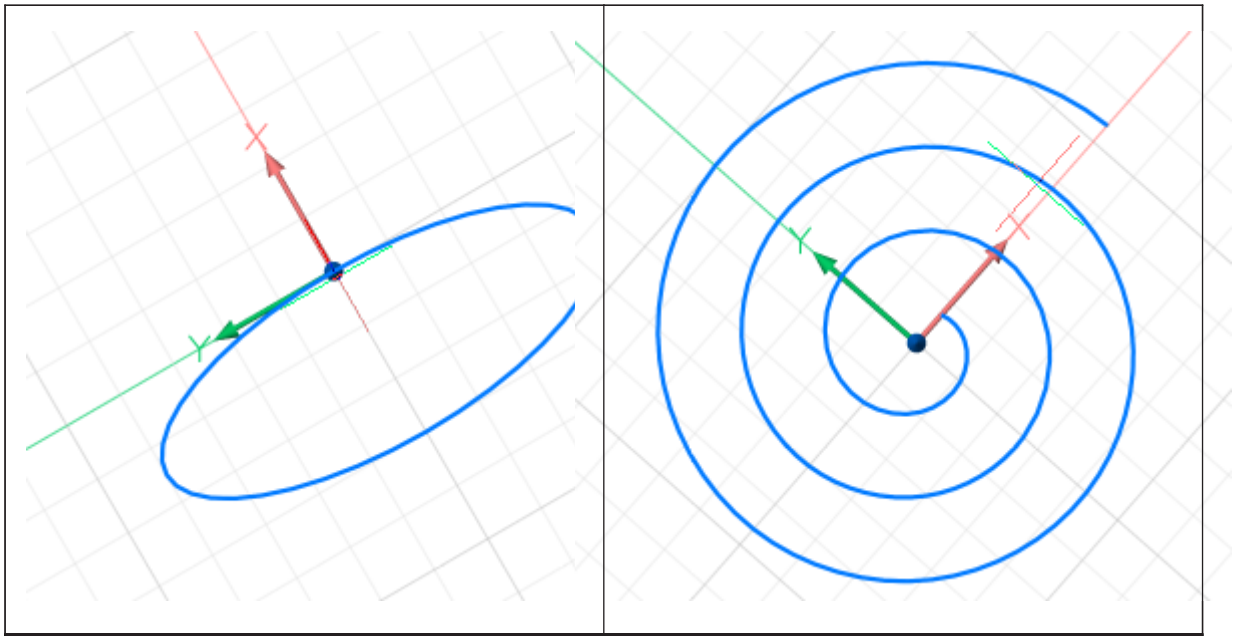
If GRIDMODE is On, the grid is aligned with the hovered entity or face.

3 IfUCSICON= 3 (On at Origin), the UCS icon displays.

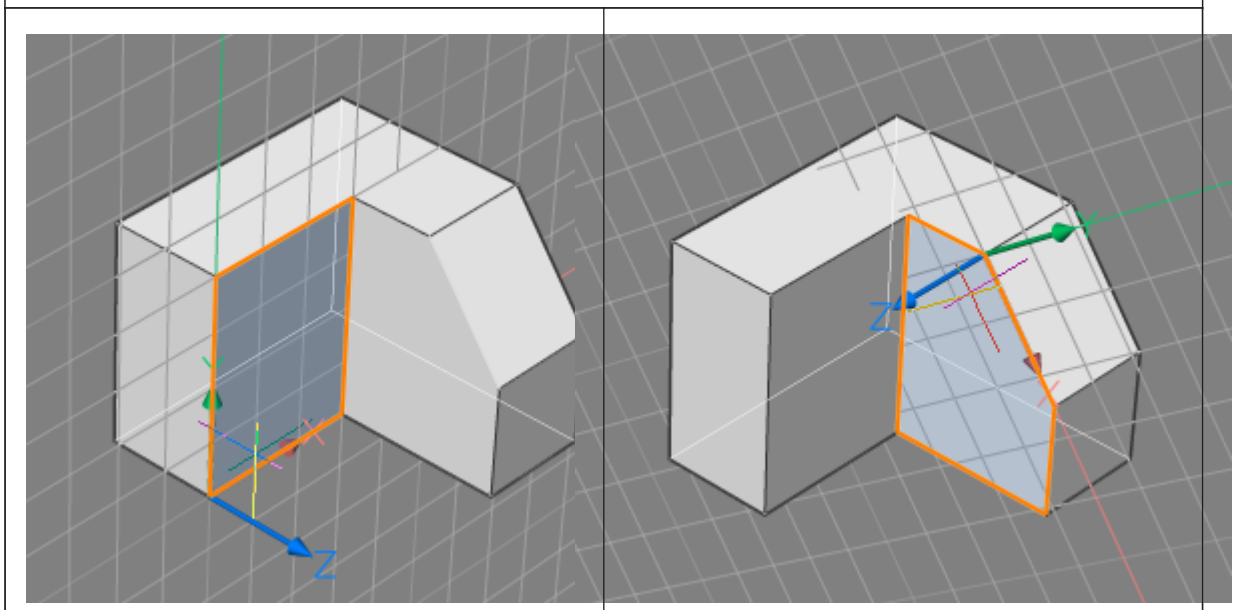
4

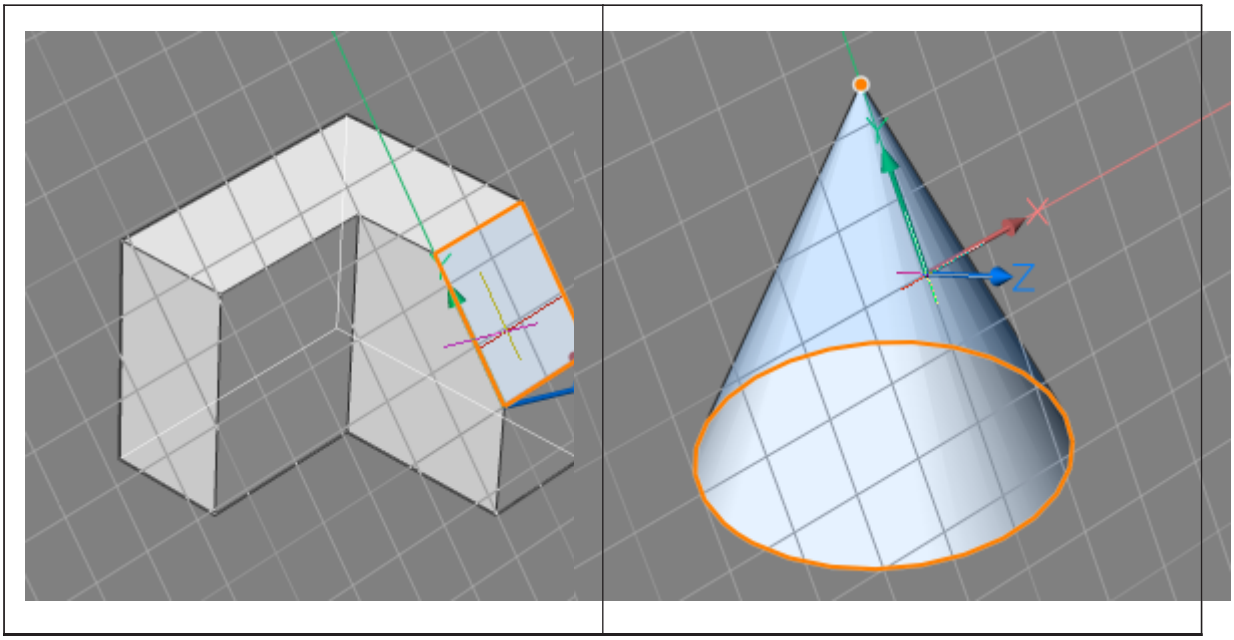
Dynamic UCS on other entities





Dynamic UCS on 3D solid faces





The dynamic UCS is oriented depending on how the cursor touches the hovered entity or face, thus providing users a way to manipulate the orientation of the DUCS. On 3D solid faces the origin of the DUCS is set to the start point of the solid edge and the X-axis is chosen co-directional to the tangent at the start point of the edge. On curved faces, the X-axis is tangent to the face, the Y-axis lies in the face and the Z-axis is perpendicular to the face.

5 (option) Tap the Shift key to lock the DUCS, which allows you to start drawing outside the hovered entity or face. Tap the Shift key again to unlock the DUCS.

6 Execute the entity creation command.

Nota: When bitcode 2 of the OSOPTIONS system variable is selected, entity snap ignores negative Z-values.

7 When the command concludes, the current coordinate system (WCS or UCS) is restored.

6.19 Measuring

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.

Nota: BricsCAD® reports the results in the Command line or in the Prompt History window when the Command line is closed. Depending on the number of prompt history lines in the Command line, it can be necessary to press the F2 function key to open the Prompt History window. Press the F2 function key again to close the Prompt History window.

6.19.1 Measuring distances

To measure a distance between 2 points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Hover over the second point using an appropriate entity snap mode.

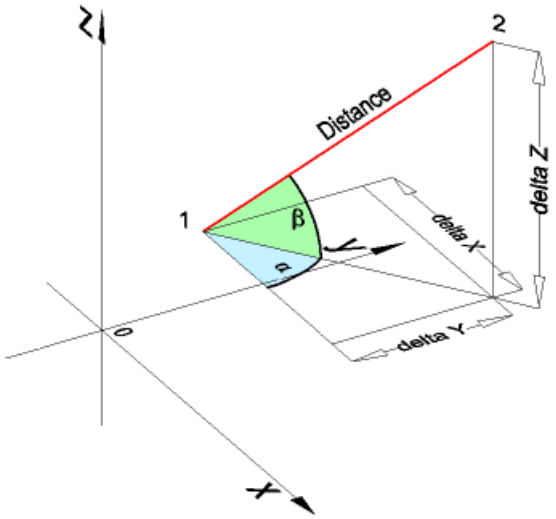
When Dynamic Input (DYN) is On, the distance displays in thedynamic dimensionfield.

4 Click to confirm the second point.

BricsCAD® prints the results in the Command line:

Distance, Angle in XY Plane (α), Angle from XY Plane (β)

Delta X, Delta Y, Delta Z



The values are in drawing units and with respect to the currentcoordinate system.

To measure a distance between multiple points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Choose the Multiple points option.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

4 Identify the second point.

BricsCAD® displays the distance between the two points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

5 Identify the next point.

BricsCAD® displays the total distance between the points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

6 Keep adding points, then press Enter or right click to conclude.

6.19.2 Measuring lengths

To measure the length of single linear entity, click the entity then read the **Length** field in the Properties panel.

To combine the lengths of a series of linear entities:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Choose the **Add areas together** option.

Prompts you: Draw first point to add to current area or [add Entities to area/Subtract from area]:

3 Choose the **ad Entities** option.

Prompts you: Select entities to add to area:

4 Select the first linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

5 Select the second linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

6 Keep selecting linear entities or press Enter or right click to stop.

Nota: You can combine the lengths of lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

6.19.3 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 panel.

To measure areas and perimeters by specifying points in a drawing:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Click the first point.

Prompts you: Next point.

3 Click the second point.

Prompts you: Next point.

A dashed outline displays dynamically.

4 Keep adding points, then press Enter or right click.

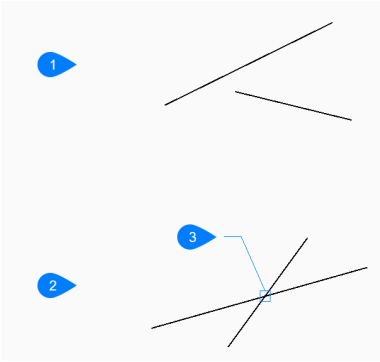
BricsCAD® reports in the command line:

Area = xx, Perimeter = xx

6.20 Entity snaps overview

6.20.1 Apparent intersection entity snap

The Apparent intersection entity snap enables the cursor to snap to the intersections of entities, even when they only appear to intersect in 3D space.



1 In one 3D viewpoint, the lines do not intersect.

2 In another 3D viewpoint, the lines appear intersect.

3 Apparent intersection.

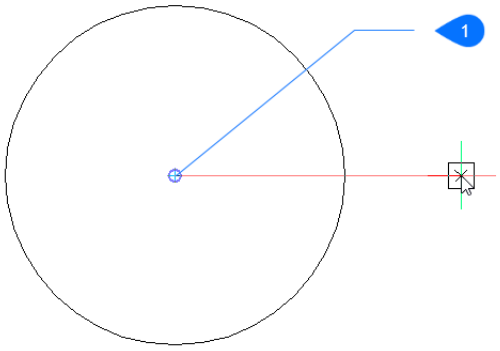
6.20.2 Center entity snap

The Center entity snap enables the cursor to snap to the center point of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces

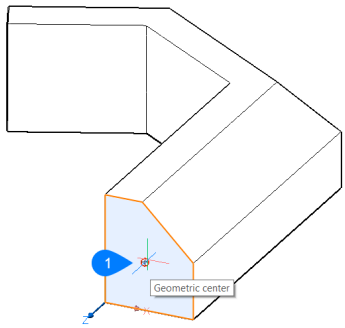
- Elliptical 3D faces



1 Center of circle.

6.20.3 Center of a Face 3D entity snap

The Center of a Face 3D entity snap enables the cursor to snap to the geometric center of a 3D face.



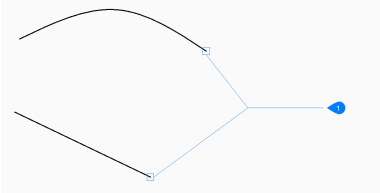
1 Geometric center entity snap.

6.20.4 Snap a entidade pela Extremidade

O snap à entidade pela Extremidade permite que o cursor se encaixe nos pontos finais das entidades.

Incluindo:

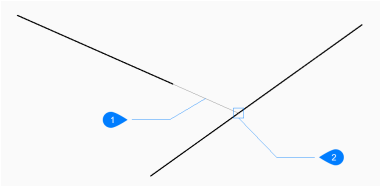
- Linhas
- Arcos
- Segmentos de polilinha
- Splines abertas
- Bordas 3D



1 Snap entidade na Extremidade.

6.20.5 Extension entity snap

The Extension entity snap enables the cursor to snap to the intersection of one or two extended entities.



1 Extension

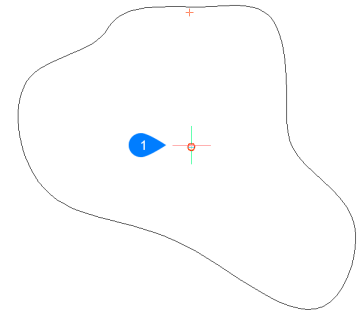
2 Extension entity snap.

6.20.6 Geometric center entity snap

The Geometric center entity snap enables the cursor to snap to the centroid of closed entities.

Including:

- Polylines
- Splines
- Regions
- Planar 3D faces



1 Geometric center entity snap.

6.20.7 Insertion entity snap

The Insertion entity snap enables the cursor to snap to the insertion points of entities.

Including:

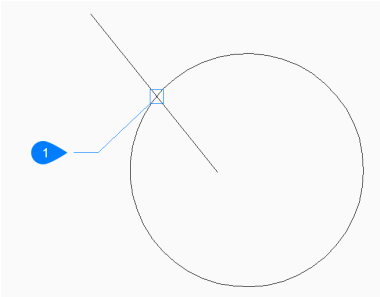
- Text
- Mtext
- Tables
- Attribute text
- Blocks
- External References
- PDF References
- Raster images



1 Insertion entity snap.

6.20.8 Snap a entidade na Interseção

O snap à Interseção da entidade permite que o cursor se encaixe na interseção de entidades que se cruzam fisicamente.



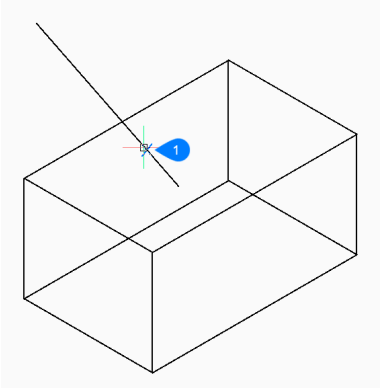
1 Snap a entidade na Interseção.

6.20.9 Snap à Interseção a entidade 3D

O snap à entidade 'Intersecao 3D' permite que o cursor se encaixe na interseção de faces (3D) com entidades lineares.

Incluindo:

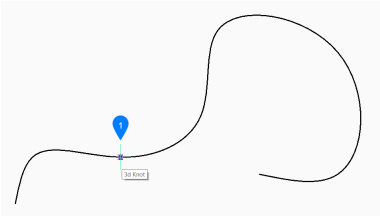
- Linhas
- Segmentos de polilinha
- Bordas 3D
- Linhas de rastreamento polar
- Linhas de rastreamento de snaps a entidade



1 Snap a entidade na interseção 3D.

6.20.10 Knot 3D entity snap

The Knot 3D entity snap enables the cursor to snap to knots on splines.



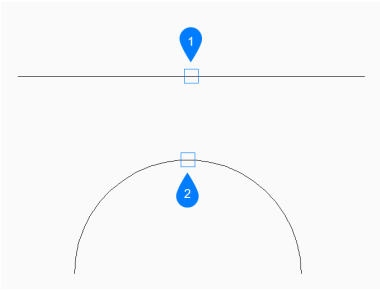
1 Knot 3D entity snap.

6.20.11 Midpoint entity snap

The Midpoint entity snap enables the cursor to snap to the midpoint of entities.

Including:

- Lines
- Polyline segments
- Arcs
- Elliptical Arcs
- Splines
- 3D edges

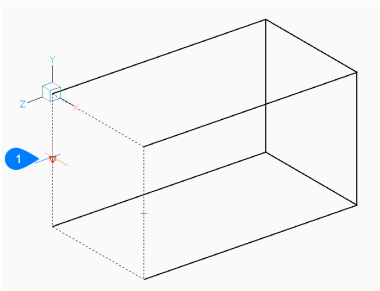


1 Midpoint (of line).

2 Midpoint (of arc).

6.20.12 Midpoint 3D entity snap

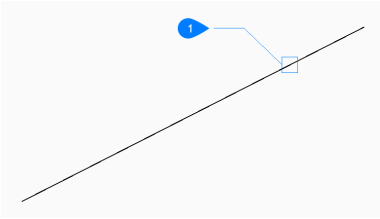
The Midpoint 3D entity snap enables the cursor to snap to the midpoint of 3D edges.



1 Midpoint 3D entity snap.

6.20.13 Nearest entity snap

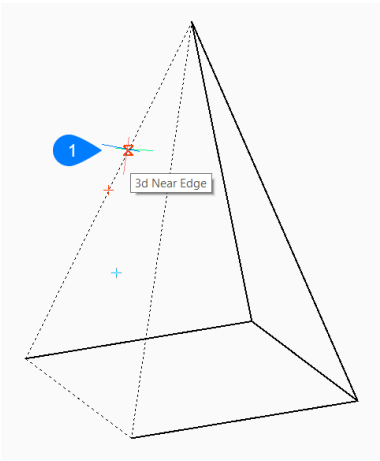
The Nearest entity snap enables the cursor to snap to 2D or 3D geometry nearest to the cursor.



1 Nearest entity snap.

6.20.14 Nearest 3D entity snap

The Nearest 3D entity snap enables the cursor to snap to a point on a 3D entity that is nearest to the cursor.



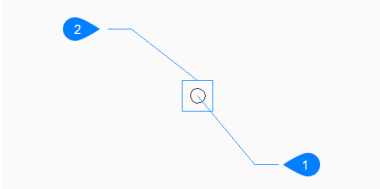
1 Nearest 3D entity snap.

6.20.15 Node entity snap

The Node entity snap enables the cursor to snap to point entities and the insertion points of entities.

Including:

- Text
- Mtext
- Attributes
- Tables

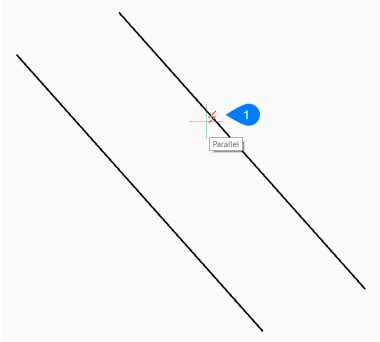


1 Point.

2 Node entity snap.

6.20.16 Parallel entity snap

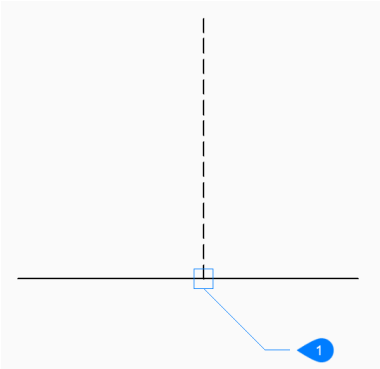
The Parallel entity snap enables the cursor to snap to a point parallel to an existing entity.



1 Parallel entity snap.

6.20.17 Perpendicular entity snap

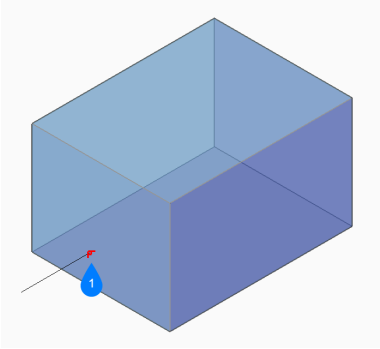
The Perpendicular entity snap enables the cursor to snap to a point perpendicular to an existing entity.



1 Perpendicular (to entity).

6.20.18 Perpendicular on a face 3D entity snap

The Perpendicular on a face 3D entity snap enables the cursor to snap to a point perpendicular to a 3D face.



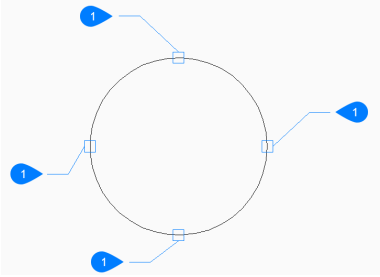
1 Perpendicular to a face 3D entity snap.

6.20.19 Quadrant entity snap

The Quadrant entity snap enables the cursor to snap to the quadrant points of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces
- Elliptical 3D faces



1 Quadrant.

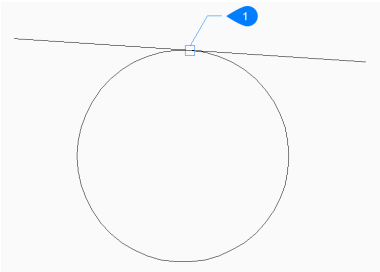
6.20.20 Tangent entity snap

The Tangent entity snap enables the cursor to snap to the tangent points of curved entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Splines

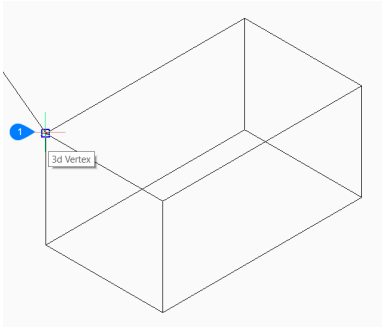
- Circular 3D faces
- Elliptical 3D faces



1 Point of tangency.

6.20.21 Vertex 3D entity snap

The Vertex 3D entity snap enables the cursor to snap to the closest vertex of a 3D entity.



1 Vertex 3D entity snap.

7. Visualizar seu desenho

7.1 View commands overview

BricsCAD provides various tools and commands to control the display and view orientation of your drawing.

These tools can be found in the **View** menu, toolbar or ribbon tab.

Icon	Tool Name	Command	Description
	Redraw	REDRAW	Refreshes the screen display in the current viewport.
	Redraw All	REDRAWALL	Refreshes the screen display in all open viewports.
	Regenerate	REGEN	Recalculates the screen display in the current viewport.
	Regenerate All	REGENALL	Recalculates the screen display in all open viewports.
	Update Fields	UPDATEFIELD	Updates the selected fields.
	Real Time Pan	RTPAN	Pans the view dynamically.
	Pan	PAN	Pans the view.
	Real Time Zoom	RTZOOM	Zooms in/out dynamically.
	Zoom In	ZOOM + *I(*)*	Zooms in on the center of the window by a factor of 2.
	Zoom Out	ZOOM + *O(*)*	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents	ZOOM + *E(*)*	Displays all the entities in the drawing (referred to as the drawing extents).
	Zoom Window	ZOOM + *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	ZOOM + *P(*)*	Restores the displayed view prior to the current one.

Icon	Tool Name	Command	Description
	Zoom All	ZOOM + *A(*)*	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	ZOOM + *L(*)*	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	ZOOM + *C(*)*	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	ZOOM + *R(*)*	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around	RTLOOK	Look around in a 3D scene.
	Look Up and Down	RTUPDOWN	Move up / down and left / right in a 3D scene.
	Walk	RTWALK	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere	RTROT	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	RTROTCTR	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.
	Real-Time Free Sphere	RTROTF	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	RTROTX	Rotates the view dynamically about the screen X-axis.
	Real-Time Y	RTROTY	Rotates the view dynamically about the screen Y-axis.
	Real-Time Z	RTROTZ	Rotates the view dynamically about the screen Z-axis.

Icon	Tool Name	Command	Description
	Set View Point	DDVPOINT	Opens the **Preset Viewpoints** dialog.
	Plan View	PLAN	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	DVIEW	Defines parallel and visual perspective views.
	Save/Restore View	VIEW	Saves and restores named views.
	Viewports	VPORTS	Manages viewports in model space.
	Paper Space Views	MVIEW	Creates viewports in paper space.

(*) <command> <option>: type **zoom** followed by Enter, then type the command option letter, followed by Enter

7.2 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.

The layer utility commands sit on the **Layers** toolbar and on the **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Command	Icon	Description
LAYON		Turns on all layers that have been turned off.
LAYOFF		Turns off layers associated with selected entities.
LAYFRZ		Freezes layers associated with selected entities.
LAYTHW		Thaws all frozen layers.
LAYLCK		Locks layers associated with selected entities.
LAYULK		Unlocks layers associated with selected entities.

Command	Icon	Description
LAYISO		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 layer until the LAYUNISO command is executed.
LAYUNISO		Restores layers isolated by the LAYISO command.

7.3 Hide and isolate entities

Command	Icon	Description
HIDEOBJECTS		Hides the selected entities.
ISOLATEOBJECTS		Displays the selected entities only.
UNISOLATEOBJECTS		Brings back all previously hidden entities.

The tools to launch these commands can be found on:

- the **Hide/Show Entities** and **Access 3D** toolbars.
- the **Select** Quad tab.
- The **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Show, Hide and Isolate entities is also available in the right-click context menu in the **Structure** panel.

Nota: The OBJECTISOLATIONMODE system variable controls whether hiding and isolating entities is saved or not.

7.4 Redrawing and regenerating a drawing

The screen display of a drawing is really a simplified version of the drawing database. From time to time, it is necessary to synchronize the screen display and the drawing database.

The following commands are available:

- The REGEN command (short for "regenerate") regenerates the current viewport.
- The REGENALL command (short for "regenerate all") regenerates all viewports.
- The REGENAUTO command controls when BricsCAD automatically regenerates the drawing.
- The REDRAW command redraws the current viewport to clean it up.
- The REDRAWALL command redraws all viewports to clean them up.
- The REGENMODE system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of REGENMODE through the REGENAUTO command. When REGENMODE = ON, BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might still be necessary.

Take care not to confuse the REGEN command with the REDRAW command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

7.5 Panning

The PAN and RTPAN commands move 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.

7.5.1 Using the pan command

1 Launch the PAN or RTPAN command.

1 The shape of the cursor changes to a hand.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.6 Zooming

The **ZOOM** command visually changes the size of the drawing within the current viewport.

The RT**ZOOM** 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 aN RTZOOM 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.

7.6.1 Real-time zooming

1 Launch the RTZOOM command.

1 The shape of the cursor changes to a magnifying glass.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.

- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.6.2 Zooming to extents

Do one of the following:

- Double-click the middle mouse button (scroll wheel).
- Launch the ZOOM command, then type E and press Enter.

7.6.3 Dynamic zooming

Pans and zooms using a rectangular view box, representing the current viewport. The command first zooms out to display the entire drawing. You can then control the size of the view box dynamically and position the view box on the part of the drawing you want to display in the current viewport.

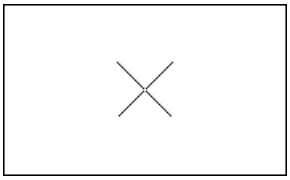
1 Launch the ZOOM command.

2 Choose the **Dynamic** option.

The drawing zooms out to extents.

A dashed rectangle indicates the area of the previous view.

A view box of the same size as the dashed rectangle is attached to the cursor.



3 (option) To resize the view box:

- Click, then move the cursor to resize.



- Click again to stop resizing.

4 Position the view box over the drawing area you want to enlarge, then right-click.

5 The drawing is zoomed in on the selected area.

7.7 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 you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse or Keyboard	Action	Result
Mouse wheel	scroll	Zoom in / out. The incremental change in zoom with each mouse-wheel action is controlled through the **Zoom Factor** variable (ZOOMFACTOR system variable). Default = 60. Values between 3 and 100 are accepted. When zooming in, the incremental step decreases gradually, allowing you to focus on a particular detail easily. In a 3D model, the entity under the cursor has the zoom focus. The ZOOMWHEEL 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 mouse button or wheel*	press and hold while moving the mouse	Real-time pan (RTPAN command)
Middle mouse button or wheel*	double-click	Zoom extents
Ctrl + Shift + Left mouse button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM command)
Ctrl + Shift + Right mouse button	press and hold while moving the mouse	Constrained real-time pan (RTPAN 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 command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ command)

Mouse or Keyboard	Action	Result
Walk through navigation		
Alt + Left Button	press and hold while moving the mouse	Walk left / right and forward / backward (RTWALK command)
Alt + Middle Button	press and hold while moving the mouse	Move up / down and left / right (RTUPDOWN command)
Ctrl + Middle Button	press and hold while moving the mouse	Look around (RTLOOK command) RTWALK command
Ctrl + Arrow Keys	press the arrow keys while holding the Ctrl key	Walk left / right and forward / backward ()
Ctrl + Shift + Arrow Keys	press the arrow keys while holding the Alt + Ctrl keys	Look up / down and left / right (RTUPDOWN 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) system variable is set **ON**.

7.8 View rotation

The following command are available for view rotation operations:

- 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 restores the plan (top) view.
- The **VPOINT** command changes the 3D viewpoint.

Nota:

- If the **CONTINUOUSMOTION system** variable is set, the view rotation continues until you press the Esc key to conclude the command.
- View rotation commands should not be used when working in 2D. Use the PLAN command or the LookFrom control to restore top view.

7.8.1 Rotating a view freely

1 Launch the RTROT, RTROTF, RTROTX, RTROTY or RTROTZ command.

1 The shape of the cursor changes according to the selected command.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.8.2 Using Preset Viewpoints

Nota: Switch off Perspective mode before starting the command.

1 Launch the DDVPOINT command.

1 The **Set Viewpoint** dialog box appears.

2 Click the **Angle Precision** button to set the display mode for the **Set Viewpoint** dialog box.

The options are: **45°**, **15°** or **5°**.

The **Horizontal** and **Vertical** select panes are divided accordingly.

3 Click an angle in the **Horizontal** and **Vertical** select panes.

The view orientation is updated.

4 (option) Click the **Plan** button to restore the plan (top) view.

5 (option) Repeatedly click the **Previous** button to restore previous viewpoints.

6 To conclude the command:
   - Click the **OK** button to keep the current view.
   - Click the **Cancel** button to restore the original view.

7.9 The LookFrom control

The **LookFrom** control allows you to select a number of preset views such as orthographic views and isometric views. By default, the **LookFrom** control is displayed in the top right corner of the graphic screen.

The NAVVCUBEDISPLAY system variable sets the display of the LookFrom control.

7.9.1 Context menu options

When you right-click the LookFrom control, a context menu appears. The options are:

- **UCS**: sets the current coordinate system.
  - **World**: restores the WCS (World Coordinate System)
  - **New**: runs the UCS command.
  - **UCS Manager...**: runs the EXPUCS command; displays the **Drawing Explorer / Coordinate Systems** dialog box.
- **View**:
  - **New**: runs the -VIEW command.
  - **View Manager...**: runs the VIEW command; displays the **Drawing Explorer / Views** dialog box.
- **Visual Style**: Allows you to set the visual style.
- **Isometric Mode**: Sets the isometric mode.
- **Twist Mode**: Sets the Twist mode.
- **WCS**: LookFrom uses the WCS as reference coordinate system.
- **UCS**: LookFrom uses the current coordinate system (WCS or UCS) as reference coordinate system.
- **Perspective**: Sets the perspective view mode.
- **Location**: Sets the location of the LookFrom control.
- **Dismiss**: Closes the LookFrom control.

  Edit the NAVVCUBEDISPLAY system variable to restore the LookFrom control.

7.9.2 LookFrom control settings

The behavior of the **LookFrom** control depends on a number of system variables and user preferences.

To access the LookFrom settings:

1 Right-click the LookFrom control and choose **LookFrom Settings...** in the context menu.

2 Run the LOOKFROM command and choose the **Settings** option.

7.9.3 Setting the LookFrom mode

The LookFrom control runs in one of two modes:

- Isometric mode for working in 3D models.
- Twist mode for 2D drafting.

To set the LookFrom mode:

1 Right-click the LookFrom control.

2 Select **Isometric Mode** or **Twist Mode** in the context menu.

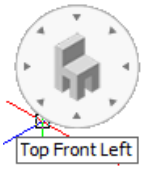
7.9.4 Using the LookFrom control in Isometric mode

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

2 Hover the cursor over a view orientation.

3 The chair in the center of the LookFrom control rotates according to the view orientation.



4 (option) Hold down the Ctrl key to look bottom-up.

5 Click to confirm.

6 The view is rotated.

7.9.5 Using the LookFrom control in Twist mode

In **Twist** mode, the **LookFrom** control rotates the view around the screen Z-axis.

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

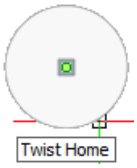
2 Move the cursor to the edge of the **LookFrom** control, on the left half for counterclockwise, on the right half for clockwise rotation. 15°, 30°,45°, 60°, 90°, 120°, 135° and 180° angles are available.



3 Click to rotate the view.

4 (option) Without moving the mouse, click again to rotate over the same angle again.

5 (option) Move the cursor to the center of the **LookFrom** control, then click to restore an unrotated view.



7.9.6 View transition options

The following system variables control the behavior of view transitions:

- 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.

7.10 Layer states

Layer states let you save and restore configurations of layer properties. You might want a layer to be displayed 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 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 commands might be useful.

The LAYERSTATE command opens the **Drawing Explorer / Layer States** dialog box.

7.10.1 Saving the current layer state

1 Click the **New** button on the **Drawing Explorer / Layer States** dialog box.

A new layer state is created.

2 Type a name for the layer state, replacing the **NewLayerState1** default name.

3 (option) When in **Detail View**, type a description in the **Description** field.

4 Select the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

5 Select a **Restore option**.

6 When in paper space, tick the **Apply properties a viewport overrides** option.

Nota: In a new layer state, all layer properties are selected to be restored by default.

7.10.2 Editing layer states

1 (option) Click the **Overwrite** button to overwrite the layer state with the current layer settings.

2 (option) Rename the layer state.

3 (option) Edit the layer state description.

4 (option) Edit the **Layer Properties to restore**.

Click the tick box to select a layer property, click again to remove a property.

7.10.3 Restoring layer states

1 Do one of the following:
   - Click the **Restore** button on the **Drawing Explorer / Layer States** dialog box.
   - Right-click, then choose **Restore** in the context menu.

2 Click the **Regen** button, unless the **Regen Auto** button is pressed.

Nota:

- The restore option of the layer state applies to layers which are added after the selected layer state was created. The options are:
  - Turn Off.
  - Freeze
  - Do nothing.
- Layer states can be restored using the **Layer State** control, which you can add to a ribbon panel or a toolbar.

7.10.4 Copying layer states between open drawings

1 Open both, the source and the target drawing.

2 Select the source drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

3 Select the layer state(s) you want to copy.

4 Press and hold the Ctrl key to select multiple layer states.

5 Hold down the left mouse button, then drag-and-drop the selection onto the target drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

The selected layer states are copied to the target drawing.

7.10.5 Exporting layer states

1 Select the layer state on the **Drawing Explorer / Layer States** dialog box.

2 Click the **Export** button.

The **Export Layer States** dialog box appears.

3 In the **File Name** field, type a name for the layer state file.

4 Select a destination folder.

5 Click the **Save** button.

7.10.6 Importing layer states

1 Click the **Import** button on the **Drawing Explorer / Layer States** dialog box.

The **Import Layer States** dialog box appears.

2 Select the layer state file to import.

3 Click the **Open** button.

The layer state is imported.

7.11 Named views

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.

7.11.1 Exploring views

1 Launch the VIEW command.

1 The **Drawing Explorer / Views** dialog box appears.

2 (option) Click the blank tile in front of the **View Name** to restore the view.

The current view is marked.

3 (option) Click the **New** button in the **Drawing Explorer** toolbar to save the current view (launches the -VIEW command).

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.

7 (option) Click the **Background** column of a view to choose a background. The background definition of the view is applied during rendering.

7.11.2 Saving a view

1 Do one of the following:

- Launch the -VIEW command.
- Click the **New** button in the **Drawing Explorer / Views** toolbar.

2 Choose the **Save** option.

-VIEW command workflow:

- **You are prompted**: Save view as:
- Type a name for the view in the Command line, then press Enter.

Drawing Explorer workflow:

- The **Drawing Explorer / Views** dialog box appears.
- Type a name to replace the NewView1 default name.

Nota: If you type a name in the Command line that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

7.11.3 Restoring a view

In the **Drawing Explorer / Views** dialog box, click the **Current** field of the view.

7.11.4 Editing the view background

1 Select the view in the **Drawing Explorer - Views** dialog box.

2 Select the **Background** property.

3 Click the **Browse** button at the right hand side of the settings field.

The **Background** dialog box appears.

4 Click the **Type** list button, then select the background type.

The options are **None**, **Solid**, **Gradient** and **Image**.

5 Select a background type.

6 The settings for the selected type are displayed in the **Background** dialog box.

7 Specify the background settings.

8 Click the **OK** button on the **Background** dialog box.

9 (option) Click the **Current** field.

Nota: The GRADIENTMODE system variable sets the default for gradient backgrounds: 2 or 3 colors.

7.12 Visual styles

A visual style is a collection of settings that control the display of 3D solids and surfaces.

The VISUALSTYLES command opens **Drawing Explorer – Visual Styles**.

Here you can:

- Edit existing visual styles.
- Create new visual styles.
- Delete visual styles.
- Copy and paste visual styles between open drawings.

7.12.1 Setting the current visual style

The current visual style can be set in the **View / Visuals** style field in the **Properties** panel and in the right-click context menu of the **LookFrom** widget.

Click the desired visual style in the list.

The default options are:

- 2D Wireframe
- Conceptual
- Hidden
- Modeling
- Realistic
- Shaded
- Shaded with edges
- Shades of gray
- Sketchy
- Wireframe
- X-Ray

- BIM (BIM templates only)
- Mechanical (Mechanical templates only)

7.13 Model space and paper 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 VPORTS 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 the entities in model space. Layout viewports can be placed anywhere in a paper space layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The MVIEW 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.

7.13.1 Switching between model space and paper space

- To switch between Model Space and Paper Space layouts, use the **Model** and **Layout** tabs at the bottom of the graphic window.
- When working in a layout, you can access model space through a paperspace viewport:
  - Double-click inside a viewport to access model space. Double-click outside a viewport to go back to paper space.
  - Click the **Paper/Model** field in the **Status** bar to toggle between the workspaces. This method has to be used when it is not possible to click outside the viewport.
  - Use the **MSPACE** and **PSPACE** commands. These commands are available in the right-click context menu of the **Paper/Model** field in the **Status** bar.

7.13.2 Using the layout tabs

- The name of the active tab is displayed in bold.
- Click the navigation buttons at the left hand side to switch.
- Drag layout tabs to change their order.
- Click the button between the navigation buttons and the **Model** tab to open the **Layout Manager** dialog box.
- Right-click a layout tab to display a context menu.
  - **New**: Create a new layout.
  - **From Template**: Imports a layout from another drawing.
  - **Rename**: Renames the layout.
  - **Copy**: Inserts a copy of the layout.
  - **Move Right / Move Left**: Moves the layout.
  - **Export...**: Exports the layout to a new drawing.

7.14 Model space viewports

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 coordinate system (WCS or UCS), grid display and snap can be set per viewport.

The VPORTS command creates and manages model space viewports.

7.14.1 Working with multiple viewports

- Click in a viewport to make it current.
- The border of the current viewport is highlighted.
- The cursor is shown in the current viewport only.
- You can start a command in one viewport, and finish the operation in a different viewport.

7.14.2 Joining adjacent viewports

You can join viewports which share an edge of equal length.

1 Launch the **VPORTS**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.

4 The border of the selected viewport is highlighted.

You are prompted: Select inside viewport to join.

5 Click the viewport to join again to confirm.

6 The two selected viewports are joined.

7.15 Paper space viewports

The **ALIGNSPACE** command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

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.

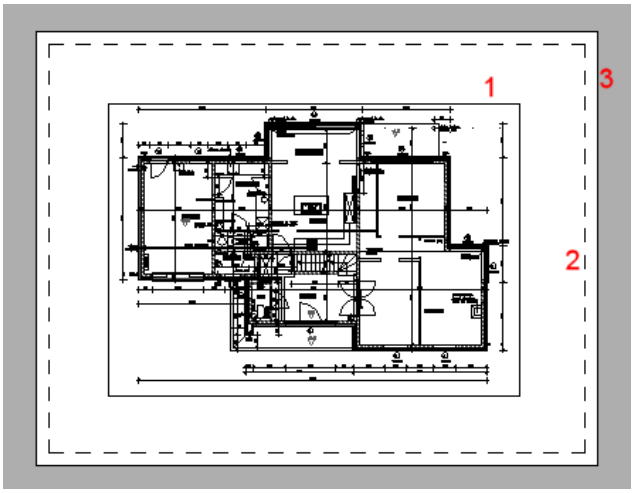
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space in layout, click any layout viewport to make it the current viewport, 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.

7.15.1 Displaying the paper sheet and the printable area



1 Viewport.

2 Printable area.

3 The display of the printable area is controlled through the DISPPAPERMARGINS system variable.

4 Paper sheet.

5 The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

7.16 Creating paper space viewports

The MVIEW and VPORT commands create one or more viewports in layout tabs.

- A new viewport shows all entities on the layers which are currently visible in model space.
- It is recommended to use a dedicated layer for viewport entities. If you don't want the viewport outlines to be printed, set the **Do Not Plot** property of the viewport layer.

7.16.1 Creating an array of layout viewports

The MVSETUP command (short for "multiple viewports setup") creates multiple paper space viewports, and then aligns, rotates and scales them.

7.16.2 Defining the viewport properties

1 Select the viewport.

1 The viewport properties appear in the **Properties** panel.

2 Click the settings field of a property to modify.

Name	Description
Center	XYZ coordinates of the center point of the viewport with respect to the bottom left corner of the printable area of the layout.
Height (*)	Height of the viewport in **drawing units**.
Width (*)	Width of the viewport in **drawing units**.
On	Controls the display of the content of the viewport.
Clipped	Indicates whether the viewport is clipped or not.
Display locked	Locks the scaling of the viewport content to preserve the scaling factor. The VPLOCK field in the **Status** bar displays the Display Locked status of the selected viewport(s). Click the field to toggle the **Display Locked** property of the selection.

Name	Description
Scale	The SCALELISTEDIT command allows you to edit the available scales in the current drawing. **Annotation scale**: Sets the annotation scale for the viewport. Editing the Annotation scale automatically sets the Standard scale. If the ANNOALLVISIBLE system variable is OFF, annotative entities which do not support the annotation scale are not displayed. **Standard scale**: By default, the Standard scale matches the Annotation scale, but you can adjust it manually. 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 you to set the viewport scale to a scale which is not available in the drawings scale list.
UCS per viewport	If **Yes**, lets you define a UCS for this viewport.
Visual Style	Sets the visual style for the viewport. This option only applies to 3D models.
Shade Plot	Allows you to adjust the plotted shading from the visual style. Default is '**As Displayed**'.
Default Lighting	Allows you to switch off the lights in the model and use the default lighting.

7.17 Editing paper space viewports

7.17.1 Setting the layer properties in a paper space viewport

The layer visibility can be controlled per paper space viewport. When working in a paper space layout, a number of VP* (Viewport) properties are available, which affect the current viewport only and can be adjusted from the corresponding default properties:

- VP Freeze
- VP Color
- VP Linetype
- VP Lineweight
- VP Transparency
- VP Plot

1 Double-click inside the paper space viewport.

2 Launch the EXPLAYERS command.

3 The **Drawing Explorer / Layers** dialog box appears.

4 (option) Click the icon in the **VP Freeze** column to toggle the display of a layer.

5 (option) Click the icon in the **VP Color** column, then select a color in the **Select Color** dialog box.

6 (option) Click the **VP Linetype** column, then select a linetype in the drop-down list.

7 (option) Click the **VP Lineweight** column, then select a lineweight in the drop-down list.

8 (option) Click the **VP Transparency** column, then select a transparency value in the drop-down list.

9 (option) Click the **VP Plot** column, then select a plot style in the **Select Plot Style** dialog box. This option applies to STB (Style Tables) drawings only.

Nota:

- When the **Indicate layers in use** option in the **View** menu of the **Drawing Explorer** dialog box is set, the icon in the **Current** column indicates whether VP property overrides exist.
- Use the **Viewports / Vpfreeze** option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

7.17.2 Clipping a viewport

The **VPCLIP** command (short for "viewport clipping") clips viewports in layouts.

Only circles and closed polylines can be used to create a clipped viewport. Therefore, use the **Create polyline representations of an ellipse** option of the PELLIPS system variable when creating an elliptical viewport.

7.17.3 Removing the clipping boundary of a viewport

The **Delete** option of the VPCLIP command removes the clipping boundary.

7.17.4 Rotating the display in a paper space viewport

The display of a paper space viewport can be rotated only if the display of the viewport is not locked.

The **Align** > **Rotate** option of the MVSETUP command defines the angle of the content of a viewport relative to the X-axis of the world coordinate system (WCS).

If the VPROTATEASSOC system variable is ON, the ROTATE and MANIPULATE commands rotate the content of a viewport over a specified angle. The viewport is clipped by a rectangle of the dimensions of the original viewport. Use the **Delete** option of the VPCLIP command to remove the clipping.

In order to undo the rotation:

- Set the rotation angle = 0 (zero) in the **Align** > **Rotate** option of the MVSETUP command.
- Double-click inside the viewport, then launch the PLAN command and press Enter twice (= set plan view of the WCS).

7.17.5 Aligning viewports

1 Launch the MVSETUP command.

2 Choose **Align** in the prompt menu.

You are prompted: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3 (option) Choose the **Horizontal** alignment option.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved vertically to align the two points horizontally.

4 (option) Choose the **Vertical alignment option**.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved horizontally to align the two points vertically.

7.17.6 Aligning model space and paper space points

The ALIGNSPACE command adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space.

Aligning 1 point

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Right-click or press Enter.

5 **You are prompted**: Alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Activate the viewport which will be used for alignment.

8 Click inside the viewport you want to align.

9 Right-click or press Enter to confirm.

10 The display in the selected viewport is moved to make the two points coincident.

Aligning 2 points

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Pick a second point in the layout viewport.

5 **You are prompted**: First alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Second alignment point in Paper space:

8 Pick a second point in the layout.

9 **Prompts you:** Activate the viewport which will be used for alignment.

10 Click inside the viewport you want to align.

11 Right-click or press Enter to confirm.

12 The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

7.18 Layouts

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 paper size, 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 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 you 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 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.

7.18.1 Creating a new layout

1 Launch the LAYOUT command, then select the **New** option.

2 **You are prompted**: Enter a layout name. <Layout1>:

3 Type a unique name for the layout, then press Enter.

4 The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

5 Right-click or press Enter to accept the default name (e.g. Layout1).

6 The new layout tab is added.

Nota: Right-click a layout tab, then choose **New** in the context menu or click the plus (+) button after the last layout tab to add a new layout using a default name. See step 3 in the above procedure to rename the layout.

7.18.2 Copying layouts

1 Right-click the layout tab you want to copy.

1 A context menu appears.

2 Choose **Copy** in the context menu.

You are prompted: Enter new layout name:

3 Type a new unique name in the Command line, then press Enter.

4 A copy of the selected layout is added.

7.18.3 Arranging the layout tabs

Do one of the following:

- Drag the layout tab to the desired position.
- Right-click the layout tab, then choose a **Move** option in the context menu.

7.18.4 Importing layouts

1 Right-click the **Model** tab or one of the layout tabs.

A context menu appears.

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.

6 Hold down the Ctrl key to select multiple layouts.

7 Click the **OK** button.

The selected layouts are imported.

7.18.5 Renaming a layout

1 Double-click the layout tab you want to rename.

1 The **Rename Layout** dialog box opens.

2 Type a new unique name in the **Name** field.

3 Click the **OK** button.

7.18.6 Deleting a layout

1 Right-click the layout tab you want to delete.

1 A context menu appears.

2 Choose **Delete** in the context menu.

3 An **Alert** box appears.

4 Click the **OK** button to confirm.

7.18.7 Using the Layout Manager

1 Do one of the following:

- Launch the LAYOUTMANAGER command.
- Click the button between the navigation buttons and the **Model** tab.

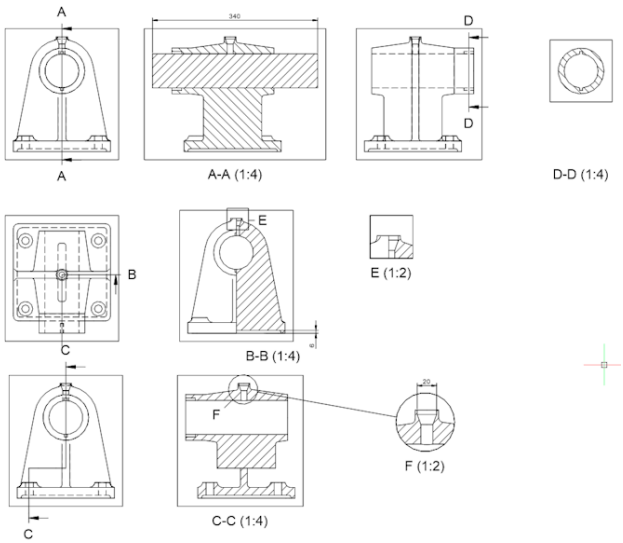
2 In the **Layout Manager** dialog box you can:

- Create layouts.
- Copy layouts.
- Delete layouts.
- Change the order of the layout tabs.
- Publish layouts.

7.19 Generated drawing views

The Generated Drawing Views functionality allows you to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

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 command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the VIEWUPDATE command.



The VIEWBASE command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The VIEWSECTION command creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.

The VIEWSECTIONSTYLE command allows you 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 you 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 = 0.

The VIEWEXPORT command exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. This command can be used in Paper Space only.

The VIEWEDIT command allows you to change the scale and the hidden line visibility of drawing views.

Tools to generate drawing views of your 3D solid model are available on:

- The **Drawing Views** toolbar, which is a flyout of the **View** toolbar.
- The **Section Types** and **Detail Section Types** toolbars.
- The **View** | **Drawing Views** menu.
- The **Drawing Views** panel of the **Annotate Ribbon** tab (Mechanical Workspace).

7.19.1 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.

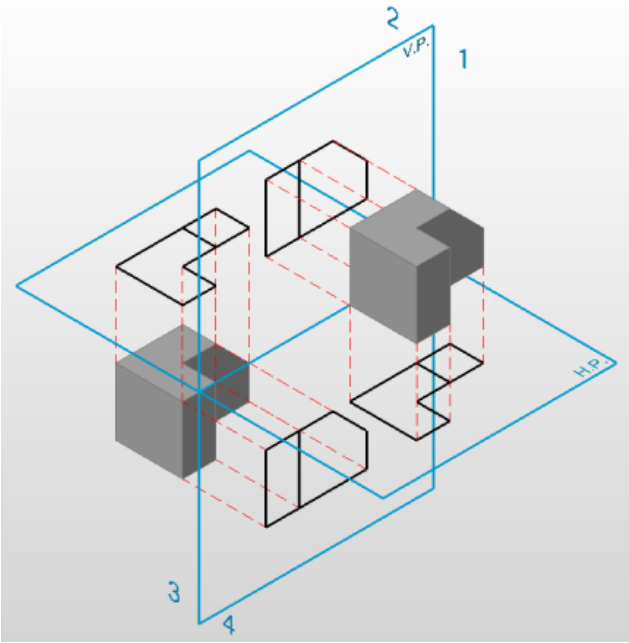
To generate the standard views of your 3D solid model, run the ViewBase command from Model Space.

You can choose between first angle (European) projection or third angle (American) projection.

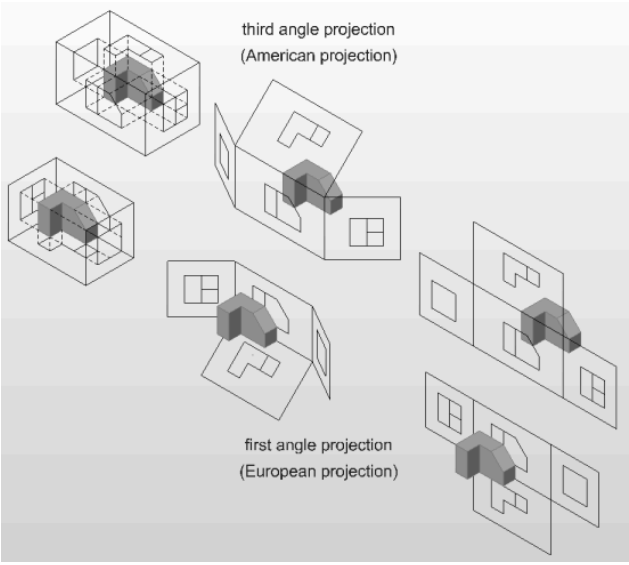
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the 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 3D 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).



With multiview orthographic projections, up to six views 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 the 6 faces of a box around the model. When unfolding the box, all views are organized in a 2D drawing.



The projection type is set by the **Projection Type** option of the VIEWBASE command or by editing the PROJECTIONTYPE system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE 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.

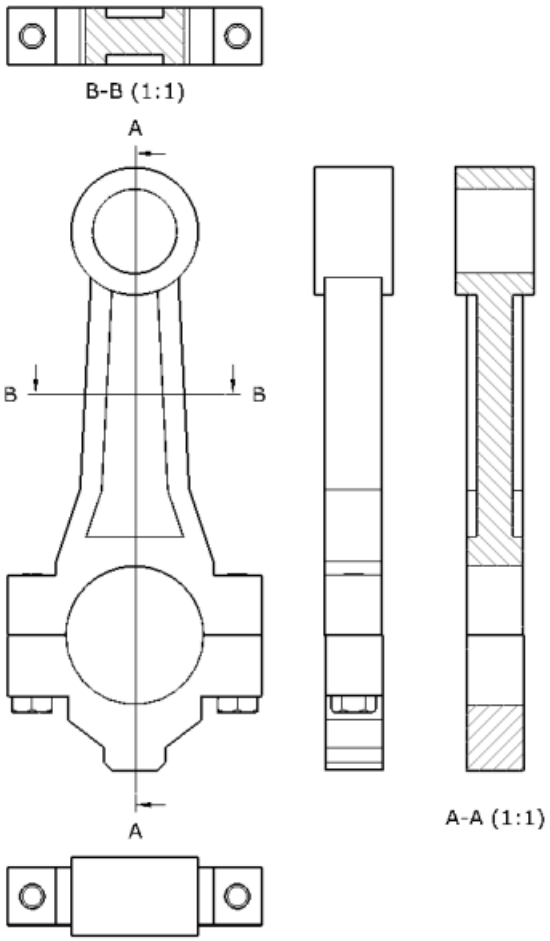
The display of the BM_Isometric_Hidden layer is turned off by default.

Additional projected views can be added using the VIEWPROJ command. A projected view inherits the scale, display settings and alignment from the parent view.

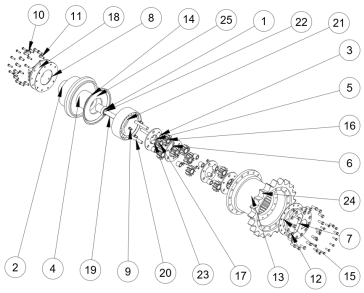
7.19.2 Generating section views

A 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 VIEWSECTION command from the paper space layout where the standard drawing views are located.



7.19.3 Special views



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

It is possible to generate drawing views for a selected exploded representation. Normally, an orthogonal projection is used for that; however, BricsCAD allows you 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 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, so annotations and balloons can be placed. To do that, the views should be created or updated with GENERATEASSOCVIEWS = ON.

7.19.4 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 containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- running the VIEWUPDATE command.
- running the BMUPDATE command.
- 3D geometry.

Nota:

- Hidden line removal calculation might take some time for complex 3D models. In such cases, it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.
- If GENERATEASSOCVIEWS = ON, associative dimensions will be updated according to the modified 3D geometry.

7.19.5 Exporting views to model space

The associativity mechanism of drawing views in a paper space layout allows for 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 command allows you to move or copy drawing views to

model space 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.

7.19.6 Customizing drawing views

The first time the VIEWBASE command is launched, four new layers are created:

- BM_Ortho_Hidden: hidden lines in drawing views;
- BM_Ortho_Visible: visible lines in drawing views;
- BM_Tangent_Hidden: hidden tangent lines in drawing views;
- BM_Tangent_Visible: visible tangent lines in drawing views.

Use the VIEWEDIT command to change the visibility of these layers for all views or for a selection of views only.

- Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.
- Use ViewSectionStyle command to adjust different properties of section views such as style, color, and height of identifier text;
- start and end symbols, their colors, size and direction of arrows scale, patter, color, and angle of hatches;
- color, type, and weight of a line representing a cutting plane.

Use the VIEWDETAILSTYLE command to adjust different properties of details views such as:

- style, color, height of identified text, and the type of identifier
- symbol, and its color and size
- color, type, and weight of detail view boundary

8. Desenho 2D

8.1 Criar entidades.

Em um desenho, você pode criar uma variedade de tipos de entidades diferentes. Entidades de desenho podem ser muito simples, como linhas, círculos, arcos, pontos e linhas semi-infinitas, ou ser complexas, como polilinhas, splines e planos. Além de entidades 2D, você também pode criar entidades de superfície 3D ou sólidos 3D no BricsCAD.

Para expandir a qualidade do seu desenho, BricsCAD fornece uma série de ferramentas para adicionar texto, hachura, dimensões.

8.1.1 Procedimento geral para criar novas entidades

Novas entidades são criadas utilizando as configurações atuais de criação de entidades, na camada atual, no plano-XY do sistema de coordenadas atual, por padrão, este é o Sistema de Coordenadas Mundo (WCS).

1 (opção) Definir a camada atual.

2 (opção) Ajuste as propriedades de criação da entidade na seção **Geral** do painel **Propriedades**.

3 Inicie o comando.

4 Execute o comando de acordo com os prompts de comando.

8.2 Configurar criação de Entidades.

As configurações e preferências padrão para criar entidades de desenho são especificadas pelas variáveis do sistema.

Você pode visualizar e modificar essas variáveis:

- Na caixa de diálogo Configurações. Expanda **Desenho** > **Criação da entidade**. No topo da lista, as propriedades gerais como cor, tipo de linha, e camada, são exibidas. Abaixo dessas, as configurações para propriedades específicas da entidade são exibidas.
- No nó **Geral** do painel Propriedades quando nada estiver selecionado.

As configurações em **Geral** são:

- **Cor**: Define a cor da entidade. Padrão = PorCamada.
- **Camada**: Define a camada atual.
- **TipoLinha**: Define o tipo de linha atual. Padrão = PorCamada.
- **Escala de tipo de linha**: Define a escala de tipo de linha atual. Padrão = 1.
- **EspLinha**: Define a espessura da linha atual. Padrão = PorCamada.
- **Transparência**: Define a transparência atual. Padrão = PorCamada.
- **Elevação**: Especifica a altura (valor Z) acima do plano-XY do sistema de coordenadas atual (WCS ou UCS). Padrão = 0.

Nota: A maioria das configurações de criação de entidades são salvas no desenho, então elas só se aplicam ao desenho atual. Se você quiser ajustar essas configurações para todos os seus desenhos

futuros, edite seu(s) desenho de template padrão, e ajuste as configurações lá. Os desenhos de template são melhor salvos na pasta salva através da configuração TEMPLATEPATH. A variável de sistema BASEFILE especifica o caminho e o nome do arquivo de template padrão.

8.3 Trabalhar com camadas

Você pode criar e gerenciar camadas na seção **Camadas** do Explorer do Desenho (comando EXPCAMADAS) ou no painel Camadas (comando ABRIRPAINELCAMADAS).

O Explorer do Desenho não pode ser aberto enquanto você usa outros comandos, enquanto o painel Camadas pode ser aberto. Você pode movê-lo e redimensioná-lo conforme desejado, ou até mesmo levá-lo para um segundo monitor.

8.3.1 Camada atual

Novas entidades são desenhadas na camada Atual. Para desenhar novas entidades em uma camada diferente, você deve primeiro tornar essa camada a camada Atual.

- A camada atual é salva através da variável de sistema CLAYER.
- A variável de sistema CENTERLAYER* controla a camada em que as linhas de centro e as marcas de centro, criadas pelo comando COTACENTRO são colocadas.
- A variável de sistema DIMLAYER* controla a camada para as entidades de dimensão criadas pelos comandos de dimensionamento.
- Entidades criadas pelo comando INTERFERE são criadas na camada especificada pela variável de sistema INTERFERELAYER.

* O valor padrão é <Usar Atual>

Para definir a camada atual, siga um destes procedimentos:

- Não tendo nenhuma entidade selecionada, selecione **Camada** no painel Propriedades, e escolha a nova camada atual na lista suspensa.
- Na barra **Propriedades da Entidade**, clique na seta para baixo ao lado do campo **Camada**, e escolha uma camada na lista suspensa.
- No painel **Camadas** na Faixa de opções, clique a seta para baixo e escolha uma camada na lista suspensa.
- Clique o botão-direito no campo onde aparece o **nome da Camada** na barra de Status, e selecione uma camada no menu de contexto.

Nota:

- É possível desligar a exibição da camada atual (💡). Como resultado, tudo o que você desenhar ficará invisível até que a exibição da camada atual seja ativada (💡) novamente. Um aviso será exibido se você tentar desligar a camada atual.
- Não é possível o congelamento (❄) da camada atual.

8.3.2 Estados das camadas

ESTADOCAMADA comando.

Estados de camada são conteúdo nomeado, armazenado em um desenho.

Os Estados da camada permitem salvar e restaurar configurações de propriedades e os estados das camadas. Você pode querer que uma camada seja exibida em azul às vezes e verde em outras vezes, ou você precisa que algumas camadas estejam desligadas, ou congeladas ou bloqueadas ao editar uma parte específica de um desenho. Depois de passar muito tempo ajustando todas as configurações da camada, você pode usar os Estados de camada para salvar as configurações, e posteriormente então retornar a estes. Você pode salvar múltiplos Estados da camada em cada desenho. Você pode copiar Estados de camada entre desenhos e salvar um estado de camada em um arquivo externo, que pode ser importado em outro desenho.

8.4 Entity linetype

You can 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 can also contain an unlimited number of named 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 source file is used by default 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 set by the CELTYPE system variable.

The options to set the Linetype are:

- ByLayer: Entities adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.
- ByBlock: Entities drawn as continuous lines until you include them into a block. The entities then inherit the linetype setting of the block entity when you insert the block into a drawing.
- Named linetype: Continuous or one of the linetypes saved in the drawing.

Linetype scale

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.

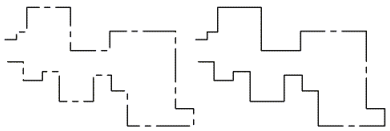
In the linetype definition the length of the lines and gaps are either defined in inch (default.lin) or mm (iso.lin). If the drawing units are different from inch or mm, linetypes need to be scaled. The linetype scaling is controlled by the LTSCALE system variable, which must be set with respect to the 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

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. ThePLINEGEN 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.



The same polyline drawn with PLINEGEN on (left) and off (right).

Paper space linetype scale

The PSLTSCALE system variable controls the linetype scaling in paperspace layouts.

If ON, linetype scaling is independent from the viewport scale.

In general, PSLTSCALE is best set ON. Notice that this variable can be set per layout.

Model space linetype scale

The MSLTSCALE system variable controls the linetype scaling in model space.

If ON, thecurrent annotation scale(CANNOSCALE system variable) is applied to linetypes in model space.

MSLTSCALE is best set OFF. It might be useful to switch it ON when printing from model space.

8.4.1 Setting the current linetype

Do one of the following:

- On theProperties panel, select**Linetype**.

Make sure that no entity is selected.

Choose a linetype in the drop down list.

- 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 theStatus Bar, then choose a linetype in the context menu.

8.4.2 Adding a new linetype

1 Do one of the following:

- Launch the LINETYPE command, then choose the **New** option.
- Select **Linetype** in the Properties panel, then click the down arrow and choose **Load...**

The Load linetypes dialog box displays.

2 (option) Click the **File...** button to load a different linetype source file.

3 Select one or more linetype(s) in the Load linetypes dialog box.

Hold down the Ctrl-key to add or remove linetypes.

4 Click the **OK** button

8.4.3 Setting the entity linetype scale

1 On theProperties panel, select**Linetype scale**.

Make sure no entity is selected.

2 Type a new value in the Linetype scale field.

The value of the Linetype Scale is saved through the CELTSCALE system variable. This variable is best set to 1, except for special linetypes, such as BATTING or GAS_LINE, which most likely need to be scaled.

Nota: The Entity Linetype Scale applies to new entities only.

8.5 Cor da entidade

A cor determina como uma entidade é exibida na tela e como é impressa. As entidades são criadas na cor atual, conforme definido pela variável de sistema CECOLOR.

Você pode escolher entre 255 **cores de índice** ou definir uma **true color**.

8.5.1 Cor Indexada

Cor de Indice é a especificação da cor de um pixel em uma tela usando um valor de 8-bit que permite até 256 cores possíveis.

Cada uma das Cores do Índice tem um número único de 1 a 255. Sete das cores do índice também podem ser referidas pelo nome: vermelho (1), amarelo (2), verde (3), ciano (4), azul (5), magenta (6) e branco/preto (7). A cor de índice 7 exibe branco em um fundo de tela preto e preto em um fundo de tela branca. A cor índice 7 sempre imprime em preto.

As duas propriedades de cores adicionais são PorCamada e PorBloco. Essas propriedades de cor fazem com que uma entidade adote a cor da camada ou do bloco no qual ela é membro. PorCamada é a cor

número 256 e PorBloco é a cor número 0. Em todos os comandos em que você usaria uma cor, você pode indicar PorCamada e PorBloco, bem como pelos números 256 e 0, respectivamente.

As opções para definir a Cor são:

- PorCamada: Entidades adotam a cor da camada. Isso permite que você altere a cor de todas as entidades ajustando a cor da camada onde se encontram.
- PorBloco: Entidades são desenhadas na cor de índice 7 (preto ou branco, dependendo da cor de fundo da tela). Quando incluídas em uma definição de bloco, essas entidades adotam a cor do bloco quando você insere o bloco em um desenho.
- Cor Indexada
- True Color

.

Nota: Se as entidades que usam uma cor de índice serão impressas nessa cor, depende da **Tabela de Cor** (CTB) ou da **Tabela de Estilo** (STB) usada para impressão. Somente se a configuração **Cor** no arquivo de definição CTB ou STB estiver definida como **Usar Cor do Objeto**, a cor impressa corresponderá à cor da entidade.

8.5.2 True colors

True Color é a especificação da cor de um pixel em uma tela usando um valor de 24-bits, que permite até 16.777.216 cores possíveis.

O número de bits usado para definir a tonalidade de cor de um pixel é sua profundidade de bits. A cor verdadeira também é conhecida como cor de 24-bit. Alguns novos sistemas de exibição de cores oferecem um modo de cores de 32-bits. O byte extra, chamado canal alfa, é usado para controle e informações de efeitos especiais.

True colors usam uma definição de cor **RGB** (**R**ed (vermelho), **G**reen (verde), **B**lue (azul). Cada um desses parâmetros tem um intervalo de 0 a 255. A definição RGB para preto é (0,0,0), a definição RGB para branco é (255,255,255).

8.5.3 Configurar a cor atual da entidade

1 Siga um dos procedimentos:

- Inicie o comando COR.
- Clique no botão da ferramenta **Selecionar Cor** ( ) na barra Configurações.
- No painel Propriedades, selecione **Cor**, então escolha **Selecionar Cor** na lista suspensa.
- Clique duas vezes no **Campo de Cor** na barra de Status.

A caixa de diálogo Selecionar cor é exibida.

2 (opção) Clique na aba **Cor de Índice** e siga um destes procedimentos:

- Clique em um dos blocos de cor.
- Digite o número da cor no campo **Número da Cor**.

3 (opção) Clique na aba **True Color** e siga um destes procedimentos:

- Clique no painel **Seleção de Cor**.
- Defina os parâmetros da cor nos campos **Parâmetro da Cor**.

4 Clique no botão **OK** para confirmar.

8.6 EspLinha

Espessuras de linha determinam como entidades grossas ou finas aparecem na tela e / ou quando são impressas.

A espessura da linha pode ser definida PorCamada, PorBloco, Padrão ou para um valor específico.

As espessuras de linha são definidas em milímetros ou polegadas, dependendo da variável de sistema LWUNITS.

Você não pode atribuir as espessuras de linha para planos, pontos, fontes TrueType e imagens raster.

As opções para definir a Espessura da linha são:

- PorCamada: A espessura da linha é determinada pela propriedade Espessura da linha da camada atual. Alterar a propriedade Espessura da linha de uma camada afetará todas as entidades que tiverem sua espessura de linha definida por Camada nessa camada.
- PorBloco: Entidades criadas usando a espessura de linha PorBloco, recebem a espessura de linha Padrão até serem incluídas em uma definição de bloco. As entidades então herdam a espessura da linha do bloco quando você insere o bloco em um desenho.
- Padrão: A espessura de linha padrão é salva por meio da variável de sistema LWDEFAULT. Alterar a espessura de linha Padrão afeta todas as entidades em todo o desenho que têm a espessura da linha definida como Padrão.

8.6.1 Variáveis de sistema

A variável de sistema CELWEIGHT define a espessura da linha para novas entidades.

A variável de sistema LWDEFAULT define o valor da espessura de linha Padrão.

A variável do sistema LWDISPLAY controla a exibição de espessuras de linha. Clique no campo LWT na barra Status para ativar/desativar a exibição da espessura da linha.

A variável de sistema LWDISPLAYSCALE controla a escala de exibição das espessuras de linha na aba Modelo. Valores entre 0 e 1 são aceitos.

8.6.2 Espessuras de linha na impressão

Entidades que utilizam uma cor de índice utilizarão espessura de linhas somente quando impressas se a **Tabela de Cores** (CTB) ou **Estilo de Plotagem** (STB) estiver definida como **Utilizar espessura de linha do objeto**. Caso contrário, a espessura da linha é definida pela cor da entidade ou pelo estilo de plotagem.

Entidades que usam true color sempre usam a espessura da linha de objeto quando impressas.

8.7 Fill mode

TheFILLcommand toggles the display of filled and hatched areas and specifies whether wide polylines, traces, solids, hatches (including solid-fill) and wide polylines are filled in. It toggles the value of the FILLMODE system variable.

When FILLMODEis Off, all filled entities display and print as outlines and reduces the time it takes to display or print a drawing.

	Polyline	Donut	Trace	Plane (Solid)
Fill On				
Fill Off				

8.7.1 Setting Fill Mode

Do one of the following:

- Run the FILL command.
- Set the value of the FILLMODE system variable.

Run the REGEN or REGENALL command to regenerate the viewport(s).

8.8 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, for example to de-emphasize less important details or entities that are used for reference only.

The options to set the Transparency are:

- ByLayer: apply the value of the transparency property of the layer.
- ByBlock: entities created using the transparency ByBlock inherit the transparency of the block when you insert the block into a drawing.
- 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.

8.8.1 Printing transparencies

Transparencies will only be printed if the Plot Transparencies option is checked on the Print or Page Setup dialog boxes.

8.9 Editing entity properties

Entity properties can be edited either in the **Properties** panel, or by using the **Rollover Tips** feature.

8.9.1 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 entity.

The properties displayed in the panel depend on the entity type(s) that you have selected. When multiple entities are selected on the shared properties display, the value of shared properties that vary between the selection set, display ***varies***.

As different from AutoCAD, the **Properties** panel in BricsCAD displays the **Handle** (read-only) of the selected entity, which can be useful to store and recall for programing purposes.

8.9.2 Editing with Rollover Tips

When Rollover Tips = ON (**RT** field in the **Status** bar), a selection of entity properties display at the cursor. Click a property to edit it.

When the Quad is on, click the **Quad** title bar to switch between Rollover Tips and the Quad cursor menu.

Right-click the **RT** field in the **Status** bar and choose **Customize...** in the context menu to open the **Customize / Properties** dialog box. Here you can define which properties display in the **Rollover Tips** for each entity type.

8.10 Drawing linear entities

Commands for drawing linear entities are:

	Draws polylines with x,y,z coordinates.
	Draws arcs.
	Creates a polyline that forms a boundary around closed areas.
	Draws circles.
	Draws wide circles from polylines.
	Draws ellipses and elliptical arcs.
	Draws 2D spirals or a 3D helices.

	Draws line segments.
	Draws multilines.
	Draws connected lines and arcs.
	Draws regular polygons out of polylines.
	Draws semi-infinite lines.
	Draws rectangles and squares.
	Draws revision clouds.
	Draws sketched (freehand) lines.
	Draws open and closed splines in 2D and 3D.
	Draws wide line segments.
	Draws infinite lines.

8.11 Desenhar linhas

Uma linha consiste em dois pontos: um ponto inicial e um ponto final. Usando o comando LINHA, você pode desenhar linhas simples ou uma série de linhas conectadas, mas cada linha é considerada uma entidade separada de Linha.



A ferramenta Linha está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.
- A aba Desenhar do Quad Sem seleção.

8.11.1 Procedimento geral

1 Lançar o comando LINHA.

Solicita a você: Início da linha ou [Seguir] <Último ponto>:

2 Especifique o ponto inicial.

A linha é exibida dinamicamente.

Solicita a você: Defina o ponto final ou [Angulo/Comprimento/Seguir/Desfazer]:

3 Especifique o ponto final.

A linha está criada. Uma nova linha é exibida dinamicamente.

Solicita a você: Defina o ponto final ou [Angulo/Comprimento/Seguir/Desfazer]:

4 Especifique o ponto final.

A linha está criada. Uma nova linha é exibida dinamicamente.

Solicita que você: Defina o ponto final ou [Angulo/Comprimento/Seguir/Fechar/Desfazer]:

5 Siga um dos procedimentos:

- Continue especificando os pontos finais para cada novo segmento de linha.
- Escolha a opção **Fechar** para se conectar ao ponto inicial da primeira linha.
- Pressione Enter ou clique o botão-direito para parar.

8.11.2 Especificar o ângulo e o comprimento

As opções são:

- Use as opções de comando Angulo e Comprimento.
- Use dimensões dinâmicas (DYNDIVIS = 1 ou 2).

  Pressione a tecla TAB para alternar entre os campos Angulo e Comprimento.

8.11.3 Especificar o ângulo para outra entidade linear

Clique o botão-direito no campo DUCS na barra de Status, e escolha DUCS em outras entidades para definir a opção **Habilitar para Entidades 2D** da variável de sistema UCSDETECT.

1 Lançar o comando LINHA.

2 Passe o mouse sobre a linha de referência ou o segmento linear de polilinha.

Um UCS temporário se alinha com a linha ou com o segmento de polilinha.

O ícone UCS é exibido no ponto mais próximo, no meio ou na extremidade da entidade sob o cursor.

3 Pressione a tecla Shift para bloquear o UCS temporário.

4 Especifique o ponto inicial da linha.

Os ângulos são agora relativos à linha selecionada ou segmento linear de polilinha.

Nota: O WCS é restaurado quando você conclui o comando LINHA.

8.12 Desenhar raios LSI

Um raio (LSI) é uma linha que começa em um ponto e se estende até o infinito. Como os raios se estendem até o infinito, eles não são calculados como parte das extensões do desenho.

O método padrão para o desenho de um raio é para selecionar o ponto inicial do raio, e especificar sua direção.

Como eles são frequentemente usados como referências para criar outras entidades, Raios (LSI) e Linhas Infinitas (LINHAINF) às vezes são chamados de linhas de construção.



A ferramenta Lsi está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.

8.12.1 Procedimento geral

1 Iniciar o comando LSI.

Solicita a você: Raio infinito: Início do raio ou [Horizontal/Vertical/Angulo/Bisset/Paralelo]:

2 Especifique o ponto inicial do raio.

O raio é exibido dinamicamente.

3 Especifica a direção do raio.

O próximo raio é exibido dinamicamente.

4 (opção) Mantenha-se especificando direções para desenhar uma série de raios, partindo todos do mesmo ponto.

5 Clique o botão-direito para parar de desenhar raios.

8.13 Desenhar linhas infinitas

Uma Linha Infinita (LINHAINF) é uma linha que passa através de um determinado ponto, orientada a um ângulo específico, e estende-se até o infinito em ambas as direções. Porque linhas infinitas se estendem ao infinito, elas não são calculadas como parte das extensões de desenho.

O método padrão para desenhar uma linha infinita é selecionar um ponto e, em seguida, especificar sua direção.

Como elas são frequentemente usados como referências para criar outras entidades, Raios (LSI) e Linhas Infinitas às vezes são chamados de linhas de construção.



A ferramenta Linha infinita está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.

8.13.1 Procedimento geral

1 Inicie o comando LINHAINF.

Solicita a você:

Linha infinita: Defina o ponto ao longo da linha ou [Horizontal/Vertical/Angulo/Bisset/Paralelo]:

2 Especifique o ponto inicial da linha infinita.

A linha infinita é exibida dinamicamente.

3 Especifique a direção da linha infinita.

A próxima linha infinita é exibida dinamicamente.

4 (opção) Continue especificando direções para desenhar uma série de linhas infinitas, começando do mesmo ponto.

5 Clique o botão-direito para parar.

8.14 Desenhar polilinhas

Uma polilinha é uma sequência aberta ou fechada de segmentos de linha e/ou arco conectados, que são tratados como uma única entidade. Cada segmento de uma polilinha pode ter uma largura constante ou ser afunilada ao longo do comprimento do segmento.

O comando PLINHA cria interativamente polilinhas abertas e fechadas, ao inserir pontos.

Os comandos LIMITE e -LIMITE criam polilinhas fechadas a partir de uma área fechada, definida por outras entidades.

O comando UNIR cria polilinhas a partir de uma série de linhas conectadas, ou polilinhas, arcos.

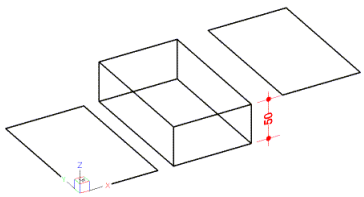
A ferramenta **Polilinha** está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.
- A aba Desenhar do Quad Sem seleção.

A propriedade **Thickness** especifica a altura de uma polilinha. Se Espessura = 0 (zero), não há diferença visual entre polilinhas e uma série de linhas e/ou arcos.

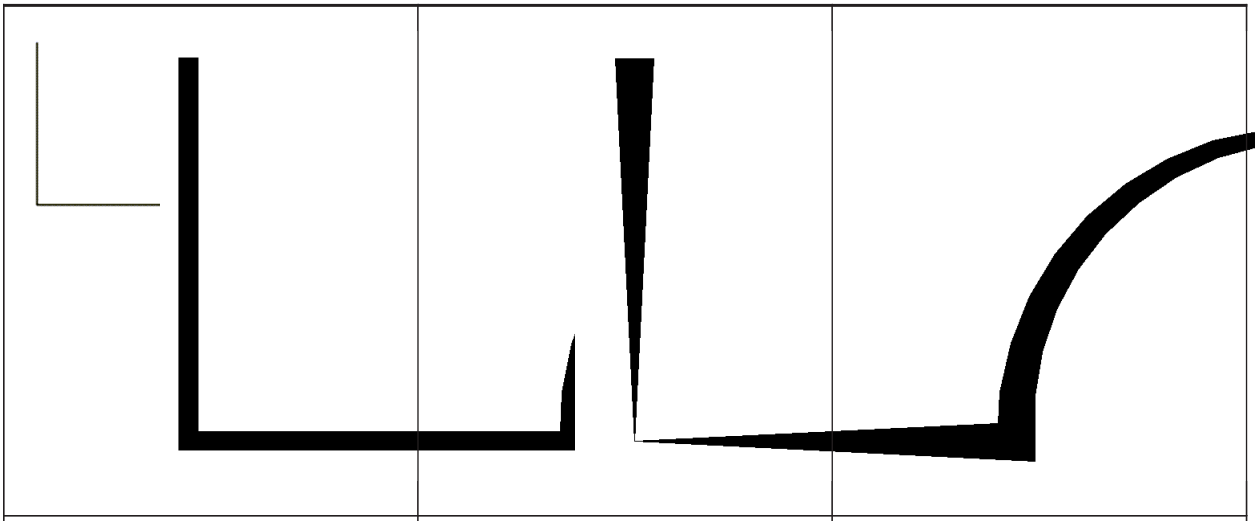
A espessura padrão é salva por meio da variável de sistema THICKNESS.

A propriedade **Elevation** especifica a distância entre o plano da polilinha e o plano XY do WCS (sistema de coordenadas Mundo).



ESPESSURA = 0 (esquerda), **ESPESSURA** = 50 (meio), **ELEVAÇÃO** = 50 (direita)

Quando uma polilinha é editada, você pode modificar a polilinha inteira, ou alterar apenas segmentos individuais, e pode adicionar ou remover vértices.



Polilinha de largura zero	Polilinha de largura constante	Polilinha de largura afunilada

Nota:

- Quando FILLMODE = 0, somente o contorno de todas as entidades preenchidas, como polilinhas largas e planos, é exibido.
- Dependendo do valor da variável de sistema GRIPS, as polilinhas têm alças em suas extremidades e vértices (GRIPS = 1) ou extremidades, vértices e pontos médios dos vértices (GRIPS=2)
- Polilinhas têm uma direção, que pode ser modificada pela opção **Direção Reversa** do comando EDITARP (editar polilinha).

8.14.1 Procedimento geral

1 Inicie o comando PLINHA.

Solicita a você: Selecione o início da polilinha ou [Seguir] <Último ponto>:

2 Especifique um ponto.

Solicita a você: Defina o próximo ponto ou [Desenhar arcos/Distância/Seguir/Meia largura/Largura]:

3 Especifique um ponto.

Solicita a você: Defina o próximo ponto ou [Desenhar arcos/Distância/Seguir/Meia largura/Largura/Desfazer]:

4 Especifique um ponto.

Solicita a você:

Defina o próximo ponto ou [Desenhar arcos/Fechar/Distância/Seguir/Meia largura/Largura/Desfazer]:

5 Continue a especificar pontos.

6 Siga um dos procedimentos:

- Escolha a opção **Fechar** para conectar ao ponto inicial, e assim criar uma polilinha fechada.
- Escolha a opção **Desfazer** para remover o vértice anterior.

- Clique o botão-direito ou pressione Enter para parar.

8.14.2 Criar polilinhas de limite

1 Inicie o comando LIMITE.

A caixa de diálogo **Limite** é exibida.

2 Clique no botão **Escolher pontos nos limites**.

A caixa de diálogo é fechada.

3 Clique na área em que você deseja criar a polilinha de limite.

A polilinha de limite se destaca.

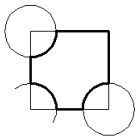
4 (opção) Manter a escolha de pontos para criar mais polilinhas-limite.

5 Clique o botão-direito para aceitar a(s) polilinha.

A caixa de diálogo **Limite** é aberta novamente.

6 Clique o botão **OK**.

A(s) polilinha é criada.



8.15 Desenhar multilinhas

Multi-linhas são feitas de duas ou mais linhas paralelas, de acordo com o estilo criado pelo comando ESTILOML.

A ferramenta Multilinha está disponível em:

- menu **Desenhar**
- a barra **Desenhar**
- painel da faixa de opções **Desenhar**

8.15.1 Procedimento geral

1 Inicie o Comando MLINE.

1 Configurações atuais: Justificação = Superior, Escala = 20.00, Estilo = Padrão

Você é solicitado: Especifique o ponto inicial ou [Justificação/Escala/Estilo]:

2 Especifique um ponto.

3 **Você é solicitado**: Definir ponto final ou [Angulo/Comprimento/Desfazer]:

4 Especifique um ponto.

5 **Você é solicitado**: Definir ponto final ou [Angulo/Comprimento/Seguir/Desfazer]:

6 Especifique um ponto.

7 **Você é solicitado**: Definir ponto final ou [Angulo/Comprimento/Seguir/Fechar/Desfazer]:

8 Siga um dos procedimentos:
   - Especifique mais pontos.
   - Escolha a opção **Fechar** para criar um loop fechado.
   - Clique o botão-direito para concluir.

Notas:

Multilinhas são criadas usando o estilo atual de multi-linhas, conforme definido pela variável de sistema CMLSTYLE.

A variável de sistema CMLJUST define a justificação da multilinha em relação aos pontos especificados. As opções são:

- **Superior**: Na linha superior do estilo de multilinha.
- **Zero**: No deslocamento zero da definição de estilo de multilinha.
- **Inferior**: Na linha inferior do estilo de multilinha.

A variável de sistema CMLSCALE define a escala da multilinha.

As propriedades de uma linha multilinha podem ser editadas no painel **Propriedades**.

8.16 Drawing splines

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved lines, which cannot be drawn as a polyline.



The Spline tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

8.16.1 General procedure

1 Launch the SPLINE command.

   Prompts you: First point of spline:

2 Specify a point.

   Prompts you: Second point:

3 Specify a point.

   Prompts you: Set next point or [Close/Fit tolerance/Undo]:

4 (option) Repeat step 3 to define more points.

5 Right click to stop adding more points.

   Prompts you: Select starting tangent point:

A line that is tangent to the spline displays dynamically from the start point of the spline.

6 Click to define the starting curve of the spline.

Prompts you: Select ending tangent point:

A line that is tangent to the spline displays dynamically from the end point of the spline.

7 Click to create the spline.

8.17 Desenhar croquis à mão livre

Um esboço à mão livre consiste em uma série de segmentos de linha reta, criados como entidades de linha individuais ou como uma polilinha.

Uma linha de croqui à mão livre não é adicionada ao desenho até você "gravar" o croqui no seu desenho.

A variável de sistema SKPOLY define se uma cadeia(s) de linhas conectadas ou polilinha(s) é criada.

A variável de sistema SKETCHINC define o comprimento ou incremento de cada linha ou segmento de polilinha. Quanto menor o incremento, mais preciso é o croquis, mas um valor pequeno aumenta o tamanho do arquivo.



A ferramenta Croqui está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.

8.17.1 Procedimento geral

1 Inicie o comando CROQUI.

Solicita a você: Comprimento dos segmentos <comprimento atual>:

2 Siga um dos procedimentos:

- Pressione Enter para aceitar o comprimento padrão.
- Digite um novo valor na linha de Comando.
- Clique em dois pontos para definir graficamente o comprimento.

3 Clique para começar a esboçar (caneta para baixo).

A mira é substituída pela ferramenta Lápis ()

Solicita a você:

Press ENTER ou clique para alternar [Apagar/Conectar/Salvar e sair/Sair sem salvar/Gravar (salvar)]:

4 Clique para começar a esboçar: à medida que você move o cursor, a linha do croquis é exibida em verde claro.

5 (opção) Clique para parar de desenhar, e clique novamente para retomar ao desenho.

6 (opção) Escolha a opção **Apagar**.

O cursor muda para a ferramenta **Apagar** ().

Mova o cursor para o final de uma linha de croqui para começar a apagar.

Clique para parar de apagar e reassumir o croquis.

7 (opção) Escolha a opção **Gravar (salvar)** para salvar a linha de esboço, e continuar esboçando.

8 Escolha a opção **Salvar e sair** para salvar as linhas do esboço, e sair do comando de croquis.

8.18 Drawing circles

The default method for drawing a circle is to specify a center point and a radius.

Other methods to draw circles can be found in the Circle submenu of the Draw menu and in theCirclestoolbar.

TheAI_CIRCTANcommand draws circles tangent to three linear entities.



The Circle tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

8.18.1 General procedure

1 Launch the CIRCLE command.

Prompts you:

Select center of circle or [2 Point/3 Point/Tangent-Tangent-Radius/turn Arc into circle/Multiple circles]:

2 Specify a point.

Prompts you: Set Radius or [Diameter]:

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.

8.18.2 Drawing a circle tangent to three entities

1 Launch the AI_CIRCTAN command.

Prompts you: Enter Tangent spec:

2 Click a linear entity.

Lines, arcs, circles, polyline segments, xlines and rays are accepted.

Prompts you: Enter second Tangent spec:

3 Click the second linear entity.

Prompts you: Enter third Tangent spec:

4 Click the third linear entity.

A circle tangent to the three selected entities is created.

8.18.3 Editing a circle

1 Select the circle.

The properties of the circle display in theProperties panel.

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.

8.19 Drawing arcs

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.

The Arc tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

8.19.1 General procedure

1 Launch the ARC command.

Prompts you: Specify the start point of the arc.

2 Specify a point.

Prompts you: Enter second point or [Angle/Center/Direction/End/Radius]:

3 Specify a point.

Prompts you: End point:

4 Specify a point.

8.19.2 Editing an arc

1 Select the arc.

The properties of the arc display in theProperties panel.

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.

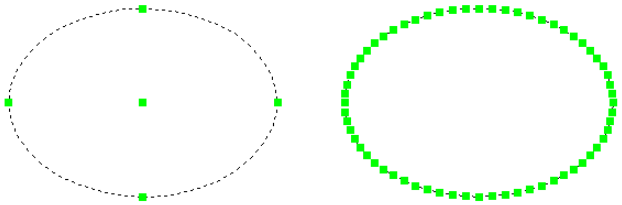
8.20 Desenhar elipses

O método padrão para desenhar uma elipse é especificar as extremidades de um eixo da elipse, e então especificar uma distância representando metade do comprimento do segundo eixo. As extremidades do primeiro eixo determinam a orientação da elipse. O eixo mais longo da elipse é chamado de eixo maior, e o mais curto é o eixo menor. A ordem na qual você define os eixos não importa. O programa determina os eixos maiores e menores com base em seus comprimentos relativos. Metade do eixo principal é chamado de raio maior, metade do eixo secundário é o raio menor. A razão (raio menor dividido pelo raio maior) define a excentricidade da elipse. Um círculo é uma elipse com uma excentricidade de 1.

Outros métodos para desenhar elipses podem ser encontrados no submenu Elipse do menu Desenhar e na barra de ferramentas Elipses.

A variável de sistema PELLIPSE controla se são criadas elipses reais ou uma representação em polilinha.

A diferença entre uma elipse real e uma representação em polilinha de uma elipse é visível quando você seleciona a elipse. Uma elipse real tem cinco alças: ponto central e as extremidades dos eixos. Uma representação em polilinha de uma elipse é uma polilinha fechada composta por segmentos de arco.



Elipse real (à esquerda) e a representação como polilinha de uma elipse (à direita)

A ferramenta Elipse está disponível em:

- O menu Desenhar.
- A barra de ferramentas Desenhar.
- O painel Desenhar da Faixa de opções.
- A aba Desenhar do Quad Sem seleção.

8.20.1 Procedimento geral para criar uma elipse

1 Inicie o comando ELLIPSE.

Solicita: primeira extremidade do eixo de elipse ou [Arco/Centro]:

2 Especifique um ponto:

Solicita a você: Segunda extremidade do eixo:

3 Especifique um ponto:

Solicita a você: Definir Outro eixo ou [Rotação]:

4 Especifique um ponto.

8.20.2 Procedimento geral para criar um arco elíptico

Um arco elíptico é uma porção de uma elipse.

1 Inicie o comando ELLIPSE.

Solicita: primeira extremidade do eixo de elipse ou [Arco/Centro]:

2 Selecione a opção Arco.

Solicita a você:

3 Especifique um ponto:

Solicita a você: Segunda extremidade do eixo:

4 Especifique um ponto:

Solicita a você: Definir Outro eixo ou [Rotação]:

5 Especifique um ponto.

Solicita a você: Defina o ângulo inicial do arco ou [Parâmetro]:

6 Siga um dos procedimentos:

- Digite um valor no campo dinâmico.
- Clique para especificar graficamente o arco.

Solicita a você: [Parâmetro/Angulo incluído] <Angulo final>:

7 Clique um ponto.

8.20.3 Editar uma elipse ou um arco elíptico

1 Selecione a elipse ou arco elíptico.

As propriedades da elipse são exibidas no painel Propriedades.

2 Digite um novo valor no campo**Centro**,**Raio maior**,**Raio menor**,**Razão do raio**,**Angulo inicial**ou**Angulo final**, e pressione Enter.

3 Pressione a tecla Esc para parar a edição.

Nota: Editar os campos Angulo inicial ou Angulo final de uma elipse transforma a elipse em um arco elíptico. Definir ambas as propriedades como 0 (zero) transforma um arco elíptico em uma elipse.

8.21 Desenhando retângulos

Um retângulo é criado como uma polilinha fechada, com quatro lados.

Um retângulo é desenhado especificando dois cantos opostos. Dimensões dinâmicas permitem especificar a largura e a altura do retângulo.

A menos que a opção **Rotacionada** seja escolhida, os lados de um retângulo são sempre paralelos ao eixo X e ao eixo Y do sistema de coordenadas atual (WCS ou UCS).



A ferramenta Retângulo está disponível em:

- menu **Desenhar**

- a barra **Desenhar**
- painel da faixa de opções **Desenhar**
- a aba **Desenhar** do **Quad Sem Seleção**

8.21.1 Procedimento geral

1 Inicie o comando RETANGULO.

1 **Solicita a você**:

Selecione o primeiro canto do retângulo ou [Chanfro/Concord/Rotacionado/Quadrado/Elevação/Espessura/Largura da linha/Área/Dimensões]:

2 Especifique um ponto.

3 O retângulo é exibido dinamicamente.

Solicita a você: Outro canto do retângulo:

4 Faça um dos seguintes:

- Especifique um ponto.
- Digite uma distância nos campos de dimensão dinâmica.
- Pressione a tecla TAB para alternar entre os campos. Pressione Enter para confirmar.

Nota:

Como um retângulo é criado como uma polilinha fechada, os retângulos têm uma direção. A direção de uma polilinha fechada é positiva se for desenhada no sentido anti-horário e negativa se for desenhada no sentido horário. Retângulos têm um sentido anti-horário se as coordenadas X e Y do segundo ponto forem mais altas ou mais baixas que as coordenadas do primeiro ponto (= o movimento do cursor é positivo ou negativo ao longo dos eixos X e Y). Retângulos têm um sentido horário se as coordenadas X ou Y do segundo ponto forem maiores ou menores que a coordenada X ou Y do primeiro ponto (= o movimento do cursor é positivo ao longo do eixo X e negativo ao longo do eixo Y ou vice-versa).

A opção Reverter Direção do comando EDITARP (Editar polilinha) altera a direção de uma polilinha.

8.22 Desenhar polígonos

Polígonos são polilinhas fechadas, compostos por no mínimo três e no máximo de 1.024 lados de igual comprimento.

Você pode desenhar polígonos usando os seguintes métodos:

- **Centro - Vértice**: primeiro defina o ponto central, depois o vértice (= método do círculo circunscrito).
- **Centro - Lado**: primeiro defina o ponto central, depois o ponto médio de um lado (= método do raio do círculo inscrito).
- **Borda**: define o comprimento do lado de um polígono.



A ferramenta Polígono está disponível em:

- menu **Desenhar**
- a barra **Desenhar**
- painel da faixa de opções **Desenhar**

- a aba **Desenhar** do **Quad Sem Seleção**

8.22.1 Procedimento geral

1 Iniciar o comando POLIGONO.

1 **Solicita a você**: Polígono: Defina o número de lados ou [Largura da linha/Vários polígonos] <4>:

2 Siga um dos procedimentos:

- Pressione Enter ou clique o botão-direito para aceitar o número atual de lados.
- Digite um valor, e pressione Enter.

Solicita a você: Especificar por: Definir centro do polígono ou [especificar por Borda]:

3 Especifique um ponto.

4 **Solicita a você**: Especifique por: Selecione o ponto médio do lado ou [especificar por Vértice]:

5 Especifique um ponto.

Nota:

- A opção escolhida na etapa 3 (ponto médio do lado ou especificar por Vértice) é lembrada na próxima vez.
- O número de lados é definido pela variável de sistema POLYSIDES, mas o valor não é salvo entre as sessões. A primeira vez que você inicia o comando POLIGONO em uma sessão do BricsCAD a variável POLYSIDES = 4.

8.23 Drawing traces

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.



The Trace tool is available on

- the **Draw** menu.
- the **Draw** ribbon panel

The default trace width is saved through the TRACEWID system variable.

8.23.1 General procedure

1 Launch the TRACE command.

1 **Prompts you:** Width of trace <current width>:

2 Do one of the following:

- Right click to accept the current width.
- Type a value, then press Enter.
- Define a new current width graphically by clicking two points.

Prompts you: Start of trace:

3 Specify the start point of the trace.

4 **Prompts you:** Next point:

5 Specify the end point of the first trace.

6 A line indicates the direction of the first trace.

The direction of the second trace displays dynamically.

7 Do one of the following:

- Right click to create the trace.
- Specify the endpoint of the next trace.

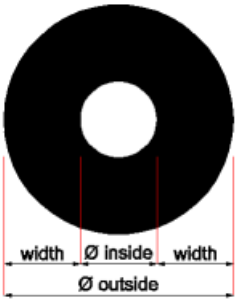
8 Do one of the following:

- Repeat step 5 to keep adding traces.
- Right click to stop.

Nota: When the FILLMODE system variable is turned off, all filled entities, such as traces, display and print as outlines.

8.24 Desenhar anéis

Os anéis são sólidos, círculos preenchidos ou anéis preenchidos criados como polilinhas largas fechadas, de 2-segmentos.



O método padrão para desenhar anéis é especificar seus diâmetros internos e externos, e especificar seu centro. Se o diâmetro interno for zero, um círculo preenchido será criado.



A ferramenta Anel está disponível em:

- menu **Desenhar**
- painel da faixa de opções **Desenhar**

8.24.1 Procedimento geral

1 Iniciar o comando ANEL.

1 **Solicita a você**: Diâmetro interno do anel [2 Pontos/3 Pontos/Tangente Rangente Raio] <atual diâmetro interno>:

2 Siga um dos procedimentos:

- Clique o botão-direito para aceitar o atual diâmetro interno.
- Digite um novo valor, e pressione Enter.

- Definir graficamente um novo diâmetro interno, clicando em dois pontos.

Solicita a você: Diâmetro externo do anel <atual diâmetro externo>:

3 Siga um dos procedimentos:
- Clique o botão-direito para aceitar o atual diâmetro externo.
- Digite um novo valor, e pressione Enter.
- Defina graficamente um novo diâmetro externo, clicando em dois pontos.

Solicita a você: Centro do anel:

Especifique um ponto.

Solicita a você: Centro do anel:

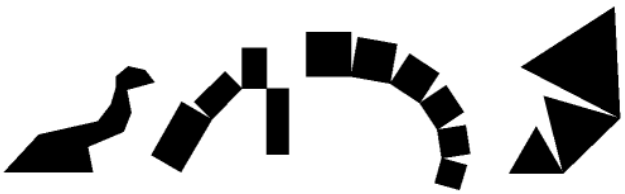
4 Siga um dos procedimentos:
- Continuar criando anéis.
- Clique o botão-direito para parar.

Nota:

- Quando a variável de sistema FILLMODE está Desligada, todas as entidades preenchidas, como anéis, exibem e imprimem como contornos.
- O valor atual dos diâmetros interno e externo é salvo através das variáveis de sistema DONUTID e DONUTOD. Esses valores não são salvos entre as sessões. A primeira vez que você inicia o comando DONUT em uma sessão do BricsCAD DONUTID=0.5 e DONUTOD=1.

8.25 Drawing solids

2D Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color.



Left to right: quadrilateral, rectangular, square and triangular solids

The Solid tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

8.25.1 General procedure

1 Launch the SOLID command.

1 **Prompts you**: Set first point of plane or [Rectangle/Square/Triangle]:

2 Specify a point.

3 **Prompts you**: Second point:

4 Specify a point.

5 **Prompts you**: Third point of plane:

6 Specify a point.

7 **Prompts you**: Fourth point:

8 Specify a point.

9 The solid is created.

Prompts you: Third point of plane:

10 Do one of the following:

- Keep specifying points to create more solids.
- Right click to stop.

Nota: When the FILLMODE system variable is turned off, all filled entities, such as solids, display and print as outlines.

8.26 Drawing revision clouds

A Revision Cloud is an open or closed polyline 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.



Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle).

Nota: The **Reverse** option is available only in **Entity** mode or when creating an open revision cloud using the **Freehand** mode.

REVCLOUDCREATEMODE: sets the default mode.

REVCLOUDMINARCLENGTH: sets the minimum arc length.

REVCLOUDMAXARCLENGTH: sets the maximum arc length.

REVCLOUDARCSTYLE: sets the default style (normal or calligraphy).

REVCLOUDGRIPS: when On, grip editing a revision cloud also affects adjacent segments.



The Revision Cloud tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

8.26.1 General procedure

1 Launch the REVCLOUD command.

1 **Prompts you**:

Specify first point or [Arc length/Entity/Rectangular/Polygonal/Freehand/Style] <default mode>:

2 Specify a point to start creating a freehand mode revision cloud.

3 As you move the cursor, the revision cloud is being created.

4 Move the cursor to the start point to finish.

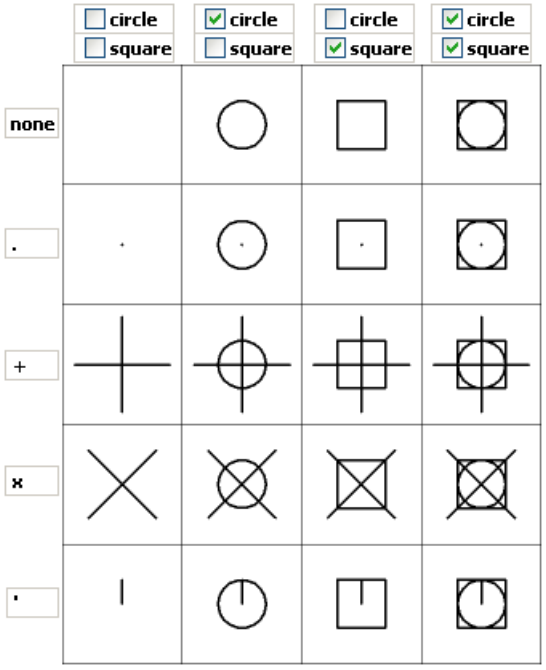
5 BricsCAD reports in the command line: Revision cloud finished.

Nota:

- If you right click at the first prompt you start creating a revision cloud in the default mode (= current value of the REVCLOUDCREATEMODE system variable).
- If you right click before the cursor is at the start point, an open revision cloud is created.

8.27 Drawing points

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, set by the PDMODE (Point Display Mode) system variable:



The size of the point representation style is controlled through the PDSIZE (Point Display Size) system variable. When PDSIZE=0 the point display size is 5% of the screen size.

Nota: If the PDSIZE=0 (zero), use the REGEN command to resize the display of the points to 5% of the screen size after zooming in or out.

The Point tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel

8.27.1 General procedure

1 Launch the POINT command.

1 **Prompts you**: Location of point or [point Settings.../Multiple points]:

2 Specify a point.

8.28 Drawing wipeouts

A wipeout is a polygonal area that hides the underlying entities 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 **Frame** option of the WIPEOUT command sets the value of the WIPEOUTFRAME system variable, which controls whether the wipe out outlines display or not, and whether they are printed or not.

Nota: Wipeouts hide existing entities only. Use the **Bring to front** option of the DRAWORDER command to hide all entities.



The Wipeout tool is available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad**

8.28.1 General procedure

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Specify a point.

3 **Prompts you**: Specify next point:

4 Specify a point.

5 **Prompts you**: Specify next point or [Undo]:

6 Specify a point.

7 **Prompts you**: Specify next point or [Undo/Close]:

8 Do one of the following:

- Keep specifying points.
- Right click or choose **Close** to stop.

8.28.2 To create a wipeout from a polyline

1 Launch the WIPEOUT command.

1 **Prompts you:** Specify start point or [Polyline/Frames]:

2 Choose the **Polyline** option.

Prompts you: Select a closed polyline [selection options (?)]:

3 Select a closed polyline, composed of line segments only.

4 **Prompts you**: Delete selected polyline? [Yes/No] <No>:

5 Do one of the following:

- Press Enter to keep the polyline.
- Choose **Yes** to delete the polyline.

8.29 Desenhar hélices

Uma hélice é uma curva que gira ao redor de um eixo, na superfície de um cilindro (hélice cilíndrica) ou um cone (hélice cônica) enquanto se eleva em um ângulo constante para o topo, partindo de uma base.

Uma espiral é uma curva planar que se enrola em torno de um ponto central fixo a uma distância continuamente crescente a partir desse ponto.

O comando HELICE pode criar tanto hélices 3D como espirais 2D.



A ferramenta Helice está disponível em:

- menu **Desenhar**
- barra **Polilinhas**
- painel da faixa de opções **Desenhar**

8.29.1 Procedimento geral para desenhar uma espiral

1 Inicie o comando HELICE.

1 Número de voltas = 3 Torcer = CCW

Você é solicitado: Especifique o ponto central da base:

2 Especifique um ponto.

3 Especificar raio da base ou [Diâmetro] <1>:

4 Siga um dos procedimentos:

- Especifique um ponto.
- Digite um valor, e pressione Enter.

Você é solicitado: Especifique o raio superior ou [Diâmetro] <raio base>:

5 Siga um dos procedimentos:

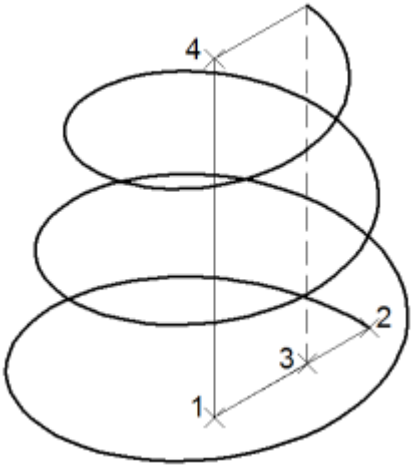
- Especifique um ponto.
- Digite um valor, e pressione Enter.

Você é solicitado: Especifique a altura da hélice ou [Extremidade eixo/Voltas/Altura da volta/Torção]

<1>:

6 Digite 0 (zero), e pressione Enter

8.29.2 Procedimento geral para desenhar uma hélice



1 Inicie o comando HELICE.

1 Número de voltas = 3 Torcer = CCW

Você é solicitado: Especifique o ponto central da base:

2 Especifique um ponto (1).

3 **Você é solicitado**: Especifique raio da base ou [Diâmetro]< 1>:

4 Faça um dos seguintes (2):
- Especifique um ponto.
- Digite um valor, e pressione Enter.

5 Faça um dos seguintes (3):
- Pressione Enter para definir o raio superior igual ao raio da base, criando assim uma hélice cilíndrica.
- Especifique um ponto.
- Digite um valor, e pressione Enter.

Você é solicitado: Especifique a altura da hélice ou [Extremidade eixo/Voltas/Altura da volta/Torção] <1>:

6 Faça um dos seguintes (4):
- Especifique um ponto.
- Digite um valor, e pressione Enter.

8.29.3 Editar uma hélice

1 Selecione a hélice.

1 As propriedades da hélice são exibidas na categoria **Geometria** no painel **Propriedades**.

2 Editar as propriedades:
   - **Restringir**: restringe tanto a altura geral, como a altura de cada volta ou o número de voltas. As propriedades não restritas são ajustadas automaticamente.
   - **Altura**: Altura total da hélice. altura = altura da volta * número de voltas.
   - **Voltas**: Número de voltas.
   - **Altura da volta**: Altura de uma volta.
   - **Raio da base**: raio inicial.
   - **Raio superior**: raio final.
   - **Torção**: no sentido CW (horário) ou CCW (anti-horário).

Nota:

- Defina o **Raio superior** = **Raio da base** para hélices cilíndricas.
- Defina **Altura** ou **Altura da volta** para zero para traçar espirais.
- O comando EXPLODIR transforma uma hélice em uma spline.

8.30 Desenhar polilinhas 3D

Uma polilinha 3D é composta por segmentos lineares no espaço 3D.



A ferramenta POLI3D está disponível em:

- menu **Desenhar**
- barra **Polilinhas**
- painel da faixa de opções **Desenhar**

8.30.1 Procedimento geral

1 Iniciar o comando POLI3D.

1 **Você é solicitado**: Selecione o início da polilinha ou [Seguir] <Último ponto>:

2 Especifique um ponto.

3 **Você é solicitado**: Próximo ponto.

4 Especifique um ponto.

5 **Você é solicitado**: Próximo ponto ou [Fechar/Desfazer]:

6 Siga um dos procedimentos:
   - Escolha a opção **Fechar** para conectar ao ponto inicial e criar uma polilinha 3D triangular planar.
   - Especifique um ponto.

   Você é solicitado: Defina o próximo ponto ou [Fechar/Desfazer]:

7 Siga um dos procedimentos:
   - Escolha a opção **Fechar** para se conectar ao ponto inicial.
   - Especifique um ponto.

Você é solicitado: Defina o próximo ponto ou [Fechar/Desfazer]:

8 Continue especificando pontos ou pressione Enter para parar.

8.30.2 Editar uma polilinha 3D

1 Selecione a polilinha 3D.

1 As propriedades são exibidas no painel Propriedades.

2 Em **Misc**:

- **Ajustar/Suave**: Selecione **Nenhum**, **Cúbica** ou **Quadrática**.

  As opções **Cúbica** e **Quadrática** criam uma representação em Spline da polilinha 3D.

- **Fechado**: Quando definido como **Não**, remove o segmento de fechamento.

8.31 Creating regions

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 using a wireframe visual style. However, if the original entity had a width or lineweight, that information is lost when you create the region.

In rendered visuals styles, region entities are displayed using a solid fill.



8.31.1 To create regions

1 Launch the REGION command.

1 **You are prompted**: Select entities or boundaries [selection options (?)]:

2 Select one or more entities.

3 Entities in set: x

You are prompted: Select entities or boundaries [selection options (?)]:

4 Select more entities or press Enter to stop selection.

5 BricsCAD reports: x region(s) created.

8.32 Creating hatches and gradient fills

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.

Nota:

- 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. By default the maximum number of dashes is 100 000.



The Hatch and Gradient tools are available on:

- the **Draw** menu
- the **Draw** toolbar
- the **Draw** ribbon panel
- the **Draw** tab of the **No Selection Quad** (Hatch only)

8.32.1 General procedure

1 Launch the HATCH or GRADIENT command.

1 The **Hatch and Gradient** dialog displays.

2 Do one of the following:

3 (option) Click the **Hatch** tab to create a hatch, then define the hatch properties.

4 (option) Click the **Gradient** tab to create a gradient fill, then define the gradient fill properties.

5 Define the properties for the hatch or gradient fill.

6 Choose the boundary for the hatch or gradient fill.

7 Set the options.

8 Click the **OK** button to create the hatch or gradient fill.

Hatches and gradient fills are created on the layer defined by the HPLAYER system variable. Default = current layer.

8.32.2 To define hatch properties

1 Select the **Hatch** tab on the **Hatch and Gradient** dialog box.

2 (option) Click the **Inherit properties** button.

The **Hatch and Gradient** dialog closes temporarily.

You are prompted: 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**:
- **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= ON)
  - 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 (Support File Search Path) system variable.

4 If the **Pattern Type** is **Predefined** or **Custom**:
- Double click the **Swatch** field,
- Select a **Pattern Name** in the **Hatch pattern** palette, then double click the pattern or click the **OK** Button.
- Type a scale in the **Scale** settings field or choose the scale from the drop down list.
- Type an angle in the **Angle** settings field or choose the angle from the drop down list.

5 If the **Pattern Type** is **User Defined**:
- Specify the **Pattern Spacing** in drawing units.
- Optionally set the **Cross Hatch** option.

6 Define the **Hatch Origin**.

The options are:
- **Use current origin** : The origin as defined by the HPORIGIN 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 system variable.

7 Set the **Color** for the hatch pattern.

8 Set the **Background color**.

8.32.3 To define gradient fill properties

1 Select the **Gradient** tab on the **Hatch and Gradient** dialog box.

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 box, 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.

  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.
- Select one of the 9 gradient patterns.

- The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.

4 (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.

5 **Angle**: Specifies the angle of the gradient fill, relative to the current UCS.

8.32.4 To define the boundary

1 Select a **Boundary retention** option.

2 Select an **Island** detection option.

This applies to enclosed areas inside hatch boundaries.

3 Set the **Boundary tolerance**. This value defines the maximum gap size in the boundary. When it is set to 0 (zero), BricsCAD sets the tolerance with respect to the current view size. Zoom out to increase the tolerance.

This value is saved through the HPGAPTOL system variable.

4 (option) Click the **Select** button to select a **Boundary set**.

The **Hatch and Gradient** dialog temporarily closes to let you select entities.

If a selection set was active when you launch the HATCH or GRADIENT command, this selection set will be used as the **Boundary Set**. In this case, the **Hatch and Gradient** dialog closes and immediately reopens when you click the **Select** button.

5 Do one 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.

8.32.5 Set the options

- The **Annotative** option creates annotative hatches.
- Select the **Associative** option to associate the hatch with its boundary.
- Select the **Create Separate Hatches** option. When multiple boundaries are detected or multiple entities are selected, separate hatches are created, instead of a single hatch.
- Choose a **Draw order** option.

  This option is reset to **Send behind boundary** when the drawing is closed and reopened.
- **Layer**: choose an existing layer or **Use Current** (default).
- **Transparency**: sets the transparency; values between 0 and 90 are accepted.

8.33 Editing hatches and gradient fills

Hatches and gradient fills can be edited:

- In the **Properties** panel.
- Using the HATCHEDIT command, in a dialog box.
- Using tools in the Edit command group in the Quad when a hatch or gradient fill entity is selected.

8.33.1 To edit a hatch or gradient fill in the properties panel

1 Select the hatch or gradient fill in the drawing.

1 The properties of the selected entity display in the **Properties** panel.

2 Select the property you want to edit in the **Properties** bar.

The selected property field is activated.

3 Edit the selected property.

4 Press the Esc key to stop editing the hatch.

Nota:

- When multiple hatches or gradient fills are selected properties which are different display as ***varies***. If you edit such property, the changes apply to entire selection.
- Linetype, Linetype scale and Lineweight properties apply to **User Defined** pattern types only.

8.33.2 To edit a hatch or gradient fill in a dialog box



The Edit Hatch tool is available on:

- the **Modify** menu
- the **Modify** toolbar

- the **Modify** ribbon panel

1 Do one of the following:
   - Launch the HATCHEDIT command.
   - Select a hatch or gradient entity.
   - Double click a hatch or gradient fill entity.

   The **Hatch Edit** dialog box displays.

2 Edit the properties and settings.

3 The changes are applied in real time.

4 Click the **OK** button to accept the changes.

8.33.3 To edit a hatch or gradient fill using the Quad

1 Pause the cursor on the hatch or gradient fill.

1 The Quad displays.

2 Move the cursor to the Quad.

3 (option) In the **Edit** tab, click the **Add Loop** button.

   You are prompted: specify internal point or [Select entities/Undo]:
   - Click inside a closed area,
   - or

     Type **S** , then press Enter to select entities.
   - Right click or press Enter to conclude.

4 (option) In the **Edit** tab, 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.

5 (option) In the **Edit** tab, click the **Send to back** button.

   All hatches in the drawing are sent behind other entities.

6 (option) In the **Edit** tab, click the **Match** button.

   You are prompted: select entities to copy properties to or [Settings/selection options (?)]:
   - Select the hatch entities to copy the properties to.
   - Right click or press Enter to conclude.

7 (option) In the **Draw** tab, click the **Generate Boundary** button.

   The boundary of the hatch is generated.

9. Modelagem 3D

9.1 Direct modeling overview

Direct modeling is much like sculpting on-screen: just like a sculptor starts with a lump of clay or a piece of stone or wood, you start with a base volume, which is then further detailed by adding, removing and deforming geometry. All operations are executed dynamically, and you decide whether to define dimensions intuitively or by specifying values. Unlike history-based modeling, the order in which features are created is not important. All operations are performed in interactive mode: the intermediate result of an operation shows dynamically, according to user input defined by mouse movement.

A key feature of direct editing capabilities provided by BricsCAD consists in preserving the design intent. 3D dimensional constraints (**Distance**, **Radius**, and **Angle**) imposed for faces and edges give you 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 specify all geometrical constraints manually since most of them can be recognized automatically by setting **Design Intent Recognition** options.

Another feature of **Direct Editing** operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations based on the currently selected or previewed entity or selection set. At the same time the Quad provides the other basic solid editing operations such as copy, union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported. When the separately licensed BricsCAD Communicator is installed, the IMPORT command supports all major 3D model formats, both industry standard and proprietary formats.

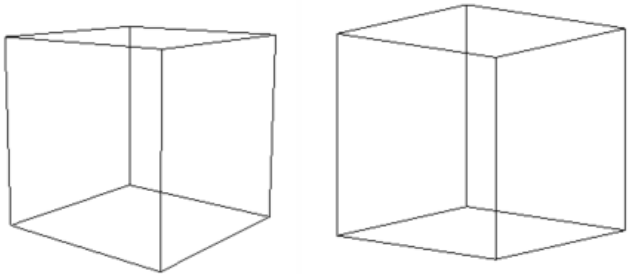
Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

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

9.2 Navigate in 3D

As you navigate around and through your 3D model, the location of the model in space remains constant. It is your current view (viewpoint), of the model that is changing. You can see the current view properties in the **Properties** panel when nothing is selected. The **Camera** (X,Y,Z) represents your location as you view the model. Pretend you are 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 control to be aware of for navigation purposes.

You can set **Perspective** to **On** or **Off** for the current view. When set to **On**, the view displays the visual perspective mode (left). When set to **Off**, it displays the parallel perspective mode (right).



9.2.1 To control the Perspective property

Do one of the following:

- Select the **View / Perspective** property in the **Properties** panel.
- Right click the **Look From** widget, then choose **Perspective** in the context menu.

9.2.2 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.

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.

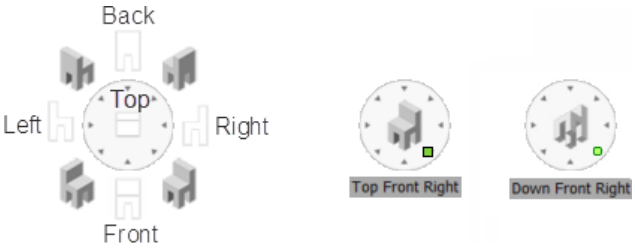
- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode. In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

See the 'View Manipulation using the Mouse and Keyboard' article for a comprehensive list.

9.2.3 The LookFrom navigation tool

The **LookFrom** tool is displayed in the upper right corner of the **Drawing** area. 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.



A right-click menu offers access to additional controls and settings.

Choose **LookFrom Settings…** in the context menu to open the **Settings** dialog box and edit the **LookFrom** related system variables.

9.3 Direct modeling commands

BricsCAD direct modeling commands can create both solid and surface geometry. To distinguish between direct modeling commands and 'normal' commands, all direct modeling commands start with 'DM'.

Direct modeling commands are available in license levels Pro, BIM, Mechanical and Ultimate. Commands that create geometry are also available in BricsCAD Shape, along with basic modification commands such as DMPUSHPULL, DMCHAMFER and DMFILLET.

9.3.1 Creation commands:

DMEXTRUDE: 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.

DMREVOLVE: creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis.

DMTHICKEN: creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities.

DMCOPYFACES: 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.

DMSTITCH: converts a set of region and surface entities that bound a watertight area to a 3d solid.

9.3.2 Modification commands:

DMCHAMFER: creates equal distance and variable chamfers between adjacent faces sharing a sharp edge.

DMDEFORMCURVE: deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

DMDEFORMMOVE: deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

DMDEFORMPOINT: 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.

DMFILLET: creates a smooth fillet between adjacent faces sharing a sharp edge.

DMPUSHPULL: adds or removes volume from a solid by moving/offsetting a face.

DMROTATE: rotates solids, faces, edges, or vertices of a solid around an axis.

DMTWIST: modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

9.3.3 Helper commands:

DMAUDIT: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DMAUDITALL: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces) in drawings inserted as external references.

DMGROUP: creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

DMSELECT: selects edges and faces of 3D solids or surfaces based on their geometric properties.

DMSELECTEDGES: selects the edges of faces and solids.

DMSIMPLIFY: simplifies the geometry and topology of 3d solid entities. The command 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.

The direct modeling commands are available in:

- The **Model** tab on the Quad cursor menu when the cursor is over an entity or subentity or when a selection set exists.
- The **Direct Modeling** Menu.
- The **Direct Modeling**, **Fillet**, **Chamfer** and **Deform** toolbars.
- The **Home / Direct Modeling** ribbon panel (**Modeling** workspace).

Icon	Command	Applies to
		Any face of a 3D solid or surface.
		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.
		Planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface. Whole 3D solids, surfaces, and other entities.

Icon	Command	Applies to
		3D solids, surfaces, and their faces.
		Surfaces.
		3D solids and surfaces.
		Planar faces of 3D solids and surfaces. This tool launches the UCS command with the Face option selected. ***Nota:*** When the Dynamic UCS feature is ON, the UCS aligns automatically to the solid face under the cursor and the UCS-icon changes accordingly.
	Solid mode	Closed 2D entities and detected closed boundaries. If the **Select detected boundaries** option of the SELECTIONMODES system variable is switched ON, boundaries enclosed by 2D linear entities in the XY-plane of the current UCS are detected automatically. Planar faces of 3D solids and surfaces. Closed 2D entities and detected closed boundaries on a planar face of a solid.
	Solid mode	Closed 2D entities and detected closed boundaries.* Planar faces of solids and surfaces. Closed 2D entities and detected closed boundaries on the planar face of a solid.
	Extract Faces	Surfaces and faces of 3D solids.
		A set of 3D solid faces.
		Faces of 3D solids and surfaces. Surfaces. Wire entities; lines, polylines, circles, ellipses, arcs, helices, and splines.
	Constant radius mode	Sharp edges of a 3D solid or surface.
	Variable radius mode	Sharp edges of a 3D solid or surface.

Icon	Command	Applies to
		Sharp edges of a 3D solid or surface.
	Advanced Mode	Sharp edges of a 3D solid or surface.
		Faces of a solid or a surface. Whole 3D solids and surfaces. Free edges of a surface (a surface edge is called "free" if it has only one adjacent face) Edges between coplanar faces.
	Surface mode	3D curves and edges of 3D solids and surfaces.
	Surface mode	3D curves and edges of 3D solids and surfaces.
		Faces of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		3D solids, surfaces or regions.

9.4 Direct modeling operations

9.4.1 Controlling the coordinate system

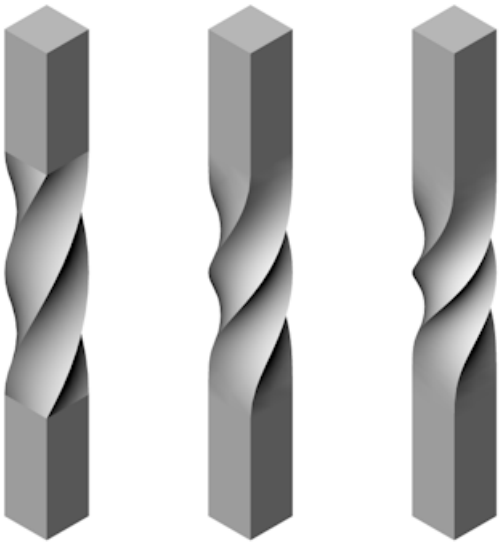
Since entities are created in the XY-plane of the current coordinate system (UCS), it is necessary to align the UCS with the 3D solid face. You have two options here:

- The **Align UCS** tool () launches the UCS command with the **Face** option selected.
- Click the **DUCS** field in the **Status Bar** to toggle the dynamic UCS feature ON and OFF. When ON, the UCS aligns automatically to the solid face under the cursor when executing a command and the UCS-icon changes accordingly. Press the Tab key to select an obscured face. Press the Shift key to temporarily lock the UCS, which allows you to work outside the hovered entity or face. Press the Shift key again to unlock it.

9.4.2 Selecting geometry

- Hit the Tab key to select obscured geometry.

- The SELECTIONMODES system variable controls which subentities highlight: edges, faces, and/or closed boundaries. Hold down the Ctrl key to overrule SELECTIONMODES, and select the subentity under the cursor.
- Window selection: to select the faces or edges of 3D solids and surfaces, press the CTRL key to cycle the options. The icon next to the cursor indicates what will be selected.
- The DMSELECT command allows you to select edges and faces based on their geometrical and topological properties:
  - Edges of a particular length.
  - Edges that form a concave/convex network.
  - Edge loops on the border of a set of connected faces.
  - Faces of a particular area.
  - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity).



G0: No continuity (left), G1: Tangent Faces (middle), and G2: Curvature continuity (right)

  - Fillets of a specific radius.
  - Cylindrical protrusions of a specific radius.
  - Protrusion or depression features; pockets, extrusions, or ribs.

The value of a particular length, area, or radius can be taken from a selected edge/face.

9.4.3 Moving faces to control dimensions

A dynamic dimension displays with the DMPUSHPULL command.

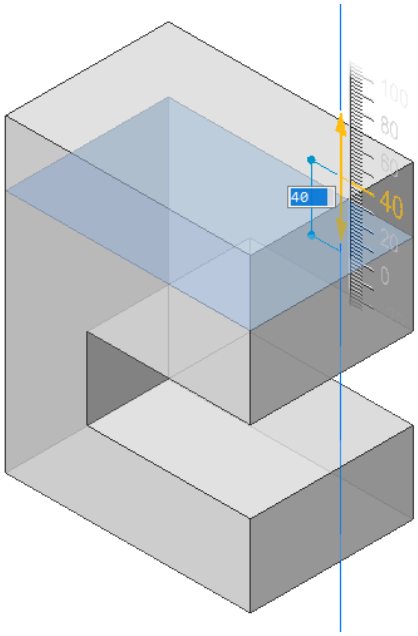
Make sure the **Quad** and **Dynamic Dimensions** are switched ON.

1 Hover the cursor over the face you want to move.

1 The face highlights.

2 Choose **Push/Pull** ( ) in the **Model** tab of the Quad.

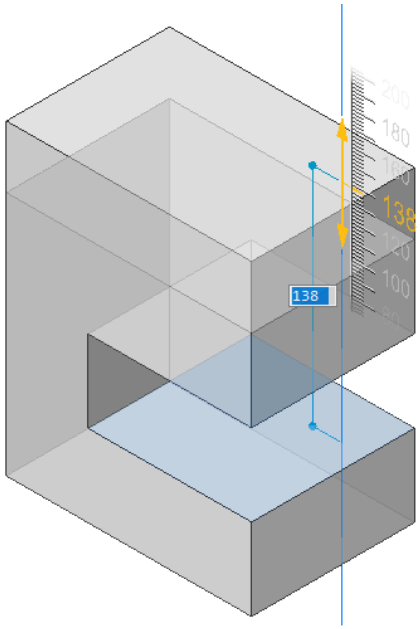
The selected face moves with the cursor and a dynamic dimension field displays the relative distance along with a ruler.



When the cursor is on the ruler, the face moves with fixed intervals. Zoom in or out to increase or decrease the interval.

When the cursor is outside the ruler, the face moves freely.

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 and hit the Tab key to reverse the display order of reference faces.

4 Do one of the following.
   - Type a value in the dynamic dimension field, then hit Enter.
   - Using the ruler: click when the desired value displays.

9.4.4 Making holes

Method 1:

1 Create the subtractor solid,

2 Use the SUBTRACT command.

3 Depending on the value of the DELETETOOL system variable, the subtractor solid is either deleted or retained.

Method 2:

1 Create the outline of the opening on the 3D solid face.

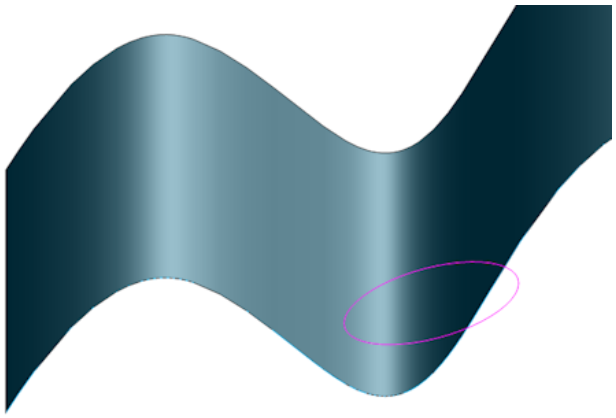
2 Place the cursor over the outline, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

3 Move the cursor inside the solid to make the opening.

4 Do one of the following:
   - Specify a point.
   - Type a value.

To make a hole in a surface

1 Create a 2D closed entity, e.g. an ellipse.

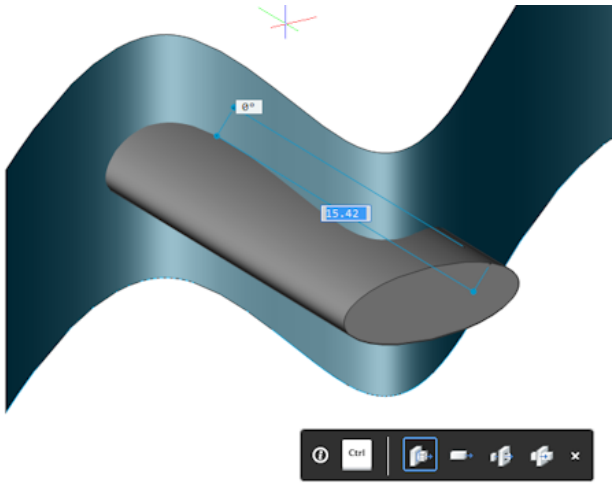


2 Place the cursor over the 2D closed entity, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

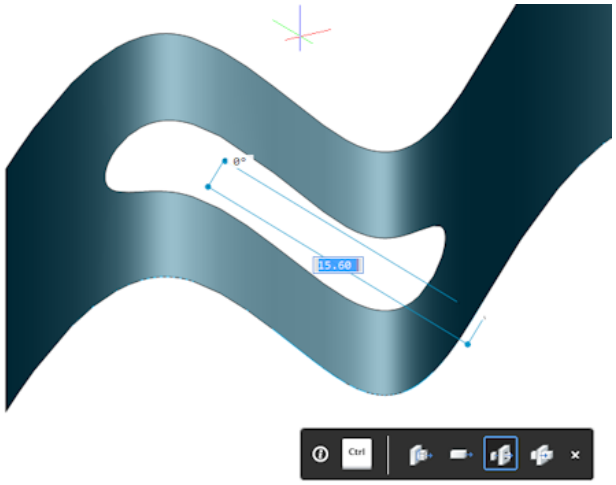
3 Extrude the 2D entity in the direction of the surface.

4 The default extrusion direction is perpendicular to the plane of the 2D entity.

The **Hotkey Assistant** (HKA) appears at the bottom of the screen.



5 Hit the CTRL key twice to select the **Subtract** option of the DMEXTRUDE command.



6 Click to create the opening.

9.4.5 Deleting features

The **Feature** option of the DMSELECT command allows to easily select all faces of a hole, depression or protrusion.

In the **Mechanical** workspace, the **Protrusion** and **Depression** tools are available in the **Select** panel of the **Solid** ribbon tab and the **Select** tab of the Quad.

The tools sit in the **Sheet Metal** toolbox. See **Available Tools** in the **Customize** dialog box.

To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib)

1 Click the **Protrusion** ( ) or **Depression** ( ) tool button in the **Select** panel of the **Solid** ribbon tab or from the **Select** command group of the Quad.

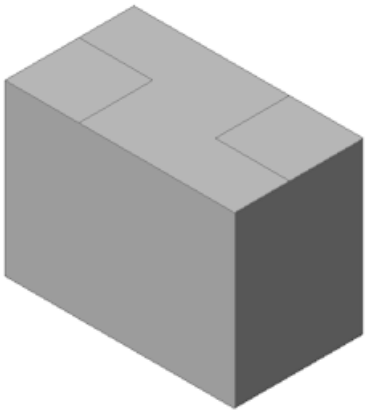
2 Select one or more faces of the feature.

3 All adjacent faces of the feature are detected.

4 Click the **Erase** ( ) tool button.

9.4.6 Removing edges between coplanar faces

When two or more faces are made coplanar, e.g. with the DMPUSHPULL command, it might be necessary to remove the shared edges using the DMSIMPLIFY command.

	
Two depressed faces	Two depressed faces moved to the top face, to create three coplanar faces.

1 +Launch the DMSIMPLIFY command.

1 **You are prompted**: Select entities to simplify or [Entire model/SEttings/selection options (?)] <Entire model>:

2 Select the solid, then press Enter.

3 The edges are removed.

9.4.7 Copying a Feature

The DMCOPYFACES command allows you to copy one or more features from a 3D solid (depressions - holes, grooves, and pockets - or protrusions - extrusions and ribs), to a new location on the same or a different 3D solid.

Make sure dynamic UCS is enabled.

1 Select all faces of the entire feature(s).

2 Click the **Copy Faces** ( ) tool:
- on the ribbon: **Solid** | **Edit** | **Copy Faces**.
- on the **Direct Modeling** toolbar.
- in the menu bar: **Model** | **Direct Modeling** | **Copy Faces**
- in the **Model** tab on the Quad

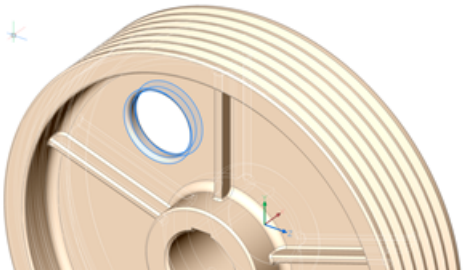
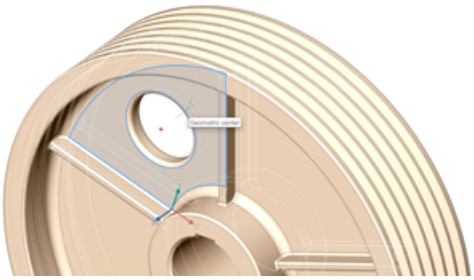
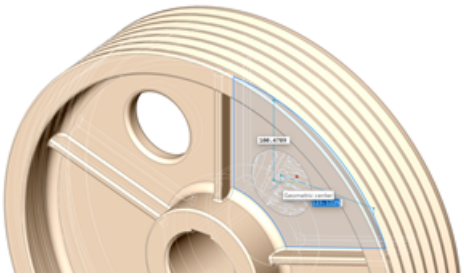
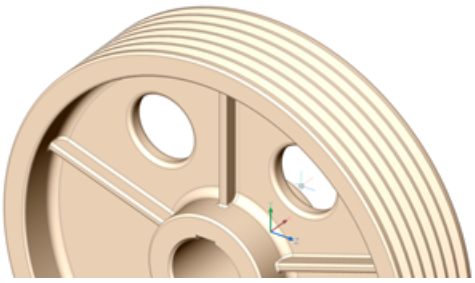
3 Specify the base point.

4 **You are prompted:** Select an insertion point or [Rotate/mUltiple/MOde]:

5 (Optional):
   - Use the **Rotate** option to rotate the faces around the Z axis of the UCS.
   - Use the **Multiple** option to insert the multiple copies of a feature.

6 Specify the insertion point.

7 The feature is copied from the base point to the insertion point with dynamic UCS.

	
Select the faces of the entire feature	Specify base point
	
Specify the insertion point	The feature is copied

9.4.8 Deformable modeling

Use the tools on the **Deform** toolbar, to deform a set of adjacent faces on a 3D solid or a surface, to create complex aesthetic shapes.

The faces are always deformed if:

- The external boundary (= edges, which are adjacent to only one of the selected faces) remains unchanged.
- The internal boundaries (= edges, which are adjacent to two of the selected faces) are deformed together with the faces.
- The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

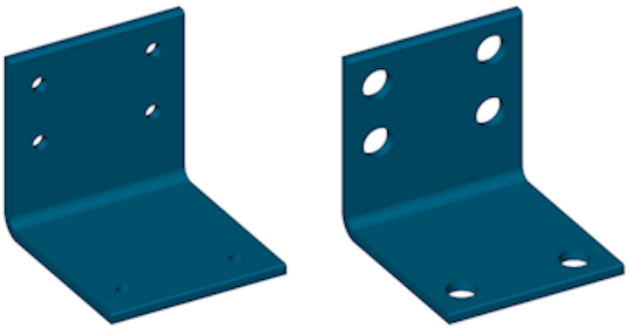
For a twist deformation, it is possible to specify the continuity between the deformed and the fixed portion of the entities.

Deformation can be defined in several ways:

- DMTWIST command: Twist the whole entity or a portion.
- DMDEFORMPOINT command: Move a point on a face in any direction in 3D space.
- DMDEFROMMOVE command: Move or rotate one or more edges.
- DMDEFORMCURVE command: Replace an edge with a given 3D curve.

9.5 Design intent recognition

Direct modeling operations allow you 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.



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.

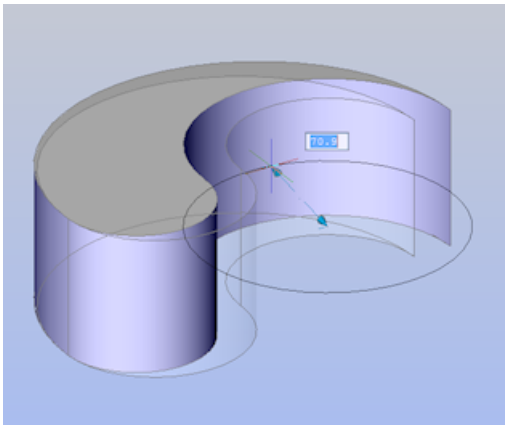
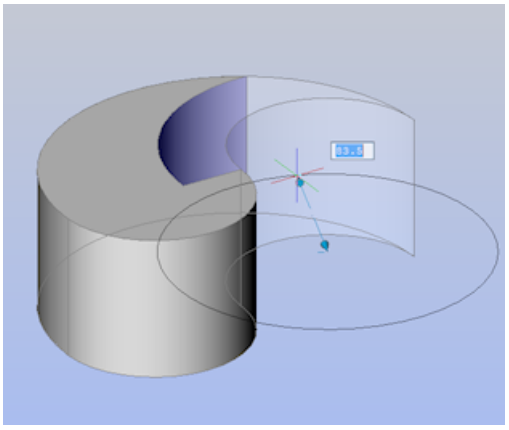
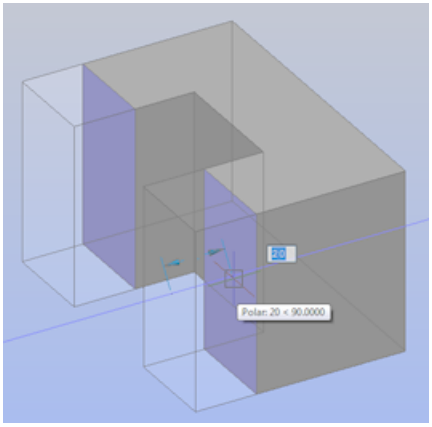
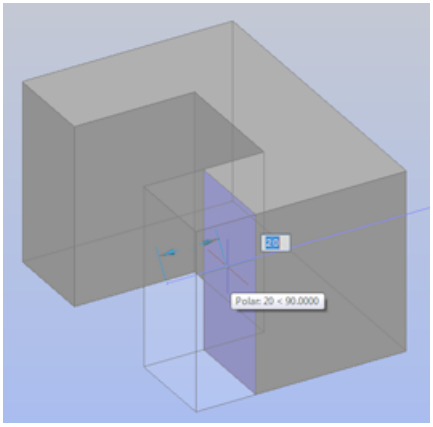
- tangent surfaces (planes, cylinders and cones) ()
- coincident planes ()
- parallel planes ()
- perpendicular planes ()
- cylinders perpendicular to planes ()
- coaxial surfaces (cylinders and cones) ()
- surfaces of equal radius (cylinders and spheres) ()
- vertices between 4 or more faces ()
- edges between coincident faces ()
- switch off/on all ()

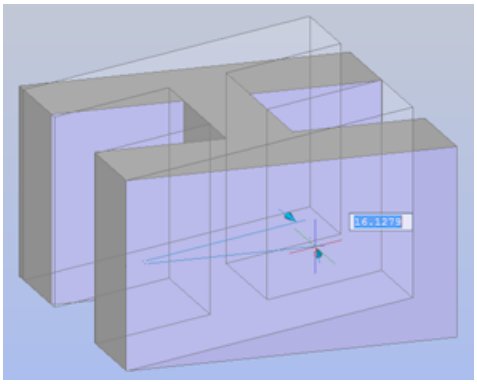
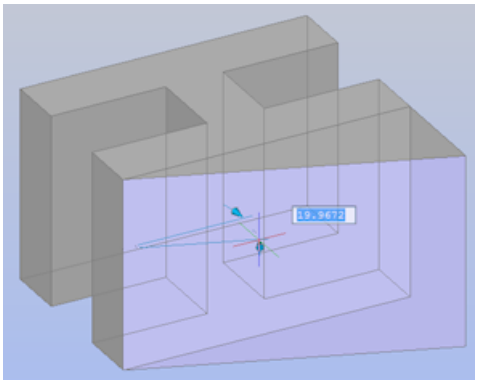
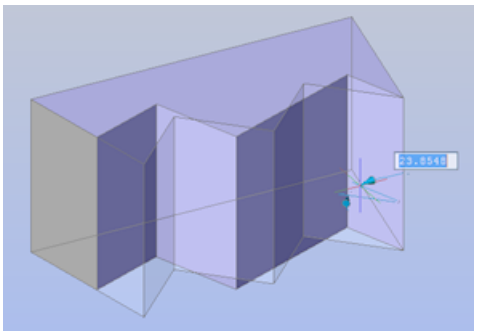
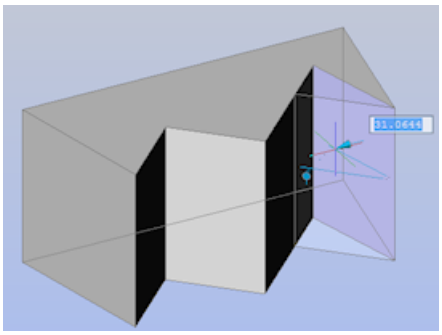
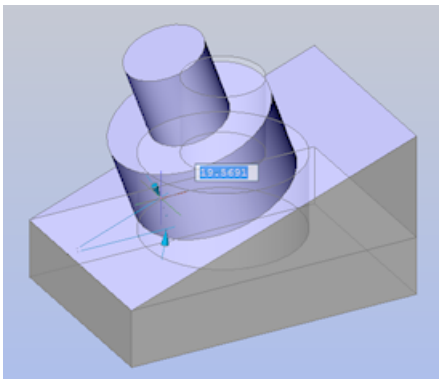
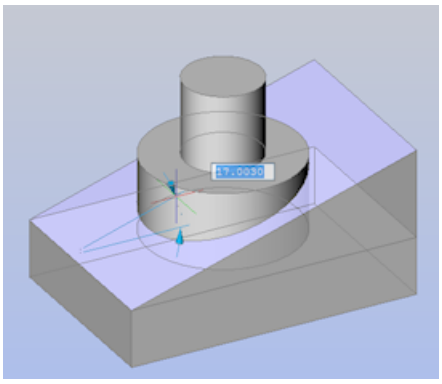
The design intent options can also be set by editing the DMRECOGNIZE system variable in the **Settings** dialog box.

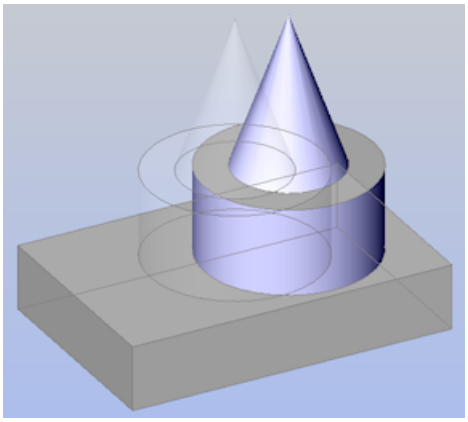
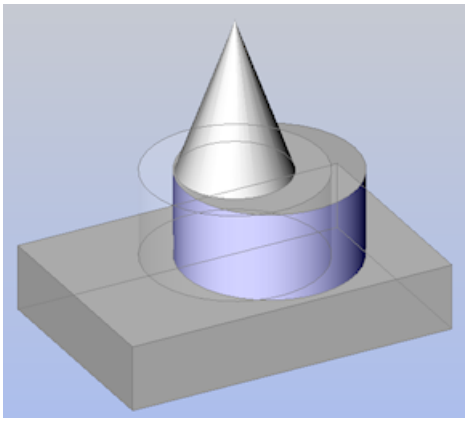
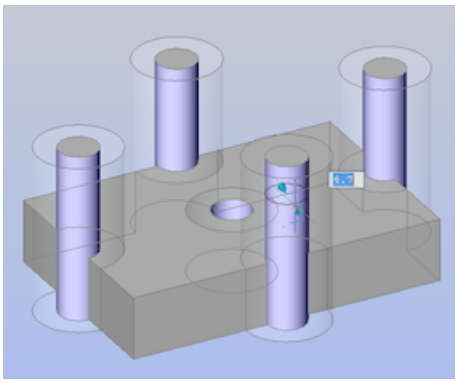
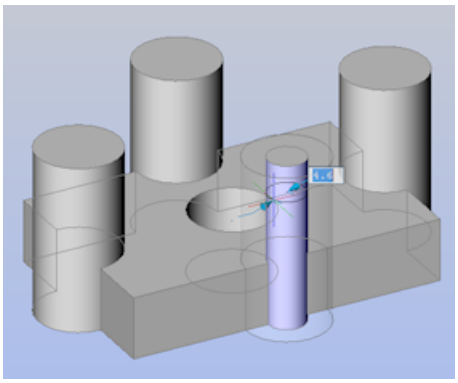
Click the **Options** tool button ( ) on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly.

Nota: 3D Constraints 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 illustrate the difference in the behavior whether automatic 3D geometry constraints are recognized or not.

	
tangent surfaces recognized ( )	tangent surfaces not recognized ( )
	
coincident plane recognized ( )	coincident planes not recognized ( )

	
parallel planes recognized ()	parallel planes not recognized ()
	
perpendicular planes recognized ()	perpendicular planes not recognized ()
	
perpendicular cylinders recognized ()	perpendicular cylinders not recognized ()

	
coaxial surfaces recognized ( )	coaxial surfaces not recognized ( )
	
equal radius recognized ( )	equal radius not recognized ( )

9.6 Working with sections

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 - Section Planes** dialog box.

The SECTIONPLANETOBLOCK command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Nota:

When using a BIM license level:

- The BIMSECTION command creates **BIM section** entities, which are Section entities with extended properties, such as type (plan, section, elevation) and initial scale of the 2D section result.

- The BIMIFY command can assign BIM properties to a Section entity.

9.6.1 Difference between Live Section and Clip Display

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed.

Live Section: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape.

Clip Display: the display of entities is clipped. Entities are not replaced by temporary versions. Clip Display allows to continue using all modeling and editing operations on all drawing content. As opposed to the Live Section status, the Clip Display status can be set for multiple section entities simultaneously.

9.6.2 Align a section plane to a face

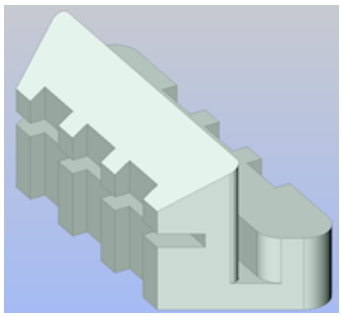
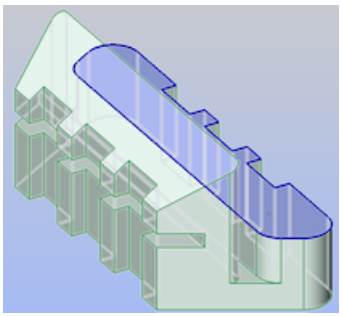
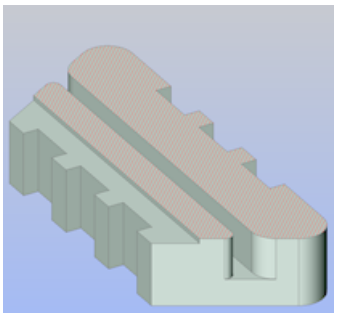
1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Select Face** option.

You are prompted: Select entity:

3 Select the face of a solid you want to align the section plane to.

		
Solid	Select face	Section plane aligned to face

4 The section plane is aligned to the selected face.

5 The **Clip Display** property of the section entity is switched **On** automatically.

6 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.3 To create a vertical section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

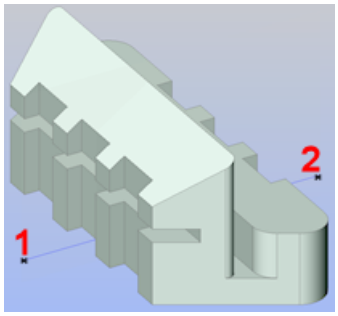
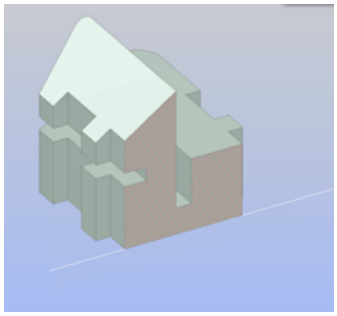
2 Specify two points.

3 The section plane is a vertical plane created through these points.

The part of the solid at the right hand side (with respect to the draw order of the points) of the section

plane is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

	
Specify two points	Vertical section plane

4 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.4 To create a horizontal section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

Nota: If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

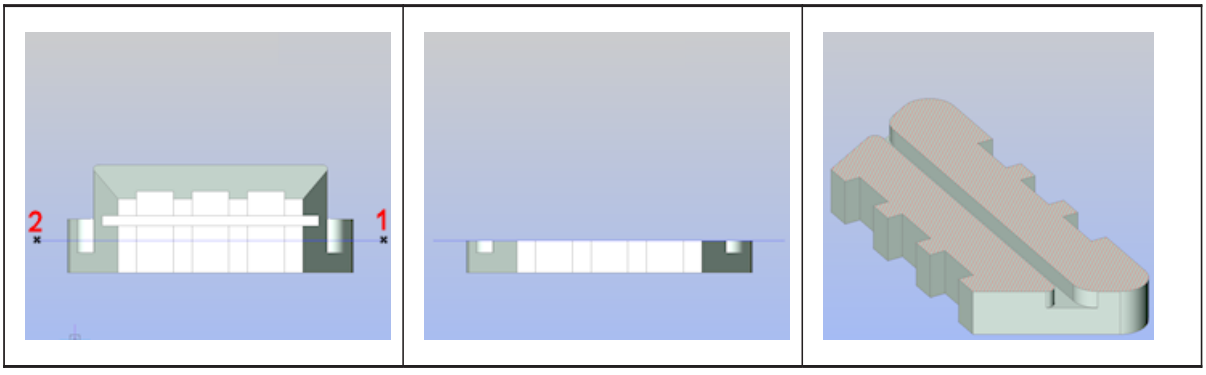
5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Specify two points.

7 The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.

8 The part of the solid at the right hand side (with respect to the draw order of the points) of the section plane is cut away.

9 The **Clip Display** property of the section entity is switched **On** automatically.



Specify two points	The part of the solid at the right hand side of the section plane is cut away	Horizontal section plane

10 (option) Edit the properties of the section plane.

9.6.5 To create a vertical jogged section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Draw** option.

You are prompted: Specify start point:

3 Specify the first point (1).

4 **You are prompted**: Specify next point:

5 Specify the second point (2):

6 **You are prompted**: Specify next point or ENTER to finish:

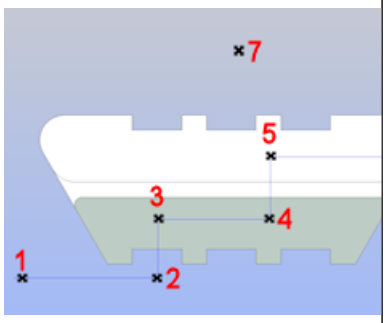
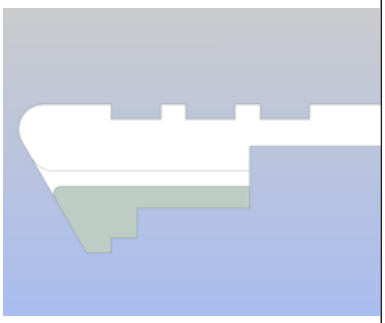
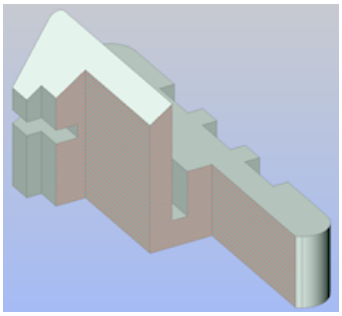
7 Specify the other points (3 - 6), then right click or press Enter to stop.

8 **You are prompted**: Specify the direction of the section entity:

9 Click a point (7) to specify the direction of the section entity.

10 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.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Vertical jogged section plane

11 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.6 To create a horizontal jogged section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

Nota: If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Choose the **Draw** option.

You are prompted: Specify start point:

7 Specify the first point (1).

8 **You are prompted**: Specify next point:

9 Specify the second point (2).

10 **You are prompted**: Specify next point or ENTER to finish:

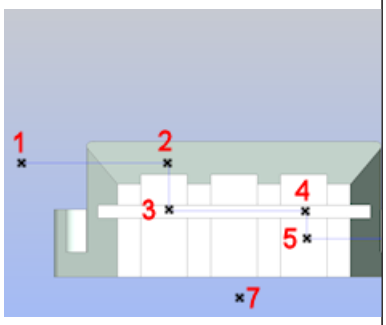
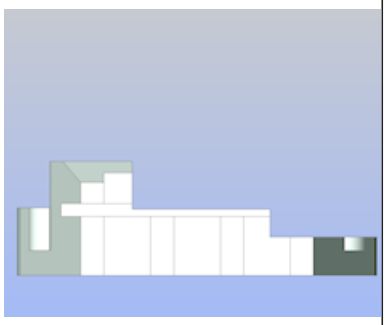
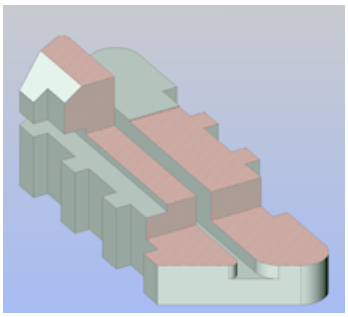
11 Specify the other points (3 - 6) , the right click or press Enter to stop.

12 **You are prompted**: Specify the direction of the section entity:

13 Click a point (7) to specify the direction of the section entity.

14 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.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Horizontal jogged section plane

15 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.7 To create an orthographic section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Orthographic** option.

You are prompted: Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:

3 Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the Command line.

4 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.

5 (option) Edit the properties of the section plane in the **Properties** panel.

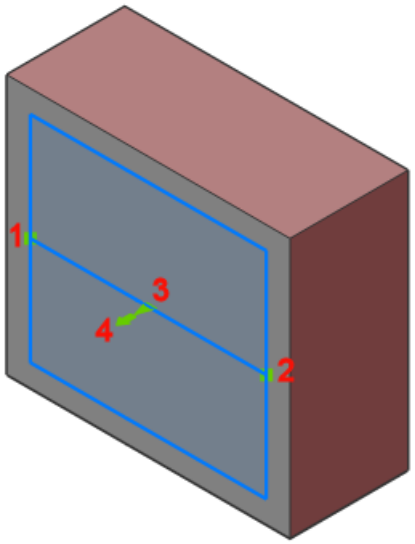
9.6.8 To modify the size and position of a section plane

1 Select the section plane entity in the drawing.

1 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 in the **Properties** panel to see the result of the modifications dynamically.

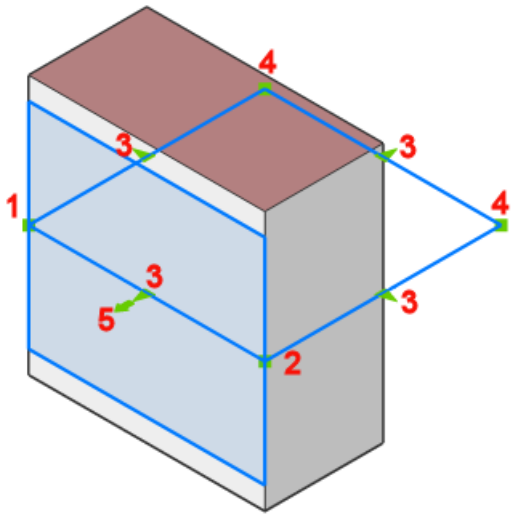
3 (option) Set the **State** property to **Plane**:



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.

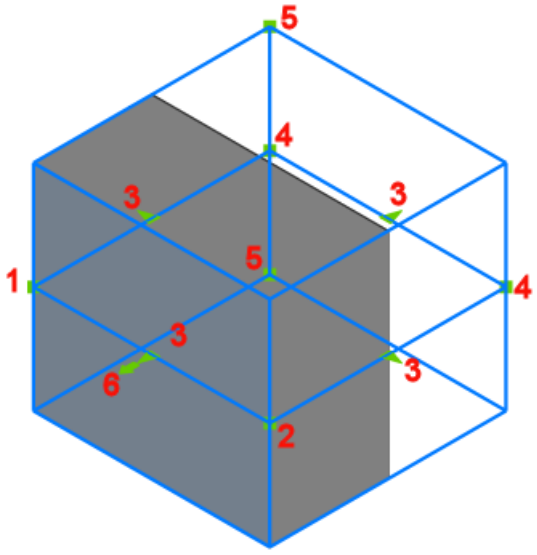
4 (option) Set the **State** property to **Boundary**:



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 ( )

5 (option) Set the **State** property to **Volume**:



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 must be on to move grip 5.
- 6 (): click to flip the view direction of the section plane.

9.6.9 To save a section

The SECTIONPLANETOBLOCK command allows to:

- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

9.7 Create a TIN Surface

9.7.1 Commands

TIN, TINASSIGNIMAGE, TINPROJECT

9.7.2 Overview

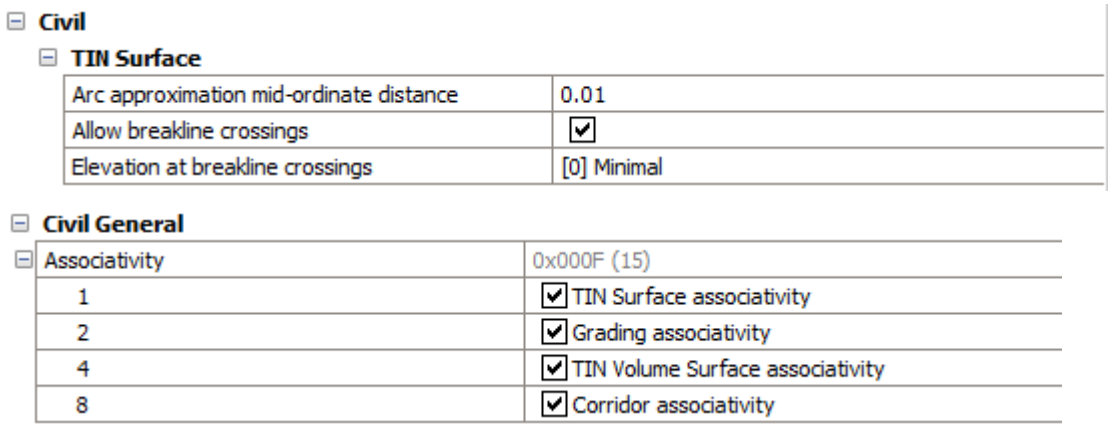
In BricsCAD, topographical surfaces are created using the TIN Surface tool (triangulated irregular network) or the TIN command.

A TIN surface uses two types of entry data: points and breaklines (polylines).

- 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 based on 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 or circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries on a large 3D surface improves the performance of a TIN Surface plan.

Creating a TIN Surface is a process that needs to be followed to map 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, created from Civil 3D surface.

9.7.3 Setting the TIN Surface preferences



Arc approximation mid-ordinate distance

Nota: 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 BricsCAD predefined templates. For instance, the distance value is set to 0.01 in BIM-m, 1in BIM-cm, 10inBIM-mm, and 0.4in BIM imperial templates.

Allow breakline crossings

Defines if crossings 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 breaklines elevation at the intersection.

Associativity

Enable/disable grading, surface and corridor associativities.

By default, the system variable is set to 15.

0: Neither TIN surface, grading nor corridor associativity is enabled.
1: Enables TIN surface associativity.
2: Enables grading associativity.
3: Both TIN surface and grading associativities are enabled.
4: Enables TIN volume surface associativity.
5: Both TIN surface and TIN volume surface associativities are enabled.
6: Both grading and TIN volume surface associativities are enabled.
7: TIN Surface, Grading and TIN volume surface associativities are enabled.
8: Enables corridor associativity.
9: Both TIN surface and corridor associativities are enabled.
10: Both grading and corridor associativities are enabled.
11: TIN surface, grading and corridor associativities are enabled.
12: Both TIN volume surface and corridor associativities are enabled.
13: TIN surface, TIN volume surface and corridor associativities are enabled.
14: Grading, TIN volume surface and corridor associativities are enabled.

15: All TIN surface, grading, TIN volume surface and corridor associativities are enabled.

9.7.4 Properties of a TIN Surface

General	
Handle	93
Color	Green
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Material	ByLayer
Data	
Name	Site
Description	School playground
TIN Visibility	
Boundary line	Off
Points	Off
Triangles	On
Contours	Off
TIN Contours	
Major contours interval	10 m
Minor contours interval	1 m
Major contours color	Red
Minor contours color	Green
TIN Statistics	
Number of points	4264
Number of triangles	8230
Minimum elevation	4.4 m
Maximum elevation	18.9192 m
2D area	10363.7386 m²
3D area	11990.9728 m²

TIN Visibility

Boundary line
Change boundary line visibility.

Points
Change points visibility.

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.

TIN Contours

Major contours interval
Sets the major contours interval.

Minor contours interval
Sets the minor contours interval.

Major contours color
Defines the color of the major contour.

Minor contours color
Defines the color of the minor contour.

TIN Statistics

The number of points
Displays the number of points that define the TIN surface.

The number of triangles
Displays the number of non-overlapping triangles in the TIN surface.

Minimum elevation
Displays the minimum elevation value found in the TIN surface.

Maximum elevation
Displays the maximum elevation value found in the TIN surface.

2D area
Displays the 2D area calculated from the flat TIN surface.

3D area
Displays the 3D area that is calculated from the TIN surface.

9.7.5 Procedure: Creating a TIN Surface from a selection set of points and polylines

1 Open the drawing file that contains a set of points and polylines.

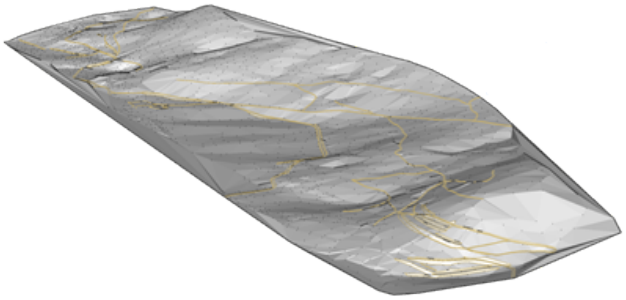


2 Launch the TIN command.

3 Select the entities in the drawing area to create a toposurface.

4 Press ENTER.

The toposurface is created from selected entities. Points are added to the surface as points, 3D polylines are added to the surface as breaklines.



9.7.6 Procedure: Creating a TIN Surface from an imported points file (.txt or .csv)

1 Launch the TIN command.

2 Select **Import from file** in the Command line.

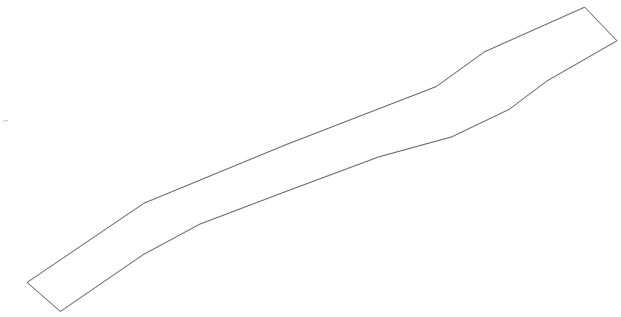
3 Choose the point data file you want to open or choose multiple files to import all at once.

4 Click Open to import the toposurface.

Nota: If the toposurface is not shown in the current viewport, use zoom extents to display it to the extents of the entities.

9.7.7 Procedure: Creating a TIN Surface of a selected area using Clip polygon

1 Open the drawing file that contains closed polyline.



2 Launch the TIN command.

3 Select the **cLip polygon** option in the Command line.

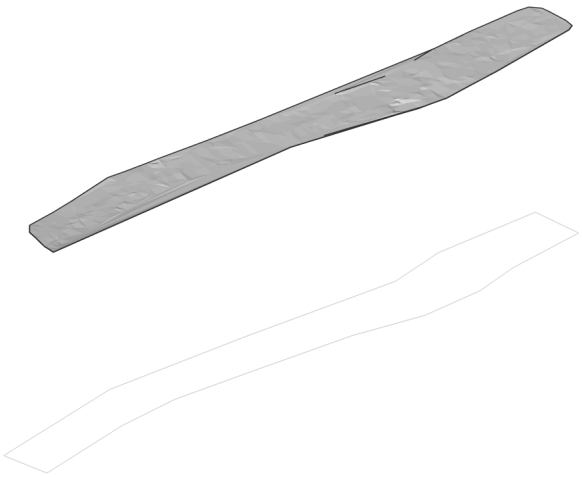
4 Select the closed polyline to use.

5 Specify if the crossed breaklines should be removed.

Nota: If the surface we are going to make contains breaklines, we define what should happen to the breaklines crossing the clipped polygon.

6 Select any of the options, **Import from file**, **Create from Faces**, **Create from civil 3D surface**.

Nota: Clip polygon works as a projection on the TIN surface. The TIN surface is created only in the area defined by the clip polygon.



9.7.8 Procedure: Creating a TIN Surface by placing points

1 Open a drawing.

2 Launch the TIN command.

3 Select the **place Points** option in the Command line.

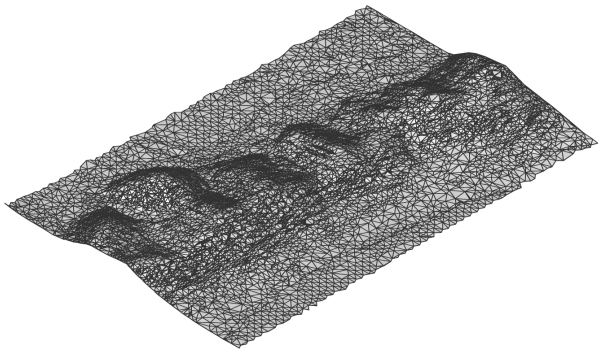
4 Pick a point where you want to start creating a TIN surface.

Specify the elevation value of each of those points, then press ENTER to accept.

Nota: At least three points are needed to create a TIN surface.

9.7.9 Procedure: Creating a TIN Surface from Faces

1 Open a drawing file that contains 3D faces.



2 Launch the TIN command.

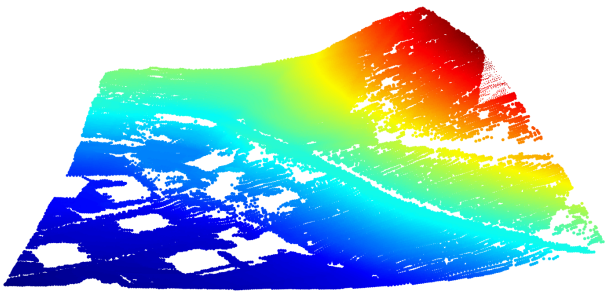
3 Select the **create from Faces** option in the Command line.

4 Select desired 3D faces.

5 Press Enter.

9.7.10 Procedure: Creating a TIN Surface from a Point Cloud

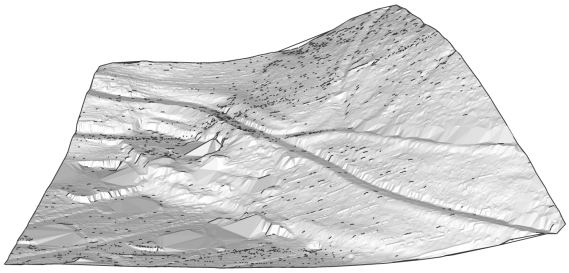
1 Open the drawing file that contains a point cloud.



2 Launch the TIN command.

3 Select the point cloud in the drawing area to create a toposurface.

4 Press ENTER. The toposurface is created from the point cloud.



9.7.11 Procedure: Creating a TIN Surface from a Civil 3D surface

1 Open a drawing with a Civil 3D surface.

2 Launch the TIN command.

3 Select **Create from civil3d surface**.

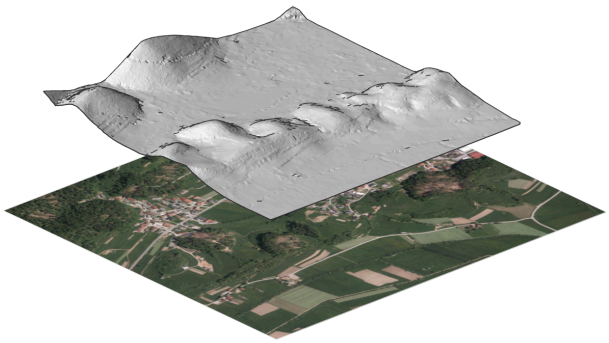
4 Select the Civil 3D surface. A TIN surface is created.

9.7.12 About TIN Assign Image

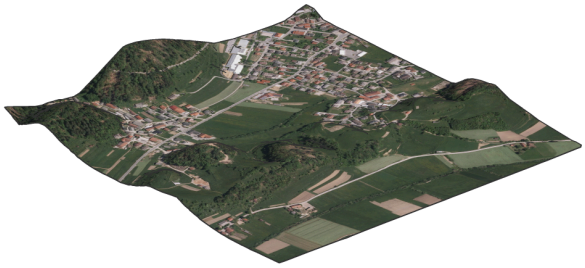
1 Open the drawing file that contains a TIN surface and an image.

2 Launch the TINASSIGNIMAGE command.

3 Select the TIN surface.



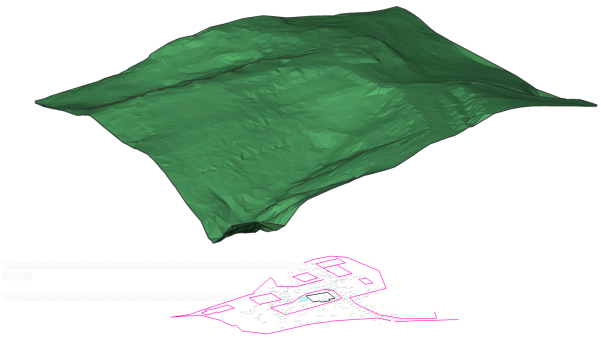
4 Select the raster image.



Nota: If the image is not shown on the TIN surface, change the Visual Style to Modeling or Realistic and set TIN surface triangles to ON.

9.7.13 About TIN Project

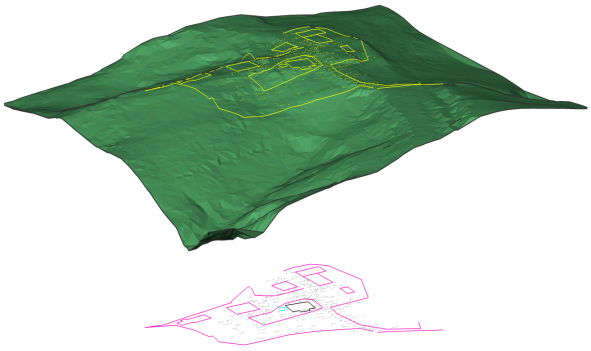
1 Open the drawing file that contains the TIN surface and the entities (points, blocks, text, lines, polylines, circles…).



2 Launch the TINPROJECT command.

3 Select the entities to project.

4 Select if the original entities are to be erased.



9.8 TIN Surface Edit

9.9 TIN Surface Direct Modeling

9.9.1 Commands

TINEXTRACT, TINMODIFY, TINMERGE

9.9.2 Overview

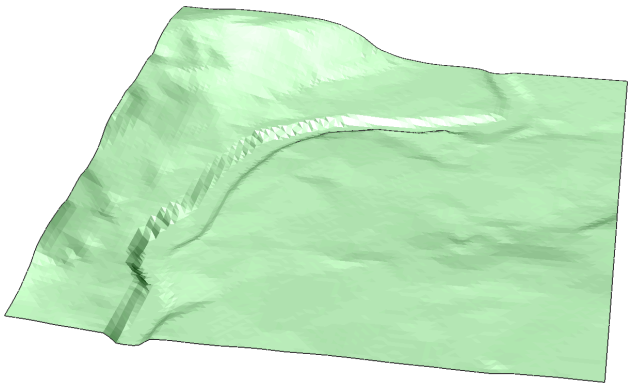
In BricsCAD, topographical surfaces created as a TIN Surface, can be adjusted using different tools.

9.9.3 About TINEXTRACT

The TINEXTRACT command is used to generate a solid, mesh, border, faces and points from a TIN surface. You can exchange these entities in other applications, to further analyze the surfaces. The solids or meshes can be created in different ways: as a vertical offset of the TIN Surface, as a model between two TIN Surfaces or as a model between a fixed elevation plane and the TIN Surface.

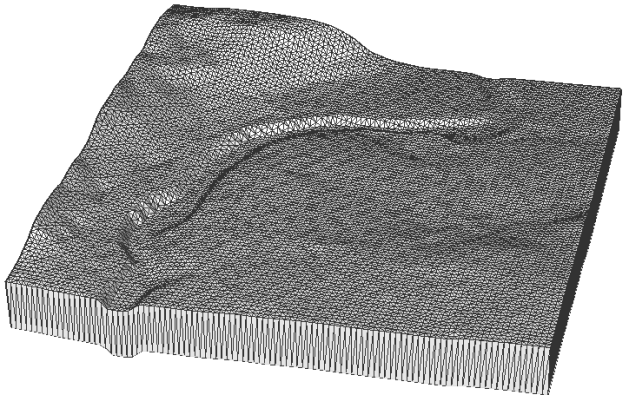
9.9.4 Procedure: Creating a solid from a TIN surface using option Elevation

1 Select a TIN Surface and type TINEXTRACT in the Command line.



2 Select the **Solid** option and press Enter.

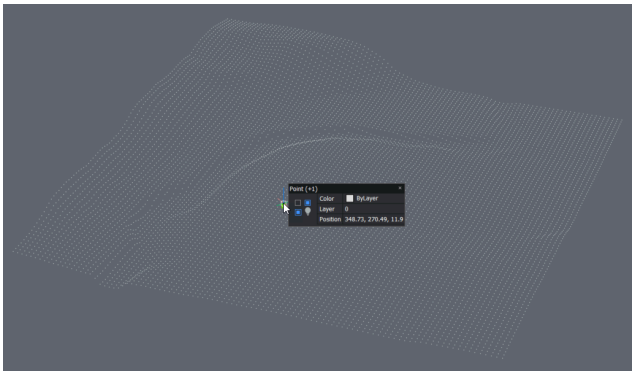
3 Choose the elevation height and press Enter.



9.9.5 Procedure: Creating Points from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

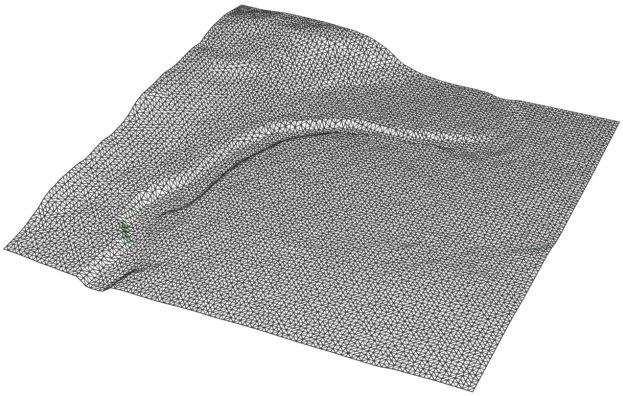
2 Select the **Points** option in the Command line.



9.9.6 Procedure: Creating Faces from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Faces** option.

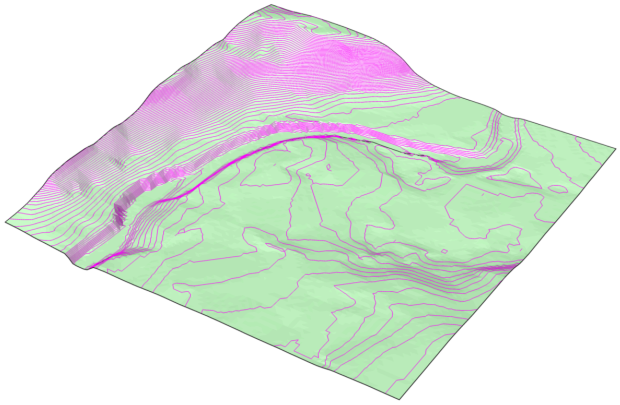


9.9.7 Procedure: Creating a Contours from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Contours** option in the Command line.

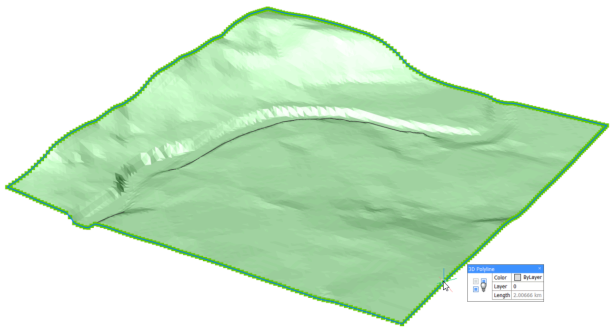
3 Type in **A** for All and press Enter.



9.9.8 Procedure: Creating a Border from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Border** option in the Command line.

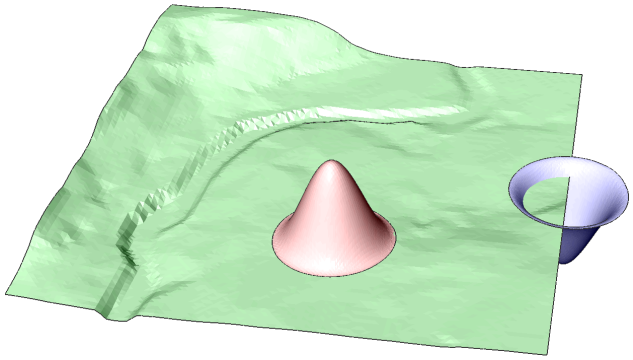


9.9.9 About TIN Merge

Different TIN Surfaces can be merged into one by using the TINMERGE command. A merge is executed between a base surface and one or more surfaces to merge with.

9.9.10 Merging TIN Surfaces

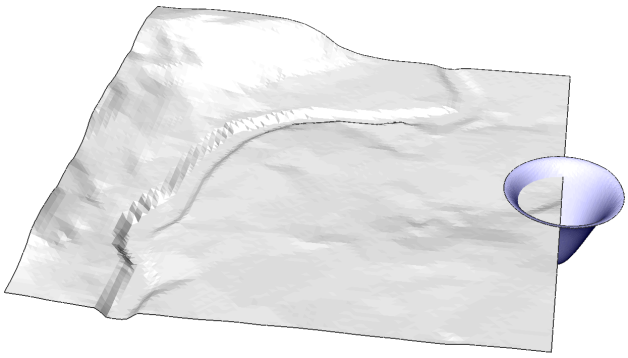
1 Type TINMERGE in the Command line.



2 Choose a base surface. In this example, we choose the green surface as the base surface.

3 Choose to keep or delete the original surfaces.

Nota: Selecting a different baseline, results in a different merge. It is important to remember that the surface to merge replaces the part of the base surface that it covers.



The first example shows the merge of the green surface as the base surface and the blue and red surface.

The second surface shows the merge of the blue surface as the base surface and the green surface.

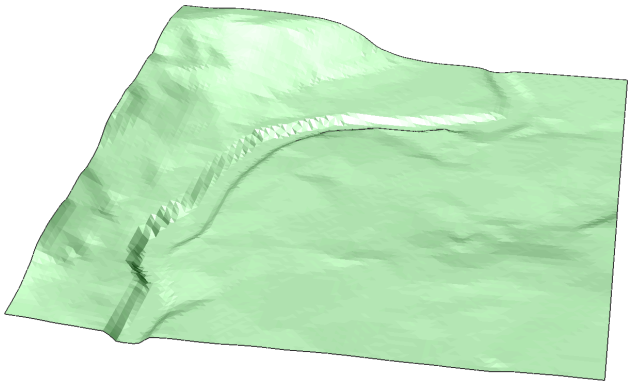
The third surface shows the merge of the red surface as the base surface and the green surface.

9.9.11 About TIN Modify

A TIN Surface can be modified by deforming or smoothening the surface. By deforming the surface, the TIN Surface is adjusted following the contours of an area defined by the user. By smoothening the surface, a selected area is leveled which is done by interpolation of this predefined area.

9.9.12 Procedure: Adding deformations on a TIN Surface using Radius

1 Select a TIN Surface and type TINMODIFY in the Command line.



2 Select the **Deform** option.

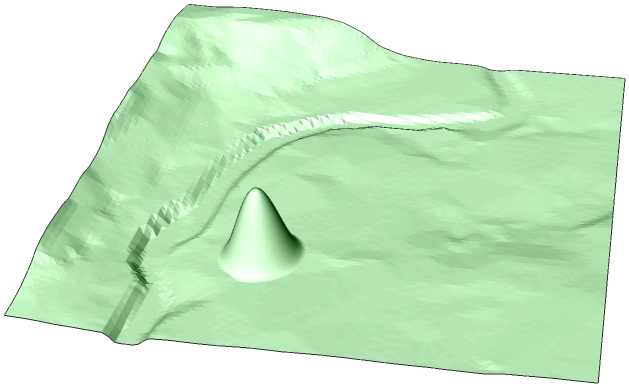
3 Choose a deformation radius.

4 Click on the area to apply the deformation.

5 Choose the elevation height.

6 Choose whether you want to add the adjustment as a new element or as a modification of the existing

TIN Surface.



Nota: By adding the modification as a new element, the structure of the original TIN Surface isn't modified but added as a new TIN surface.

7 Click on other areas to add new deformation or press ENTER to end the command.

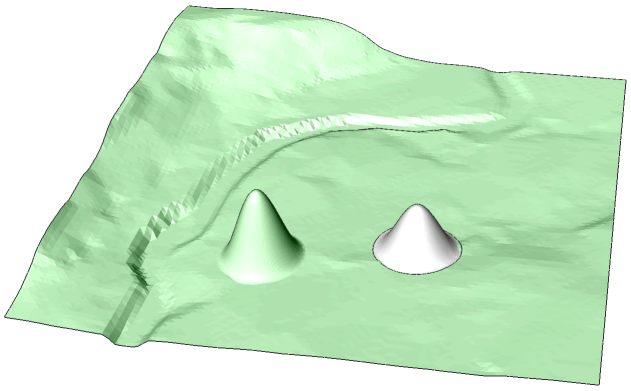
9.9.13 Procedure: Adding deformations on a TIN Surface using specify Contour

1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select the **specify Contour** option.

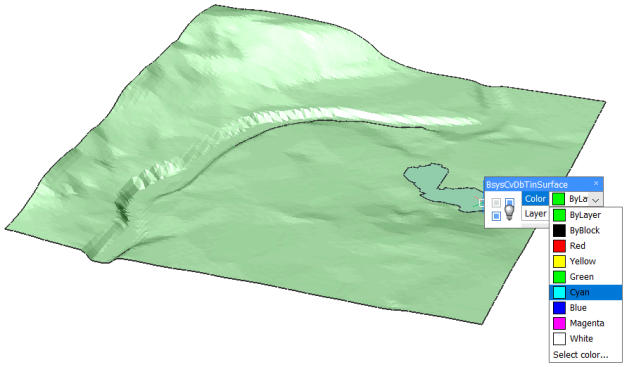
4 Pick a contour on the TIN Surface.



5 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for the adjustment.

6 Click on other contours to add new deformation or press ENTER to end the command.

7 You have now created a new element on your TIN Surface. You can change its properties as you like. For example: change the color of the element.



9.9.14 Procedure: Adding deformations on a TIN Surface using select Entity

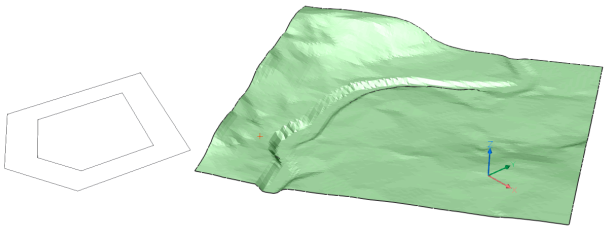
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

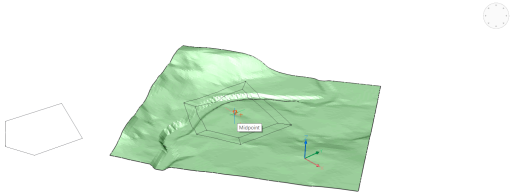
3 Type E and hit ENTER to choose **Select Entity**.

4 Select a closed entity (e.g. Polyline).

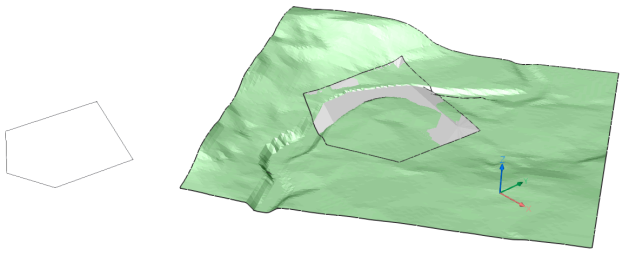
5 Choose an offset distance.



6 Click on the area to apply the deformation.



7 Choose the elevation height.



8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a New element for your adjustment.

9 Click on other areas to add new deformation or press ENTER to end the command. You have now added a deformation using **select Entity**.

9.9.15 Procedure: Adding deformations on a TIN Surface using draw Polygon

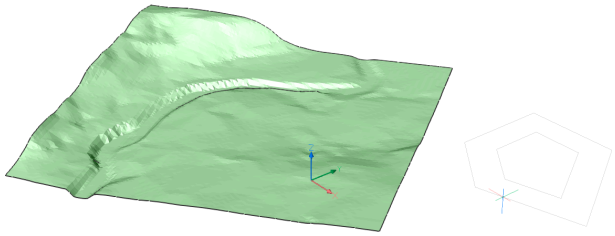
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select **draw Polygon**.

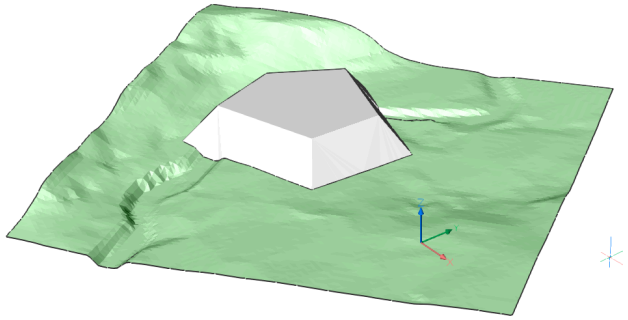
4 Draw a polygon, hit ENTER to close the polygon.

5 Choose an offset distance.



6 Select the area where to apply the deformation.

7 Choose the elevation height.

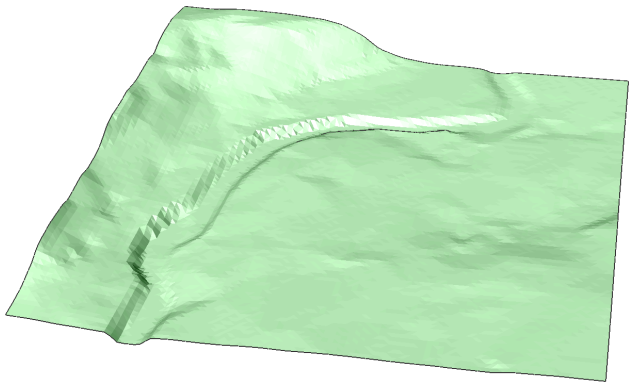


8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for your adjustment.

9 Click on other areas to add new deformation or press Enter to end the command. You have now added a deformation using draw polygon.

9.9.16 Procedure: Smoothening a TIN Surface

1 Select a TIN Surface and type TINMODIFY in the Command line.

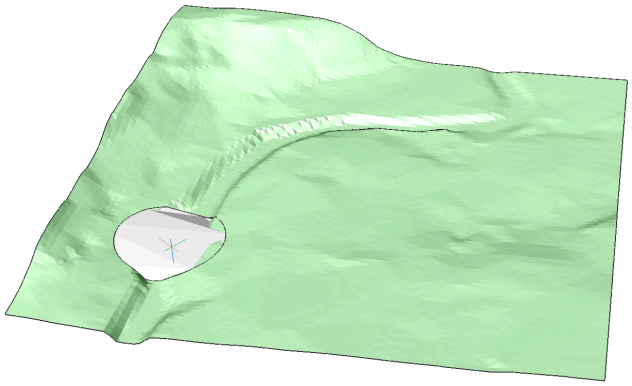


2 Select the **Smoothen** option.

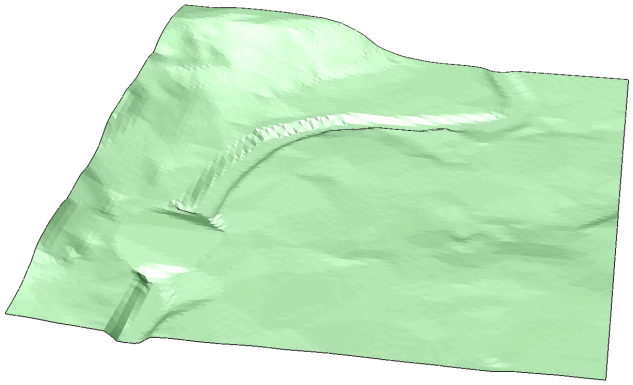
3 Choose a radius.

Nota: The procedure for **select Entity** or **draw Polygon** is analogue to **Adding deformations using select Entity** and **Adding deformations using draw Polygon** starting from step 3.

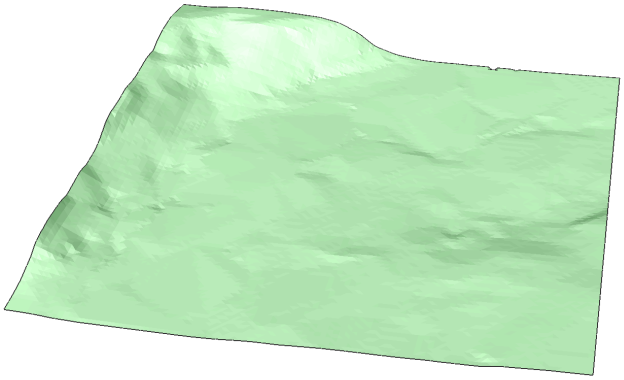
4 A preview of the intervention is shown.



5 Click to execute the command.



6 Click on other areas to smoothen or press ENTER to end the command.



9.10 Analyzing TIN Surfaces

9.10.1 Commands

TINVOLUME, TINWATERDROP

9.10.2 Overview

In BricsCAD, topographical surfaces created as a TIN Surface, can be analyzed using different tools.

9.10.3 About TIN Volume

A TIN volume surface provides an exact difference between the base and comparison surfaces. Therefore, the Z-value of any point in the volume surface is precisely the difference between the Z of the comparison surface at that point and the base surface at that point. A TIN Volume Surface is drawn as a custom entity with additional fill/cut volume properties.

You can use TIN Volume to calculate or analyze earthworks:

- Calculate fill/cut volumes inside bounding area.
- Draw zero contours which divide fill and cut areas.
- Visualize the mass distribution.

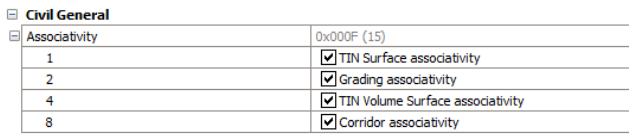
9.10.4 Procedure: Creating a TIN Volume

In the following procedures, a TIN Volume is created. Choose whether the volume is associative or not. If associativity is turned on: TIN Volume Surface is automatically updated if base or comparison surfaces are changed. To disable or enable the associativity, follow these steps:

1 Type **Settings** in the Command line and press Enter.

2 Go to **Civil** > **Civil General** > **Associativity**.

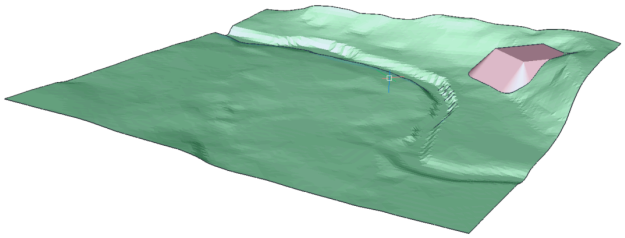
3 Check or uncheck **TIN Volume Surface Associativity**.



9.10.5 Creating a TIN Volume using a comparison TIN Surface

1 Launch the TINVOLUME command.

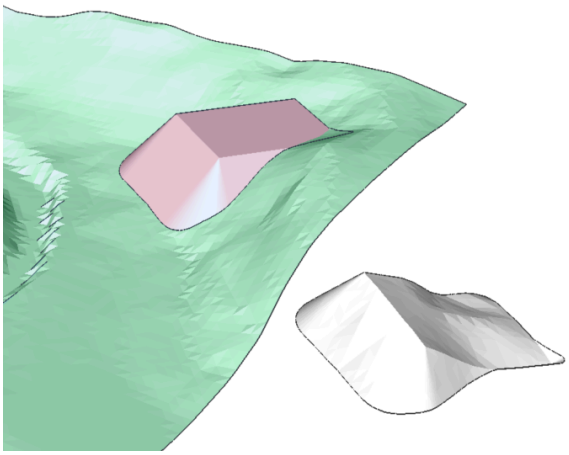
2 Choose a base surface. In this example, we choose the green TIN surface as the base surface.



3 Choose a comparison surface. In this example, we choose the red grading as the comparison surface.

4 Select a bounding area as the area to apply the operation orpress ENTER. The outer boundary of the surface will be used as a boundary.

5 The TIN Volume appears under the TIN Surface.



9.10.6 Creating a TIN Volume using Elevation

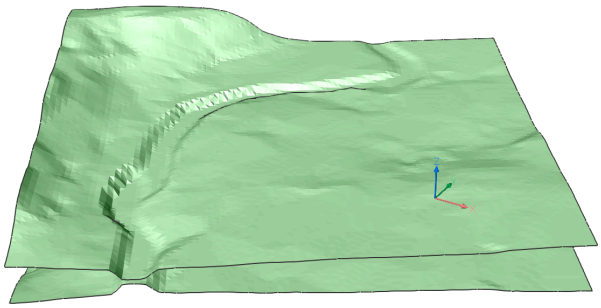
1 Launch the TINVOLUME command.

2 Choose a base surface.

3 Select **Elevation** in the Command line.

4 Choose the elevation height.

5 Select a bounding area as the area to apply the operation or press ENTER. The outer boundary of the surface will be used as a boundary.



9.10.7 About TIN Water Drop

The TINWATERDROP command creates water drop paths in real-time by moving the cursor over a TIN Surface. Water path is created as a 3D polyline in the current position if you click the mouse button.

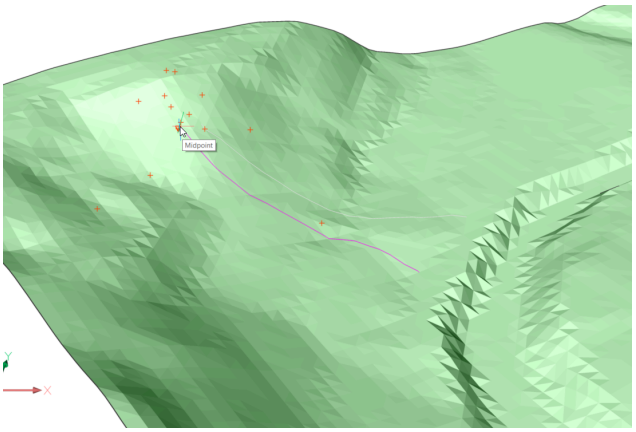
9.10.8 Procedure: Creating a water drop path

1 Launch the TINWATERDROP command.

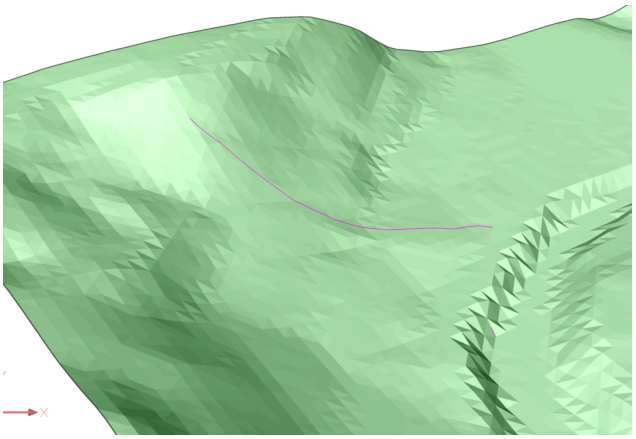
2 Select the TIN surface.

3 Move the cursor over the surface, a line representing the drop path is shown in real-time.

Nota: It would be better if the water drop path is colored so it could be seen clearer. To do so, you should change the current layer to another color, e.g. red.



4 Left click with the mouse to create a 3D polyline of the water drop path.



9.11 Working with Gradings

9.11.1 Commands

GRADING, GRADINGBALANCE

9.11.2 Overview

In BricsCAD, graded surfaces are created using the Create Grading tool.

Grading is a set of processes to maintain a base level for construction works or TIN surface design by defining a slope or adjusting the offset of a grading. For instance, creating parking areas, recreation reserves or road and railway designs.

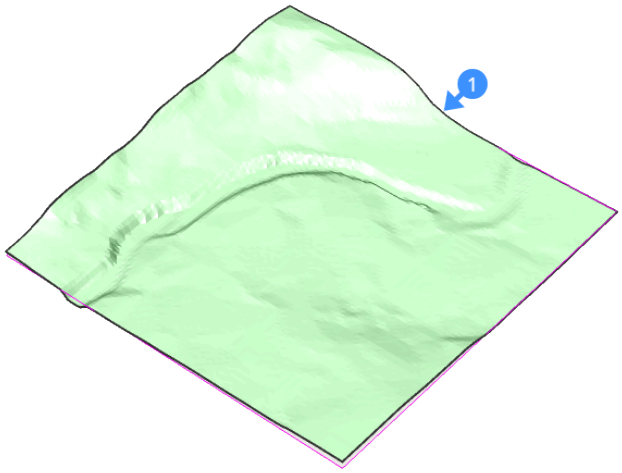
Two projection methods **Slope to surface** and **Offset to slope** are supported:

- **Slope to surface** needs an input entity (e.g., polyline, line, arc, circle), a surface and slope to grade the surface.
- **Offset to slope** needs an input entity, offset and slope.

Graded surfaces are not editable with the TINEDIT command.

9.11.3 Procedure: Grading a surface to create a pond

1 Open a drawing that has a topographical surface (1).

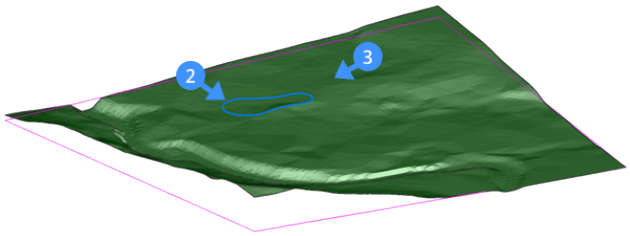


2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your pond.

Nota: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

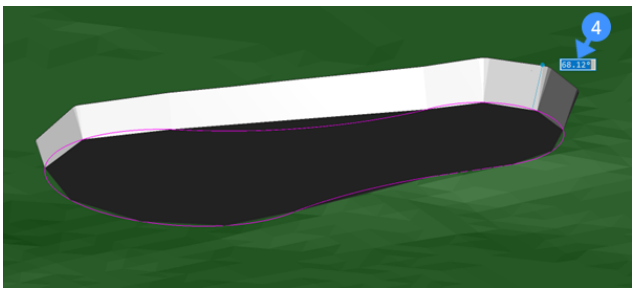
4 Select the target surface.



5 Define the slope of the grading.

Nota: Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Nota: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (4).



6 Press ENTER to accept the slope of the grading.

9.11.4 Displaying the properties of a graded surface

The properties are displayed in the Properties Panel after selecting the grading.

Data	
Grading type	Slope to surface
Target surface	BB
Name	Grading
Description	
Grading settings	
Cut slope	45
Fill slope	45
Side	Right
Infill	On
Boundary	Off
Grading statistics	
Fill volume	99.46
Cut volume	0.00
Net volume	99.46
TIN Visibility	
Boundary line	Off
Points	Off
Triangles	On
Contours	Off
TIN Contours	
Major contours interval	10
Minor contours interval	1
Major contours color	Red
Minor contours color	Green
TIN Statistics	
Number of points	389
Number of triangles	391
Minimum elevation	348.61
Maximum elevation	350
2D area	182.79
3D area	195.51

Grading type
Displays which projection method is used to grade the surface.

Target surface
Displays the handle number of the target surface

Cut slope
Displays the cut slope.

Fill Slope
Displays the cut slope.

Side

Controls on which side of the input entity the grading is drawn. Left or Right.

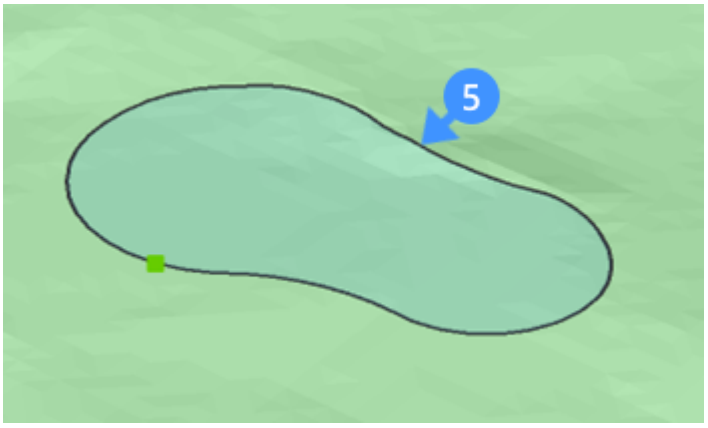
Infill

The internal are of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

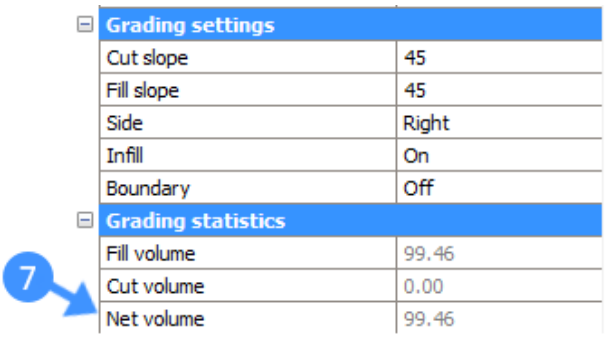
Boundary

Controls the boundary of the grading. When the grading boundary is on, that boundary is added as a **hide** boundary to the target surface.

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).







Fill volume
Displays the fill volume for a body/shape limited with the target surface and graded surface.

Cut volume
Displays the cut volume for a body/shape limited with the target surface and graded surface.

Net volume
Displays the net volume for a body/shape

9.11.5 Procedure: Grading a surface to create a platform

1 Open a drawing that has a topographical surface (8).



2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your platform (9).

Nota: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

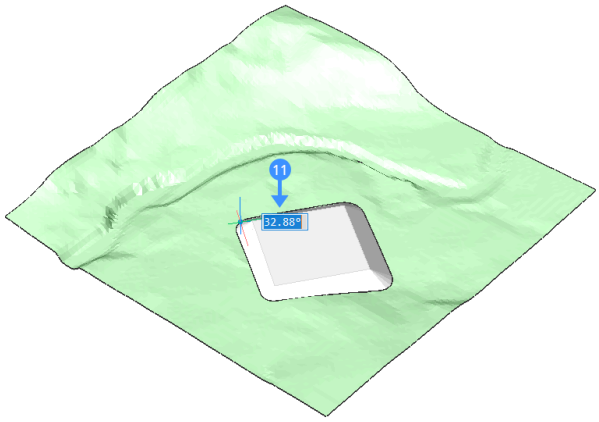
4 Select the target surface (10) to create a graded area.



5 Define the slope of the grading.

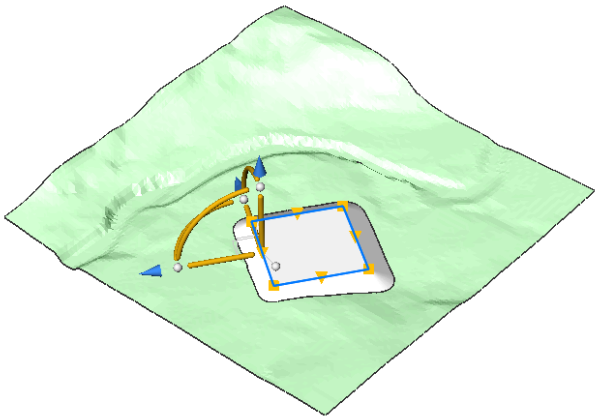
Nota: Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Nota: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (11).

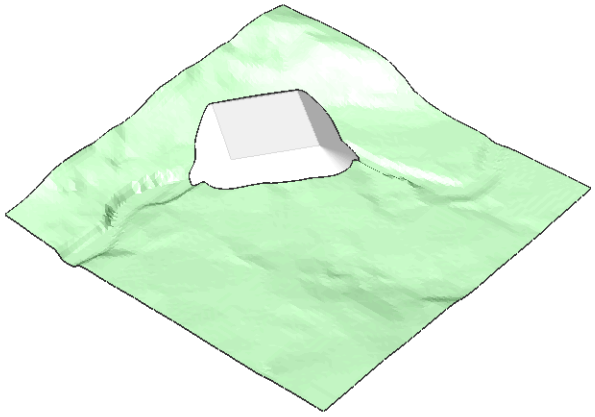


6 Press ENTER to accept the slope of the grading.

Nota: You can easily move the platform that you created. Do this by using the manipulator on the entity from which you created the platform. You can move both horizontally and vertically.



The grading will automatically be adapted to its surroundings.

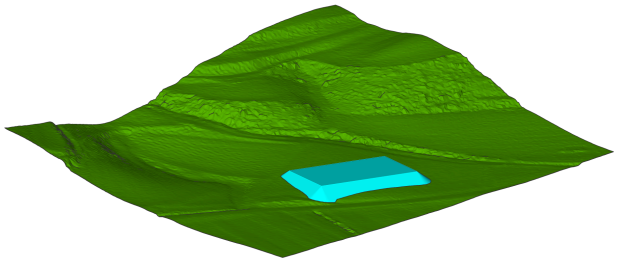


9.11.6 About Balance Grading Volumes

The GRADINGBALANCE command allows you to balance cut and fill volumes within the specified tolerance. This is achieved by raising/lowering the elevation of the grading input entity and the result is grading that has a net volume around zero, depending on the set tolerance.

9.11.7 Procedure: Adjusting the elevation of a grading to balance cut and fill volumes

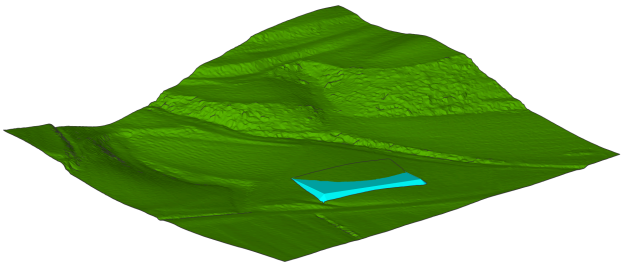
1 Open a drawing that contains a TIN surface and a grading representing a building pad.



2 Launch the GRADINGBALANCE command.

3 Select the grading.

4 Enter the tolerance in percent (1-100). 1 means that the difference between the cut and fill volume will be less than one percent and 100 means that the difference between the cut and fill volume will be less than 100 percent.



In the Properties panel you see that the cut and fill volumes are now balanced.

Grading statistics	
Fill volume	670.77
Cut volume	673.19
Net volume	-2.42

Nota: Balance Grading Volumes command works only on gradings that has Boundary set to **OFF**.

Grading settings	
Cut slope	50
Fill slope	35
Side	Left
Infill	On
Boundary	Off

9.12 Create and edit alignments

9.12.1 Commands

ALIGNMENT, ALIGNMENTEDIT, ALIGNMENTVIEW

9.12.2 Overview

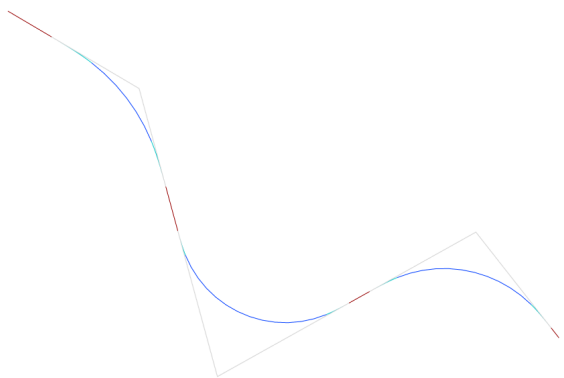
In BricsCAD, alignments are created using the Alignment tool. Alignment is the route of the linear objects, defined as a combination of lines and curves that are viewed as one object.

9.12.3 About ALIGNMENT

Alignment is a reference system used to position linear objects, like roads, railways, pipe networks, streets, retaining walls and bridges, in space. An alignment is represented with horizontal, vertical 2D curves, resulting in 3D alignment.

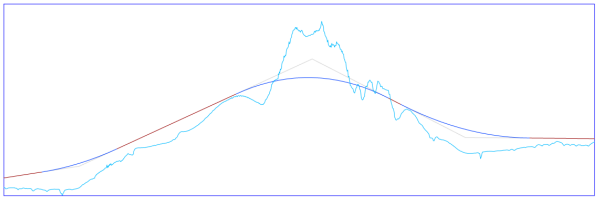
Horizontal alignment

Horizontal alignment is a 2D curve that is needed to precisely position linear objects in a horizontal direction. It becomes defined by the Points of Intersection (PI). Horizontal alignment is the horizontal trace on your TIN Surface which is used to generate the height profile and vertical alignment. A single horizontal alignment can have multiple vertical alignments. The horizontal alignment will appear under the surface.



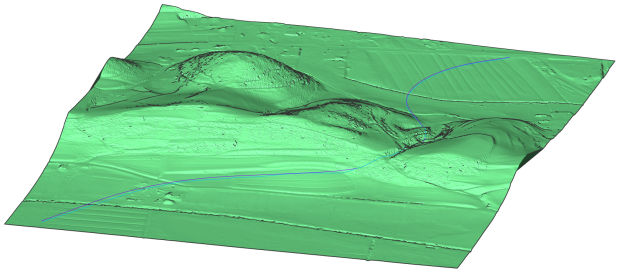
Vertical alignment

A vertical alignment is needed to precisely define a linear object in the vertical sense. The horizontal path defined by the horizontal alignment corresponds with a vertical profile of the TIN Surface. By defining Points of Vertical Intersection (PVI) on these vertical profiles of the target surfaces, the new 2D curve defines the vertical alignment.



3D alignment

A 3D Alignment is a 3D polygonal curve defined by the horizontal and vertical alignment.



9.12.4 Procedure: Creating an Alignment

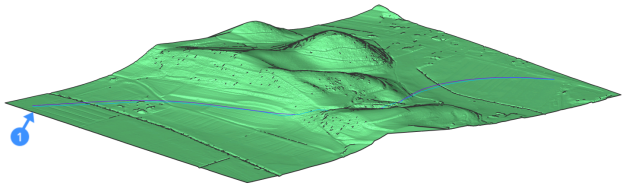
1 Open the drawing file that contains a TIN Surface.

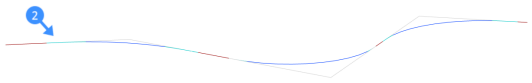
2 Launch the ALIGNMENT command in the Command line.

3 Select **Horizontal** alignment in the Command line.

4 Select the TIN surface.

5 Pick the horizontal alignment PI points on your surface. Press ENTER when done. Your 3D alignments (1) and horizontal alignment (2) are now generated.





Nota: Three alignments are shown. The horizontal alignment on plane 0,0, light blue 3D alignment shows the projection of horizontal alignment on TIN Surface and the purple 3D alignment shows the design alignment that is automatically generated.

9.12.5 Procedure: Creating a Horizontal Alignment without 3D alignments

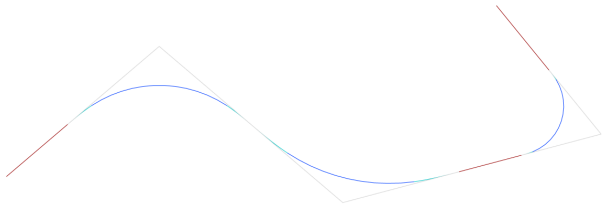
1 Open an empty drawing and launch the ALIGNMENT command in the Command line.

2 Select **Horizontal** alignment in the Command line.

3 Select the **Pick Pi point** option in the Command line.

4 Pick the horizontal alignment PI points. Press ENTER when done.

Your horizontal alignment is now generated.

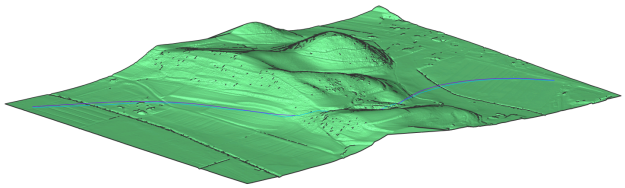


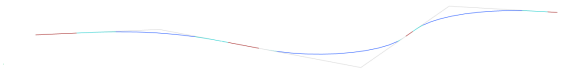
9.12.6 About Vertical Alignment View

The horizontal alignment creates a vertical section of the target surface. This 2D curve can be displayed in a separate viewport inside your model space using the command ALIGNMENTVIEW.

9.12.7 Procedure: Creating a Vertical Alignment View

1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and 3D Alignments.



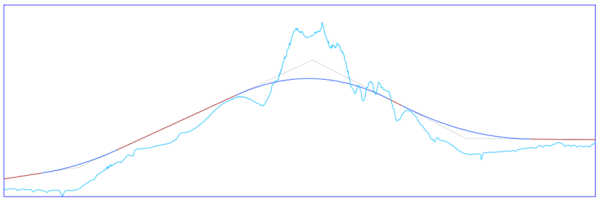


2 Launch the ALIGNMENTVIEW command in the Command line.

3 Select a horizontal or 3d alignment in the drawing.

4 Pick a point as origin for the vertical alignment view.

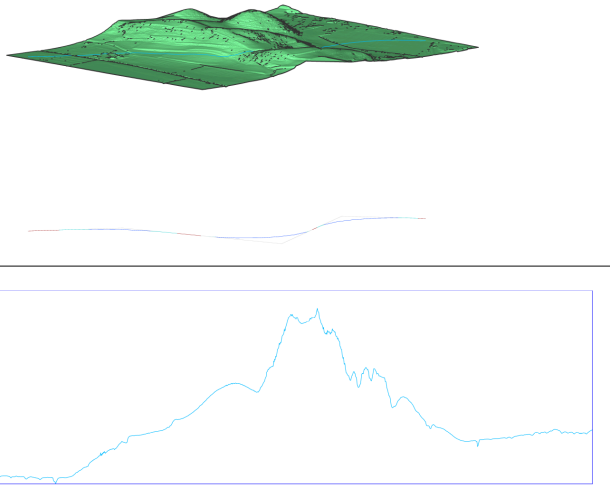
Your vertical alignment view is now generated.



Nota: Two vertical alignments are shown. Light blue represents terrain line from the TIN Surface, and another is design vertical alignment. The design vertical alignment has straight gradients and vertical curves and is created automatically based on the accuracy factor. You can change the accuracy factor in the Properties Panel from 1 to 100. When you choose 1 the accuracy will be the lowest.

9.12.8 Procedure: Creating a Vertical Alignment

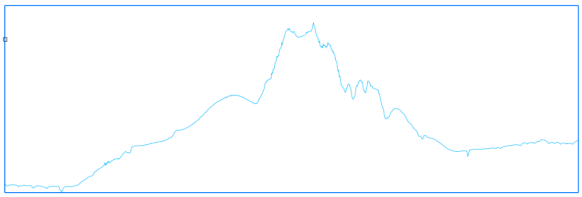
1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and Vertical Alignment View.



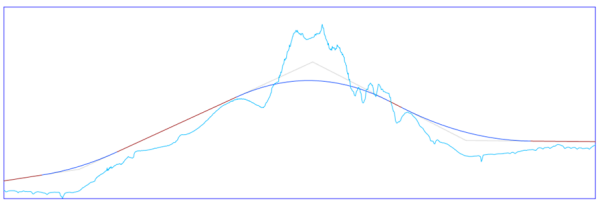
2 Launch the ALIGNMENT command in the Command line.

3 Select the **Vertical** option in the Command line.

4 Select vertical alignment view in the drawing.

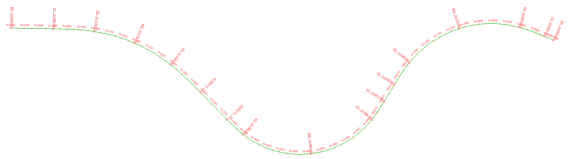


5 Pick the vertical alignment PVI points. Press ENTER when done. If you want to go one step back, use Undo. Automatic 3d alignment option generates design vertical alignment automatically. Your design vertical alignment is now generated.



9.12.9 Procedure: Creating an Alignment from Civil 3D drawing

1 Open the drawing file from Civil 3D that contains a Civil 3D Alignment.

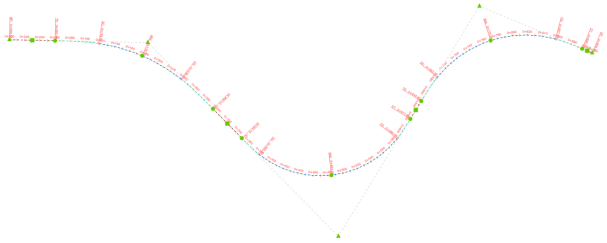


2 Launch the ALIGNMENT command in the Command line.

3 Select the **create from Civil 3d alignment**.

4 Select the civil alignment in the drawing and press ENTER.

Civil 3D alignment is converted to BricsCAD Alignment.



9.12.10 About ALIGNMENTEDIT

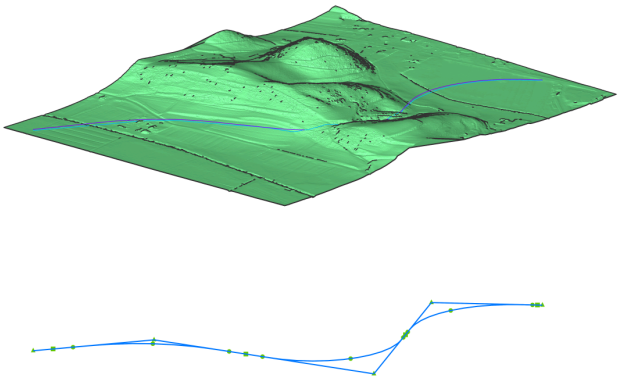
After creating an alignment, you can continue a horizontal or vertical alignment, add or delete PV/PVI points or change the TIN Surface using the command ALIGNMENTEDIT. The 3D alignment will be updated automatically.

9.12.11 Procedure: Continuing a horizontal alignment with ALIGNMENTEDIT

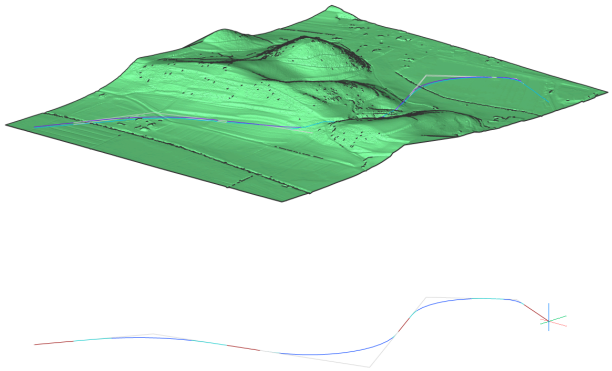
1 Draw a horizontal alignment following the steps from the Procedure: Creating an alignment.

2 Type ALIGNMENTEDIT in the Command line.

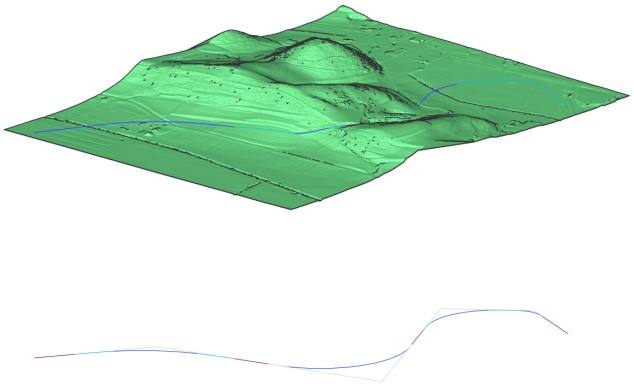
3 Select the horizontal alignment you want to edit.



4 Select points on your surface to continue your existing alignment.

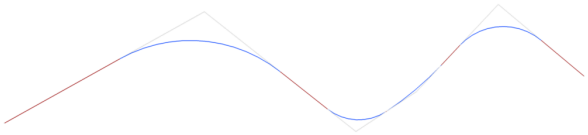


5 Press enter when finished. You have now continued your horizontal alignment. The 3D alignment will be updated automatically.



9.12.12 Procedure: Adding PI to an existing horizontal alignment

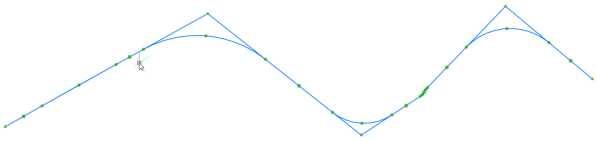
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit. You are prompted: Pick a point to continue alignment or [Add Pi/Remove Pi/change Tin surface].

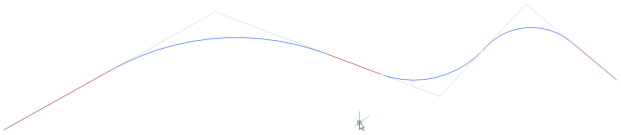
3 Select **Add Pi** in the Command line.

4 Pick point to add new PI.



9.12.13 Procedure: Removing PI to an existing horizontal alignment

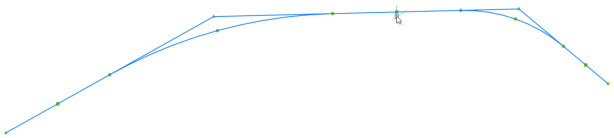
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit.

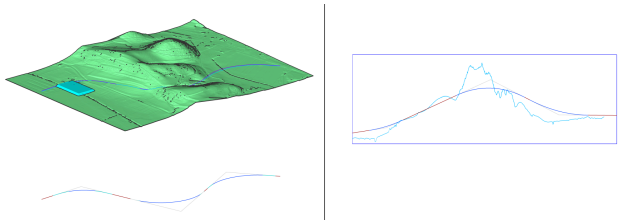
3 Select the **Remove Pi** option in the Command line.

4 Pick a PI point to remove.



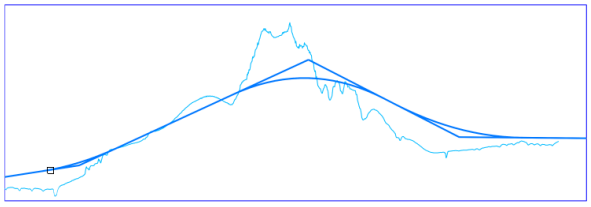
9.12.14 Procedure: Adding additional TIN Surface to Vertical alignment view

1 Open the drawing file that contains a TIN Surface, Alignments, Vertical Alignment View and grading.

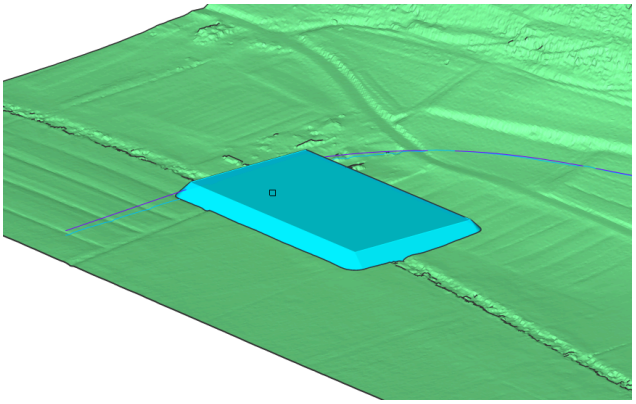


2 Launch the ALIGNMENTEDIT command in the Command line.

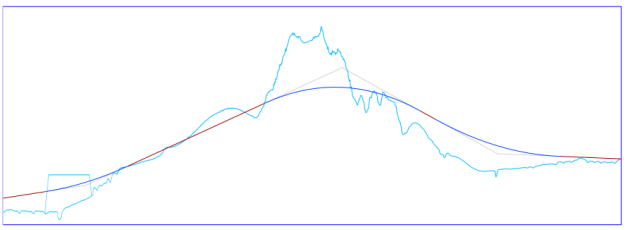
3 Select the **change Tin surface** option in the Command line.



4 Select the TIN surface to add.



In the vertical alignment view, you can see an additional light blue line representing the grading object.



9.13 Modeling a Corridor

9.13.1 Commands

CORRIDOR, CORRIDORTEMPLATE, CORRIDORTEMPLATEELEMENT, CORRIDOREDIT, CORRIDOREXTRACT.

9.13.2 Overview

In BricsCAD, Corridors that follow a certain 3D alignment are created using the Corridor tool.

9.13.3 About Corridor

A Corridor is a BricsCAD Civil drawing object used for modeling three-dimensional linear objects such as roads, railway, retaining walls, and bridges. It principally consists of a 3D alignment of the linear object and its typical cross-section – which is called Corridor Template in BricsCAD.

The CORRIDOR command enables you to create a Corridor, based on a Corridor Template, that follows a 3D alignment on a certain topographical surface.

9.13.4 About Corridor Template

In BricsCAD, a Corridor Template entity represents the framework to append a collection of template element objects. The command CORRIDORTEMPLATE enables you to create a corridor template entity and append one or more template elements to it.

9.13.5 About Corridor Template Element

In BricsCAD, a Corridor Template Element entity defines the geometry of a component used in a corridor section. The CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from closed polylines.

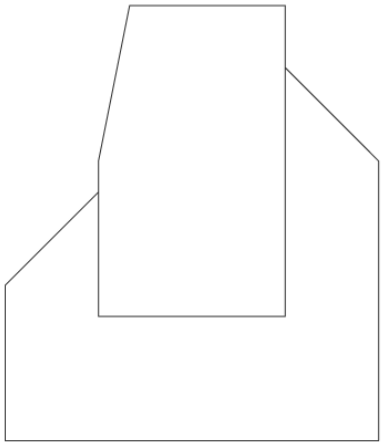
9.13.6 About Corridor Extract

The CORRIDOREXTRACT command is used to generate meshes, solids, polylines or outer boundaries from an existing Corridor.

9.13.7 Procedure: Creating a template element of kerb

1 Open an empty drawing.

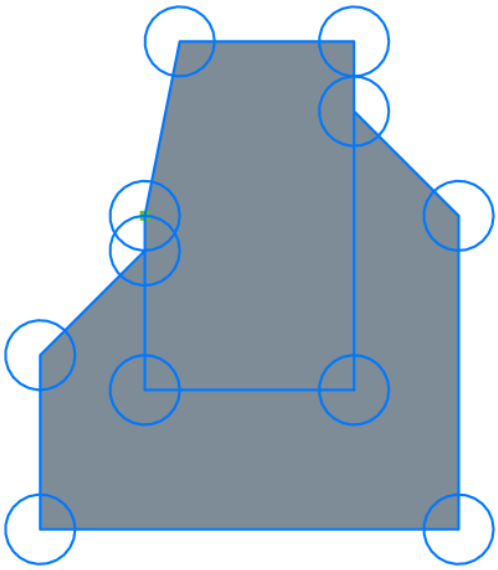
2 Draw closed polylines representing road kerb.



3 Launch the CORRIDORTEMPLATEELEMENT command in the Command line.

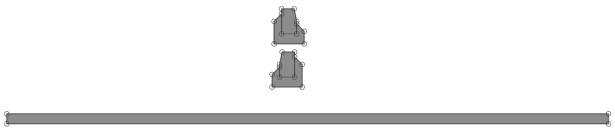
4 Select polylines representing kerb.

5 Pick one of the polyline vertices, the base point position can also be changed afterward.



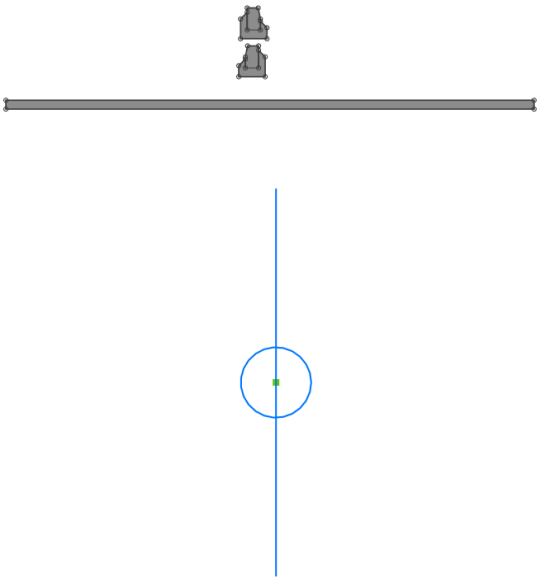
9.13.8 Procedure: Creating a template of road cross-section with asphalt layer and left and right kerb

1 Open the drawing file that contains a collection of template elements.



2 Launch the CORRIDORTEMPLATE command in the Command line.

3 Pick a point in the drawing where you want to create a corridor template.

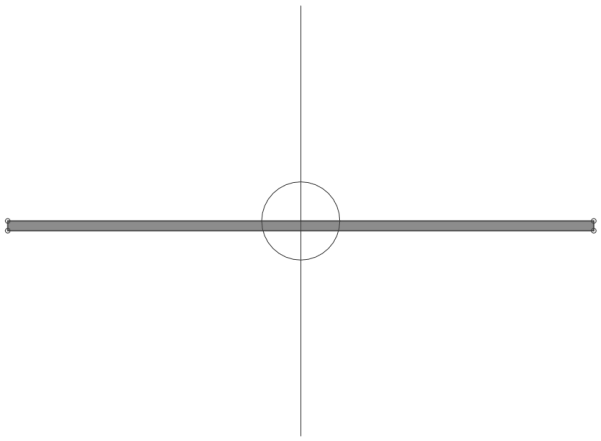


4 Launch the CORRIDORTEMPLATE command in the Command line.

5 Select the **Add template element** option in the Command line.

6 Select the template element representing the asphalt layer.

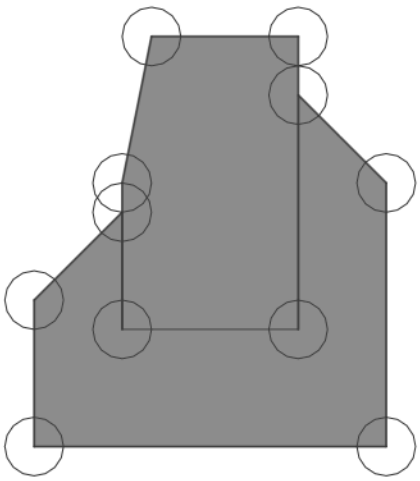
7 Select corridor template or template element point: (select the template)



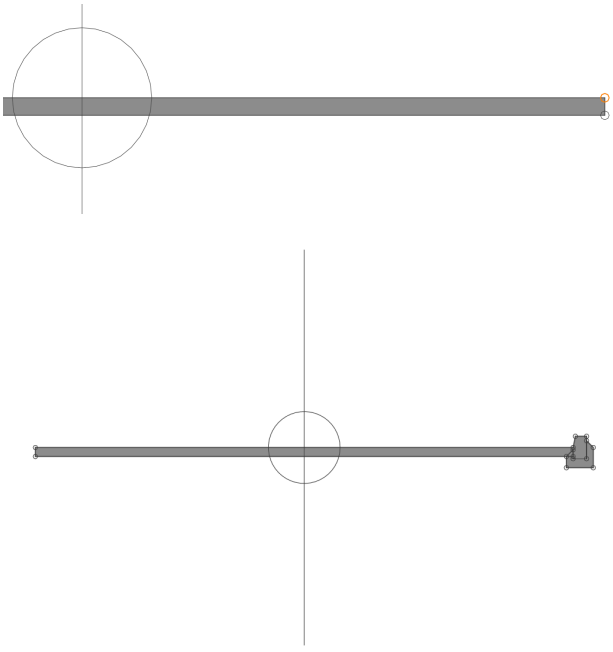
8 Launch the CORRIDORTEMPLATE command in the Command line.

9 Select the **Add template element** option in the Command line.

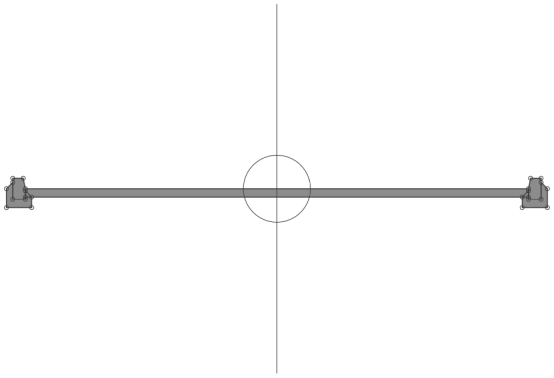
10 Select right top template element point of asphalt layer.



11 Select template element representing right kerb.

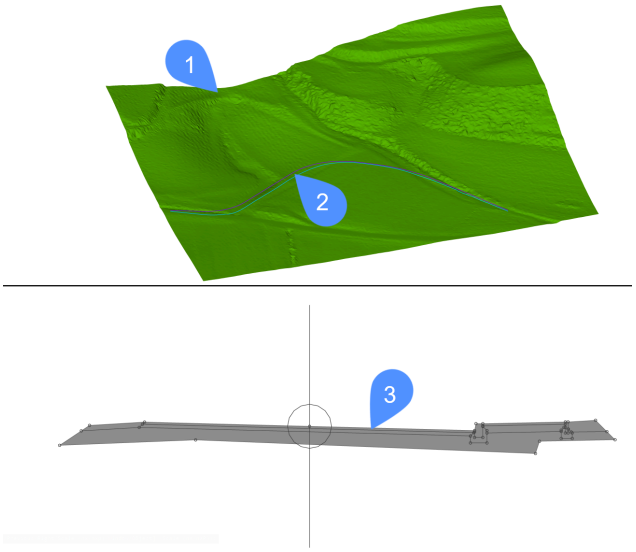


12 To add a kerb to the left side, repeat the steps described in points 9, 10, 11. By selecting the left kerb and left top point of asphalt layer this time.



9.13.9 Procedure: Creating a road corridor

1 Open the drawing file that contains the TIN Surface (1), 3D Alignment (2) and Corridor Template (3).



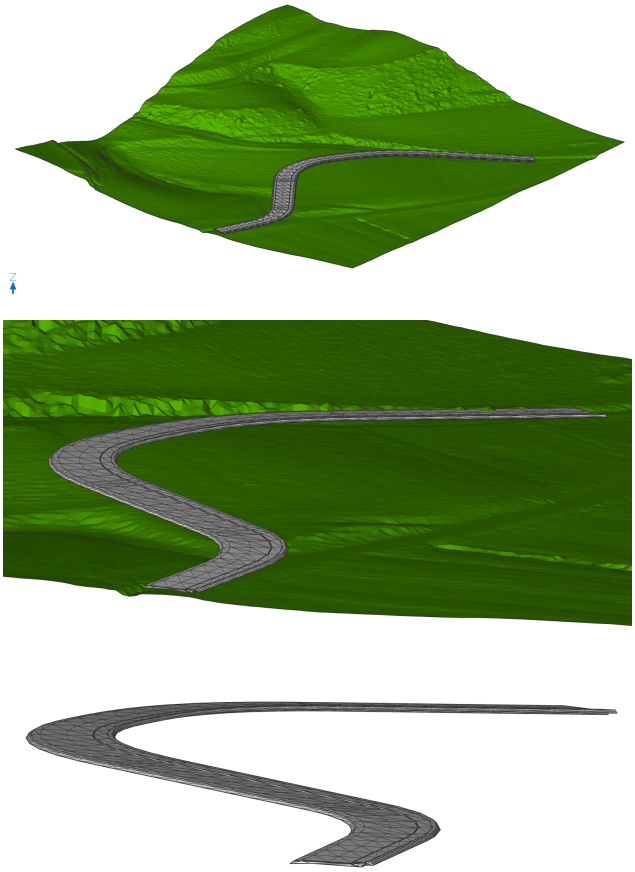
2 Launch the CORRIDOR command in the Command line.

3 Select 3D alignment (2) used as a corridor baseline.

4 Select corridor template (3).

5 Enter region start station. Press ENTER to start from beginning or type any station where you want your corridor to start.

6 Enter region end station. Press ENTER to create corridor to the end of selected alignment or type any station where you want your corridor to end.



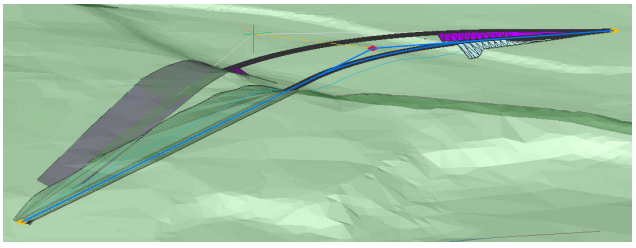
9.13.10 About Corridor Edit

The CORRIDOREDIT command adds or removes regions from an existing Corridor.

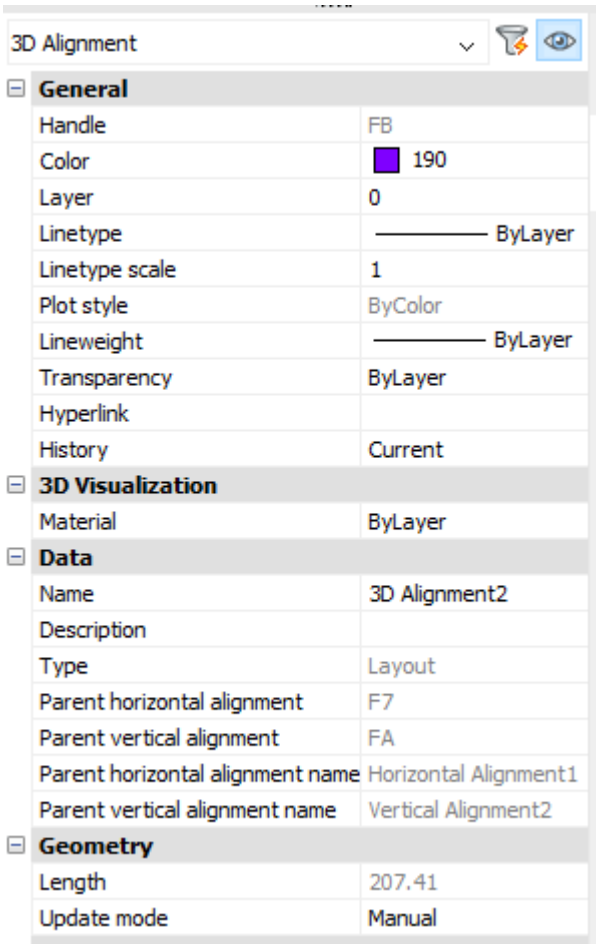
9.13.11 Procedure: Editing a corridor

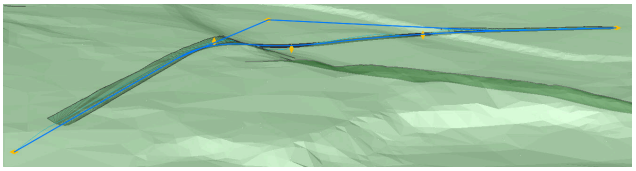
1 Open the drawing file that contains the corridor you would like to edit.

2 Select the 3D alignment that the corridor has as a reference and move the grip points to edit the corridor horizontally.

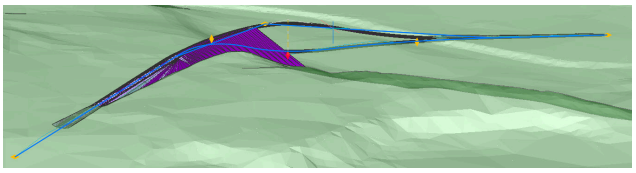


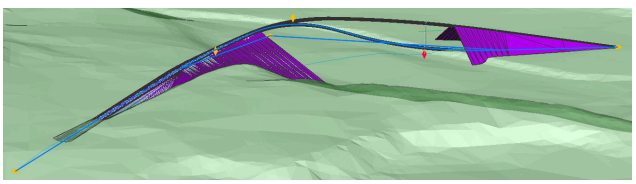
3 To edit the corridor vertically, select the 3D alignment that the corridor has as a reference and open the Properties Panel. Set the **Update mode** to Manual. Extra grip points will appear.



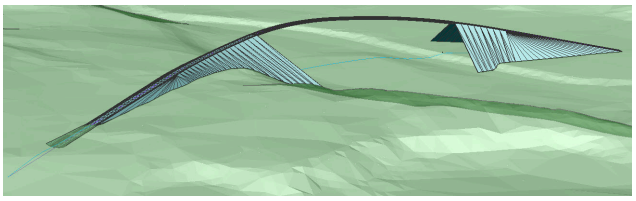


4 Move these grip points to edit the corridor vertically.





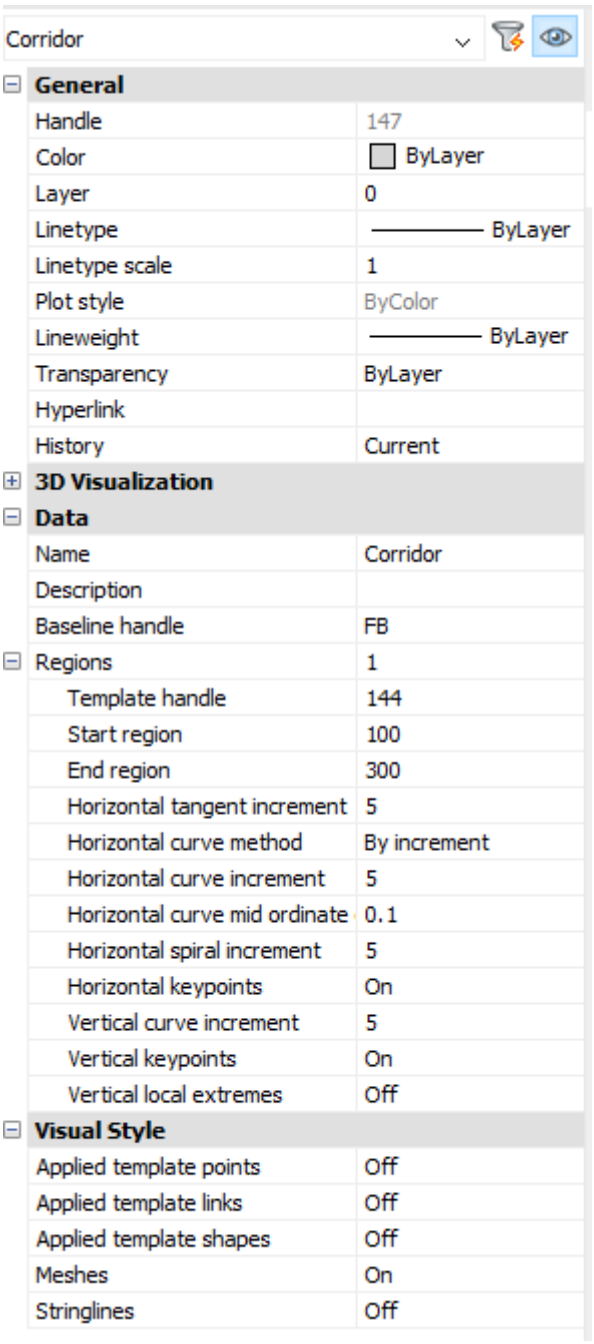
5 Press ENTER and you have finished your edited corridor.



9.13.12 Displaying the properties of a Corridor

1 Select the Corridor.

2 The properties of the Corridor are displayed in the Properties panel.

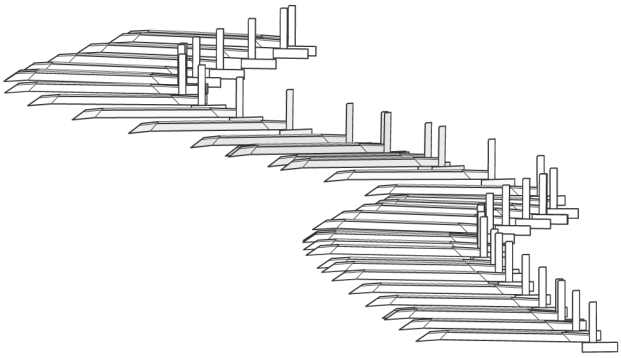


Applied template points

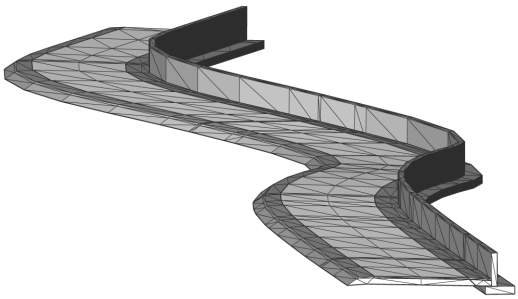
Represents the corridor with template element points.

Applied template links
Represents the corridor with template element links.

Applied template shapes
Represents the corridor with template element shapes.



Meshes
Represents the corridor with meshes.



Stringlines
Represents the corridor with lines connecting the template element points along the corridor.

10. Modelagem paramétrica

10.1 Parametric modeling overview

BricsCAD provides a rich toolset to control 2D and 3D geometric models with parameters of different kinds. This includes:

- **Global parameters**: parameters of 2D/3D dimensional constraints, user-defined parameters, configuration parameters of design tables, etc.
- **Local parameters**: parameters of arrays and constrained entities inside arrays, parameters of inserts of parametric components, etc.

All parameters and their current values can be found in different sections in the **Mechanical Browser** panel. Parameter values can be linked together via expressions. When you change the value of a parameter, the model is updated automatically. This workflow allows you to add parametric behavior to any geometry and explore design variants easily.

Nota: 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. However, you can create dependencies with parameters and expressions afterward.

10.1.1 Global and local parameters

Commands:

- -PARAMETERS command manages dimensional constraint parameters through the Command line.
- MECHANICALBROWSEROPEN command opens the **Mechanical Browser** panel.
- CLEANUNUSEDVARIABLES command purges parametric variables not used by constraint expressions and not linked to dimensions.

Parameters in BricsCAD can be either global or 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

10.1.2 Defining an expression for a parameter

You can define an expression for both parameter types, global and local. The simplest expression is a number or the name of a global parameter (local parameters cannot be referenced in expressions). A more complex expression can include standard operators and functions.

To define an expression for a parameter: in the **Mechanical Browser** panel, click on the parameter name and fill out the **Expression** field.

Nota: You can also define an expression for a parameter in the Command line using the -PARAMETERS command.

10.1.3 Defining dependent parameters

In expressions, you can use both: names of global parameters and numbers. This is known as a 'non-constant expression'. If a non-constant expression is set to a parameter, its icon changes from an open padlock () to a closed padlock () in the **Mechanical Browser** panel.

This means that the parameter becomes dependent on (an)other (defining) parameter(s) and its value will be automatically recalculated when the value(s) of the defining parameter(s) change(s).

You can create chains of parameters, where each element is dependent on the previous one. There is no limit to the length of such chain. However, be careful not to create a loop where the next element in a chain drives one of the previous elements. This might result in an infinite evaluation of expressions. BricsCAD automatically detects such problems and does not accept expressions that lead to a parametric loop.

10.1.4 Parameters of associative arrays

Associative arrays are created by the ARRAY, ARRAYRECT, ARRAYPOLAR and ARRATYPATH commands. The properties of associative arrays that can be controlled with an expression are the following.

- **Rectangular arrays**: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, and Axis angle.
- **Polar arrays**: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, and Level spacing.
- **Path arrays**: Item spacing, Start offset, Rows, Row spacing, Row elevation increment, Levels and Level spacing.

To define an expression for any of these properties, select the corresponding array entity, either in the drawing area or in the **Mechanical Browser** panel and enter the expression in the corresponding properties field in the **Mechanical Browser** panel or in the **Properties** panel.

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** panel. If you select any of them, you can define an expression.

10.1.5 Parameters of mechanical components

The BMUPDATEMODE system variablecontrols whether external assembly components are always reloaded or only when modified.

If you insert a mechanical component or just a plain .dwg file into a drawing with BMINSERT command and the insert contains global exposed parameters, these parameters are converted to local parameters. They are associated with the corresponding component insert and are visible in the **Mechanical Browser** panel under the name of this insert. Click the parameter(s) to edit.

10.1.6 Parameter properties

All parameters such as; user-defined, 3D constraints, design tables, parameters of parametric components and arrays can be managed via **Mechanical Browser** panel.

- **Name**: Identifies the parameter. The parameter name can be used in expressions of other parameters in the same **component**. Name cannot begin with a number, it should be alphanumeric without spaces.
- **Expression**: Type a value or an expression.
- **Value**: Displays the current value of the parameter.
- **Geometry-driven**: If yes, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.
- **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.

10.1.7 3D constraints properties

BricsCAD provides a set of properties for 3D constraints which allow you 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).

- **Type**: Indicates the constraint type: Path, Distance, Radius, Angle or Cone Angle.
- **Enabled**: Controls whether the constraint is On or Off.
- **Directions**: Specifies whether the directions of the constraint arguments are **Same** or **Opposite**. Changing from one to another flips constrained entities with respect to each other.

  Direction is used for lines, planes, circles, cylinders, cones and torii. It is not used for points and spheres.

  The direction of a plane is its normal vector. The direction of a circle is the normal vector of the circle's plane. The 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 with respect to the 3D solid object.

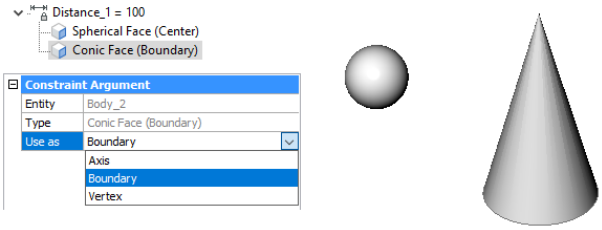
  Any option means that the directions may change upon direct modeling operations or upon a 3D constraint parameter change.

  The **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).

- **Placement**: 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 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). The **Keep** option is used by default.
- **Use as**: This property is assigned to the arguments of the **Distance** and **Concentric** constraints and applies to circles, spheres, cylinders, cones and torii and defines which geometry is used in a 3D constraint to measure the distance or to coincide with the other entity.



  There are four options for the **Use as** property:

  - **Boundary**: the constraint is applied to the boundary of the argument;
  - **Axis**: applies to an argument that has an axis: circle, cylinder, cone, or torus;
  - **Center**: applies to an argument that has a central point: circle, sphere, or torus;
  - **Vertex**: applies to a cone vertex only.

10.1.8 Context menus

In the **Mechanical Browser** panel, the following context menus are available.

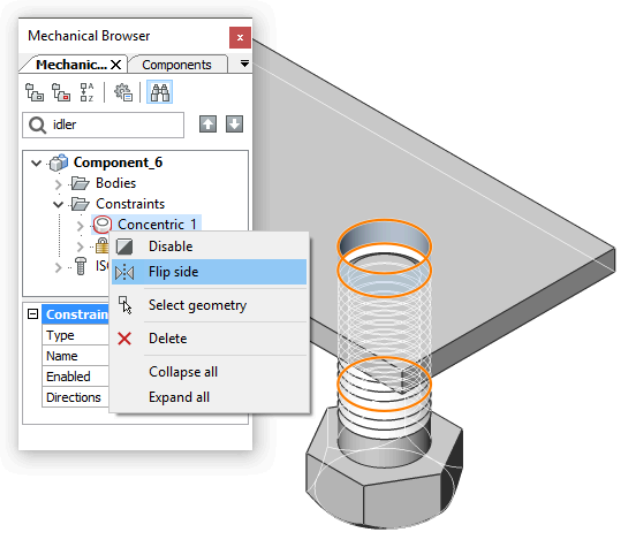
Main Node

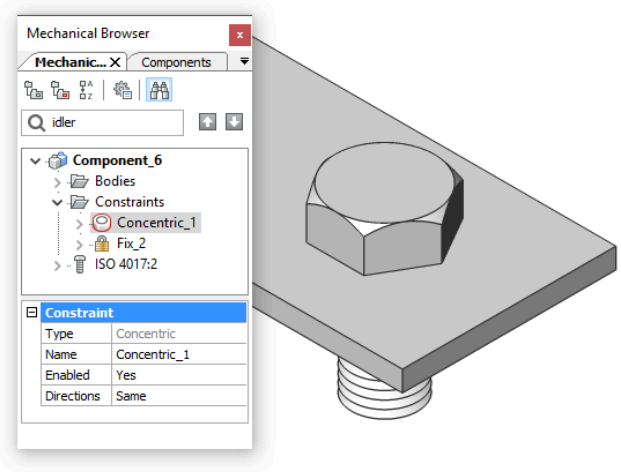
- **Add new parameter**: Creates a new parameter.
- **Collapse All**: Collapse all nodes.
- **Expand All**: Expand all nodes.

Sub-nodes

- **Delete**: deletes the selected parameter from file.
- **Create design table**: creates a design table to drive parametric block parameters.
- **Animate**: animates the parameter value within some range in the drawing.
- **Link to parameter**: links a subcomponent parameter to the main level parameter.
- **Enabled**: controls whether the constraint or parameter is evaluated or not.
- **Geometry driven**: if selected, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.

- **Flip Side**: Allows you 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.





- **Select geometry**: Selects the geometry that is affected by the constraint, in the drawing.

10.1.9 The Parameters and Constraints panel

The **Parameters and Constraints** 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.

In the **Parameters and Constraints** panel 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 a parameter by name
- Sort all parameters by name, expression, and value
- Hide all parameters except those relevant to the selected parameter
- Hide all parameters except those dependent on the selected parameter
- Animate a parameter

10.2 Operations and functions in expressions

10.2.1 Operators

Operation	Character	Description
Addition	+	Produces the sum of numeric values.
Subtraction	-	Subtracts the two numeric values.
Multiplication	*	Multiplies the numeric values.
Division	/	Divides two numeric values.
Exponentiation	^	Calculates the exponential value of the given numbers.
Remainder operator	%	Gives the remainder after the division of one numeric value to another. e.g. 5%2 = 1, because 5/2 leaves remainder of 1.

10.2.2 Expressions

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
   - 1. exponent
   - 2. multiplication and division
   - 3. addition and subtraction

3 Operators of equal precedence from left to right.

The following functions can be used in expressions:

Function	Syntax
Cosine	cos(expression)

Function	Syntax
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

Function	Syntax
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).

Nota:

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

10.3 3D constraints

10.3.1 Variational Direct Modeling

Use geometric constraints (aka variational direct modeling) to model parametric solids and surfaces. Define constraints between edges and faces to modify solid and surface models.

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 constraint creation order. It allows you to parametrize any feature of the 3D model without the worry of 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 its opposite faces.

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** (), **Path Constraint** ().

3D Constraints are available in the **3D Constraints** toolbar, the **Parametric/3D Constraints** menu, the **Home / Parametrize** ribbon panel and the **Constraints** tab on the Quad.

All 3D constraints are listed in the **Mechanical Browser** panel. Select a constraint to check and edit its properties.

The following (sub)entities and items are supported for 3D constraint creation:

- Vertices, edges and faces of 3D solids, surfaces, meshes and regions and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Lines, rays, xlines, circles and arcs.
- Segments and vertices of 3D polylines.
- Arbitrary curves for coincident and path constraints with a point as another argument.
- Planes, axes and origin points of the WCS or a block coordinate system.

3D constraints have a number of properties that allow you to control how BricsCAD treats them in various situations.

10.3.2 Geometry-driven constraints

Dimensional 3D constraints can also be **geometry-driven**. They update the model to accurately reflect the current state of the model. You can not specify their expression. Instead, their value is obtained from the model and can be changed with direct modeling operations and changes to other constraints.

Other parameters, including constraints, can depend on the values of geometry-driven constraints. In some circumstances a constraint dependent on geometry-driven constraints can make the model unresolvable. This should be avoided.

You can turn regular constraints into geometry-driven constraints and vice versa, in the **Mechanical Browser** panel and the **Parameters and Constraints** panel or using the DMCONSTRAINT3D command. After a regular constraint is turned into a geometry-driven one, its expression will be overwritten with the first modification of the model that affects this constraint.

10.3.3 3D constraints and direct modeling

3D constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint that fixes the placement of faces or edges, this constraint will be preserved during Push/Pull, Move and Rotate operations. A 3D constraint can be temporarily disabled, or erased permanently in the **Mechanical Browser** panel.

Geometry-driven constraints update their values when a direct modeling operation is completed, which possibly leads to additional charges in the model.

If some faces or edges disappear during direct modeling or Boolean solid editing operations, 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 the new entities.

Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometrical relations between the surfaces of a solid and will preserve them during direct modeling operations. This feature is called 'design intent recognition'.

10.4 Suppression feature

The suppression feature temporarily suppresses a set of entities in your drawing. If the condition is equal to zero, all entities are displayed. Otherwise, entities are suppressed.

Suppressed entities are internally placed on a special frozen layer. This allows you to speed up orbit, zoom and pan operations, improves object selection performance and reduces regeneration time for complex drawings. Suppressed entities are also excluded from mass properties calculations (generated drawing views and bill of materials). Use a parametrically-driven expression for the condition parameter of a suppression feature, to create different design variants of your parametric components.

10.4.1 Creating a suppression feature

Execute the SUPPRESS command.

10.4.2 Managing suppression features

Select the suppression feature in the **Mechanical Browser** panel, then right-click to display a context menu.

The available options are:

- **Suppress**: Suppresses the entity set.
  - The **Suppressed** icon () indicates the suppressed state.
  - The option now reads **Unsuppress**.
- **Unsuppress**: Unsuppresses the entity set.
  - The **Unsuppressed** icon () indicates the unsuppressed state.
  - The option now reads **Suppress**.
- **Dissolve**: Deletes the suppression feature.
- **Edit Selection**: Allows you to add/remove entities from the selection.

  All members of the selection are highlighted.
  - Select entities to add.
  - Shift + Select (hold down the Shift key, then use a selection method on) entities to remove.
  - Right-click to stop editing.
- **Zoom to**: Zooms onto selected entities.

  To compose a selection set:
  - Click a suppression feature to select all its members.
  - Expand a suppression feature, then click to select an entity.

- Ctrl + Click to add/remove entities from the selection.

- **Select**: Selects all members of a suppression feature.

10.4.3 Setting a condition for a suppression feature

You can set an expressional condition for a suppression feature. Depending on the value of the current expression, the entities will be suppressed or unsuppressed automatically.

In the Mechanical Browser panel:

1 Select the suppression feature (use the "Search" field to search by name).

2 Enter an expression in **Condition** field in the properties grid.

In the Parameters and Constraints panel:

1 Select the parameter with the same name as the suppression feature (use the "Search" field to search by name).

2 Enter an expression in the **Expression** column.

10.5 Working with design tables

A design table allows you to evaluate several global parameters at once with the value of one configuration parameter. You can create design tables from scratch in BricsCAD or import them from a .csv file (a comma-separated values text file). You can create such files with a spreadsheet software package or a text editor.

10.5.1 Creating a design table from a .csv file

The **From file** option of the DESIGNTABLE command imports .csv files that contain sets of parameter values, or else creates them from scratch. Parameters are added to the **Mechanical Browser** panel and to parametric component properties.

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,...,Value3ParameterN

...

The first row lists the parameter names separated by a comma. The first name must be different from the name of any global parameter in your drawing. It is called the configuration name. No spaces are allowed in the configuration name. Other names in the first row can be identical to the names of a global parameter The next rows list possible values for these parameters as strings or numbers.

Example:

Size,Length,Width,Height

Small,300,200,100

Medium, 450,300,150

Large,600,400,200

When the .csv file is imported into BricsCAD using the DESIGNTABLE command, a special global parameter is created by the name of the first name in the first row of the .csv file ('Size' in the above example). This is called the 'configuration parameter'. It will drive the values of other parameters in the design table. These are called 'configuration-dependent parameters'.

Nota: BricsCAD does not maintain an association with the .csv file used to create a design table: if this file is modified, then the design table is not updated automatically. Use the -DESIGNTABLEEDIT command to replace a design table with a new or modified .csv file.

10.5.2 Direct lookup in a design table

To evaluate a configuration parameter:

- Select it in the **Mechanical Browser** panel and type the desired value in its **Expression** field.
- Select a value in the drop-down list of the **Value** field:

If you change the value of a configuration parameter, all configuration-dependent parameter values change automatically in the corresponding design table. The values of these parameters are taken from the row in the design table. The first element coincides with the current value of the configuration parameter.

10.5.3 Reverse lookup in a design table

You can evaluate parameters that are defined by a configuration parameter with a design table independently.

Select the dependent parameter and apply an expression to it.

- If the value of this expression matches one of the values in the corresponding design table, then the configuration parameter and all dependent parameters of the design table will automatically change to the values specified in the corresponding row in the table.
- If several configurations are possible for a particular value of a dependent parameter, BricsCAD selects the one closest to the current values of the other parameters.
- If you enter an expression and the result is not available in the design table, or if you enter expressions for two or more design table parameters and 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 dependent parameters are not changed. This case corresponds to a configuration that does not exist in the design table.

10.5.4 Adding a new configuration to a design table

You can add any 'Custom' configuration to a design table and assign it a name. To do so, right-click the configuration parameter in the **Mechanical Browser** and select **Save current configuration** in the context menu. The configuration will be added to the design tables as Configuration_N:

- **Configuration**: the name of the configuration parameter.
- **N**: an index number - 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 command.

10.5.5 Creating an empty design table

If you use the procedure described above, it is possible to start your design with an empty design table and then fill it with the desired values.

Use the **Empty** option of the DESIGNTABLE command.

You will be prompted to specify:

- The name of a new design table
- The name of the corresponding configuration parameter
- The names of global parameters that will be dependent on this design table.

10.5.6 Using a parametric component with design tables

You can use both configuration parameters and configuration-dependent parameters in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables. A drop-down list of available values will be available. Both direct and reverse lookup actions are supported for such parameters.

You must associate the design table with the parametric component drawing first.

1 Open the drawing.

2 On the **Mechanical Browser** panel, you will see the list of parameters and their values.

3 Launch the DESIGNTABLE command and press Enter to accept the **From file** option.

The **Select design table file** dialog box appears.

4 Select the .csv file, then double-click or click the **Open** button.

5 Check the **Value** property for each of the main parameters. You can select the default values to use when you insert the component.

6 Check the **Exposed** property for each of the main parameters. They should all be set to **Auto**.

7 Save the component drawing.

10.5.7 Other actions with design tables

The -DESIGNTABLEEDIT command allows you to:

- Delete a design table
- Replace a design table
- Link a new parameter
- Unlink an existing parameter
- Add new configurations, remove an existing one
- Export a design table as a .csv file.

11. Nuvem de pontos

11.1 Working with Point cloud

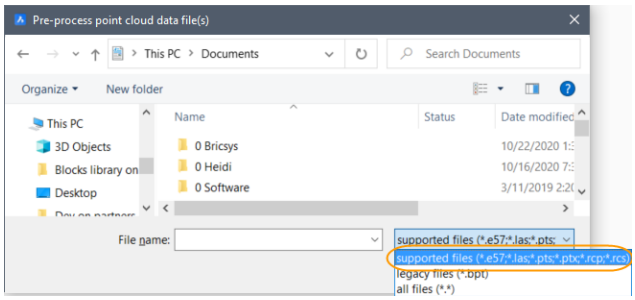
11.1.1 Commands

,,,,,,,, ,, ,,,,,,, ,, ,,

Point cloud functionality is significantly faster and more powerful in BricsCAD v21 and is even more powerful with a BIM license.

11.1.2 Pre-processing

The E57 file format is added to the supported files for pre-processing.

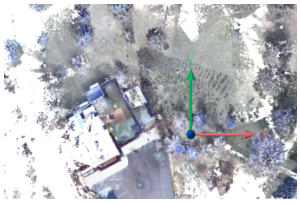


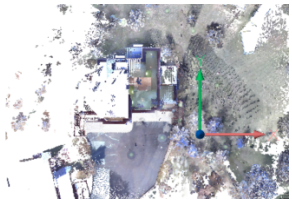
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD v21 than in v20.

The log file can be found in *C:\Users\USERNAME\AppData\Roaming\Bricsys\BricsCAD \V21x64\en_US\PointCloudCache\{Folder_for_processed_pointcloud}*, only if this path is set in POINTCLOUDCACHEFOLDER system variable. Inside this folder, the current state of processing can be observed.

11.1.3 Alignment

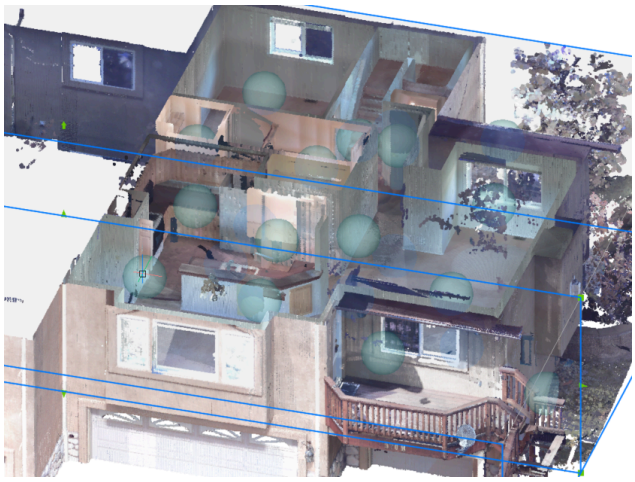
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment, it can analyze the entire point cloud, or you can specify the most relevant area.





11.1.4 Bubble Viewer

The point cloud displays bubbles at all the scan locations. It's at those locations where you will experience the most realistic visual representations.



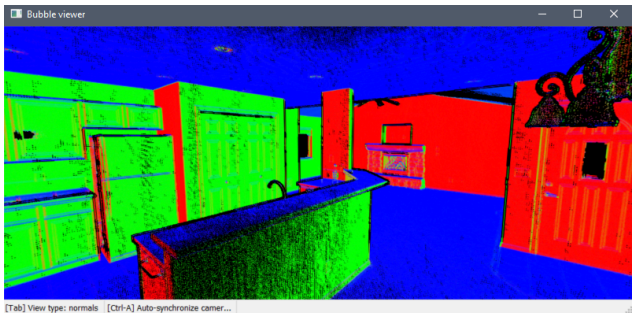
Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location. You can also zoom in and out using the mouse wheel.



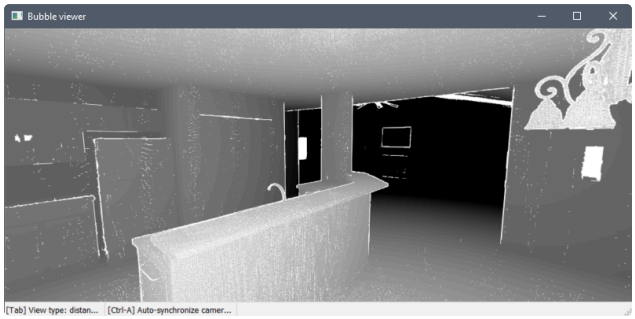
Press the Tab key to cycle between three different visual modes.

The first mode displays the points as their actual colors or in greyscale, depending how the data was scanned.

The second mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.



The third mode displays the points from light to dark as the distance from the scan location increases.



You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.



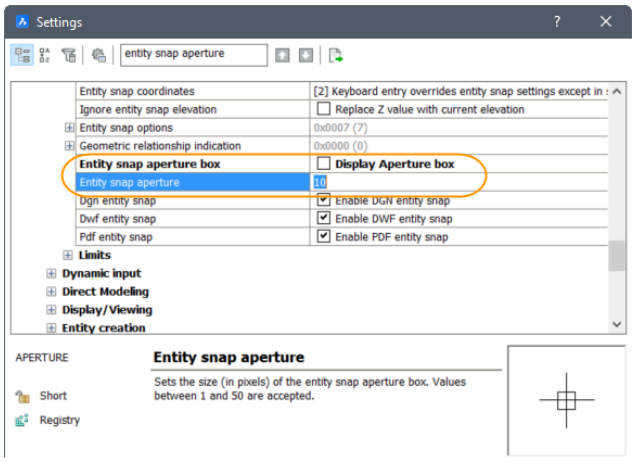
11.1.5 Entity snaps

The new Point Cloud Nearest Point entity snap significantly improves your ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.



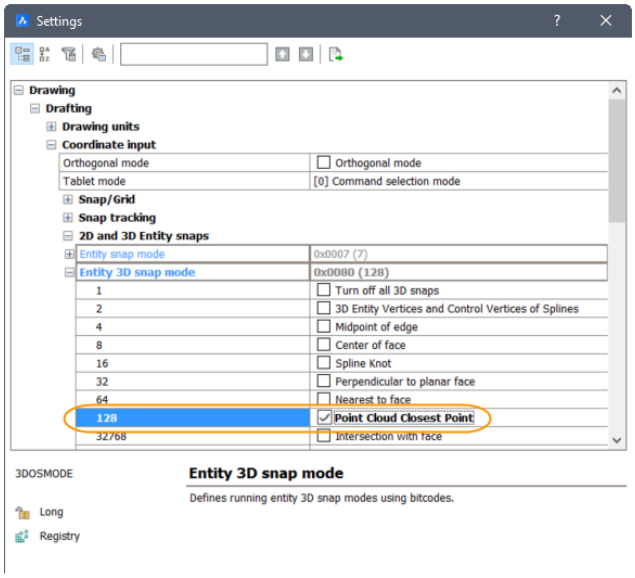


The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.



Enable the new Point Cloud Nearest Point entity snap along with other 3D entity snaps in the Entity Snap menus, toolbar, and settings.

Nota: Make sure that only the bit 128 of 3DSMODE is set.



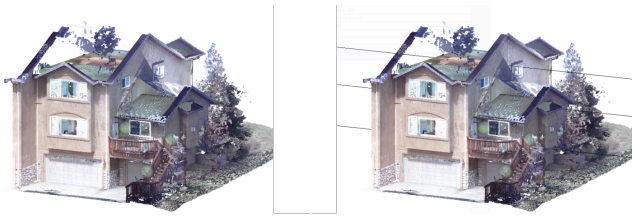
11.1.6 Export

The new POINTCLOUDEXPORT command allows you to export a cropped selection of a point cloud to a .pts file.

11.1.7 Floor Detection

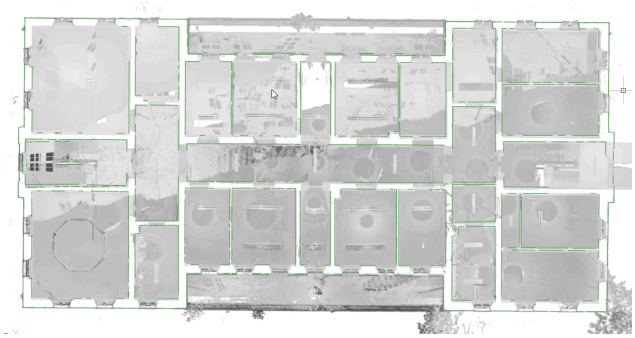
SectionPlanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and SectionPlanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

The new POINTCLOLUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

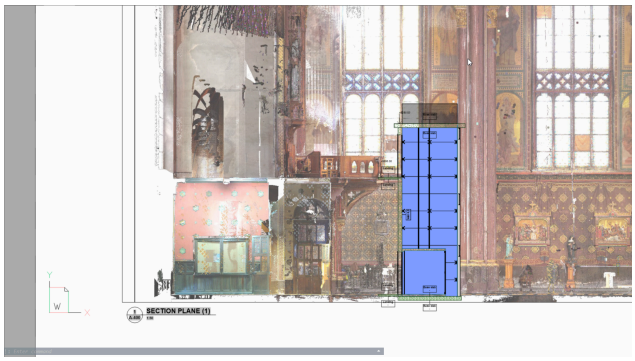


11.1.8 Point cloud projection

The newPOINTCLOUDPROJECTSECTIONcommand enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building. You can use these sections to generate 2d lines to create a 2d floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



At the same time, a raster image will be generated. In some cases, it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2d geometry but in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.

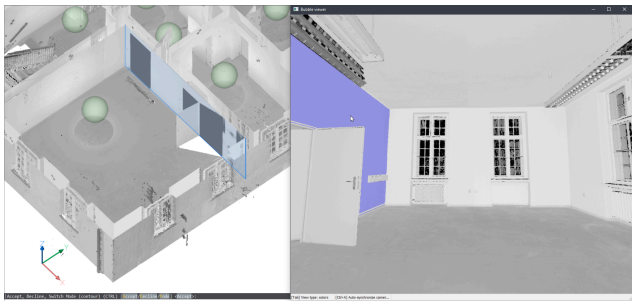


11.1.9 Planar Fit

The newPOINTCLOUDFITPLANARcommand enables you to create 3d geometry based on the point cloud. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

11.1.10 In bubble view

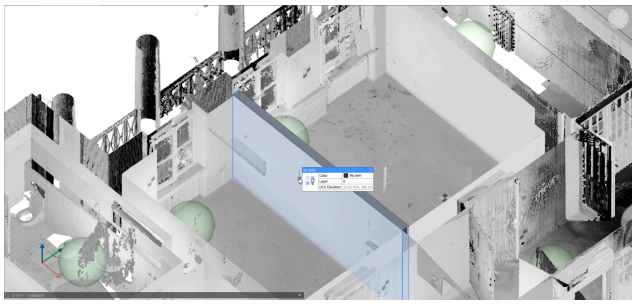
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.



11.1.11 In model space

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time, but it has 2 advantages by searching multiple scan positions:

- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.



12. Blocos, atributos, referencias externas e componentes.

12.1 Criar blocos

O comando BLOCO agrupa entidades existentes em um bloco usando uma caixa de diálogo.

O comando -BLOCO cria blocos de entidades existentes e solicita a você na barra de Comando.

Os comandos EDITARBLOCO e FECHARBLOCO criam ou editam blocos em uma sessão temporária de desenho.

O comando GBLOCO grava blocos e outras partes do desenho para um arquivo de desenho separado (abreviação para "grava bloco").

O comando COLARB cola entidades da **Área de Transferência** como um bloco no desenho atual.

O comando PLANOCORTEBLOCO salva o resultado do plano de corte selecionado em um bloco de corte 2D transversal / elevação ou em um bloco de corte 3D.

12.1.1 Para criar um bloco

1 Opcionalmente, selecione entidades para o bloco.

2 Inicie o comando BLOCO.

3 A caixa de diálogo **Definição do Bloco** é exibida:

4 Digite um nome para o novo bloco no campo **Nome**.

5 Siga um destes procedimentos para especificar o **Ponto Base**:

- Marque a opção **Especificar na tela**.

  Você será solicitado a especificar o ponto base depois de clicar no botão **OK**.

- Clique no botão **Escolher ponto** ( ).

  A caixa de diálogo **Definição do Bloco** fecha temporariamente para permitir que você escolha um ponto no desenho.

- Digite as coordenadas do ponto base no campo -X, -Y e -Z.

6 O número de entidades pré-selecionadas na etapa 1 é indicado.

7 Para selecionar mais entidades para o novo bloco, faça o seguinte:

- Selecione as entidades antes de iniciar o comando BLOCO.
- Marque a opção **Especificar na tela**.

  Você será solicitado a selecionar as entidades depois de clicar no botão **OK**.

- Clique o botão **Selecionar entidades** ( )

  A caixa de diálogo **Definição do Bloco** fecha temporariamente para permitir selecionar as entidades no desenho.

8 Defina as propriedades do bloco:

- **Anotativo**: cria um bloco anotativo.
- **Igualar orientação do bloco ao layout**: se marcado, a orientação de um bloco anotativo é

preservada em uma viewport rotacionada.

- **Escalar uniformemente**: se estiver marcada, os valores de escalar-X, -Y e -Z não poderão ser definidos individualmente.
- **Permitir explodir**: se não estiver marcada, impede o bloco de ser posteriormente explodido.
- **Unid. de bloco**: define a unidade de inserção para o bloco (consulte também: Inserir Blocos).

9 Defina o comportamento das entidades selecionadas depois que o bloco está criado:
- **Manter**: As entidades selecionadas permanecem no desenho.
- **Converter para bloco**: As entidades selecionadas são substituídas pelo recém-criado bloco.
- **Excluir**: As entidades selecionadas são excluídas.

10 Clique no botão **OK** para criar o bloco.

Notas:

- Se a opção **Excluir** na etapa 6 estiver marcada, o comando OOPS traz de volta as entidades apagadas do bloco.
- Se você digitar o nome de um bloco existente na etapa 3, será solicitado que você substitua a definição de bloco existente. As instâncias de bloco existentes serão substituídas pela nova definição de bloco.

12.1.2 Para salvar um bloco para um arquivo separado

1 Opcionalmente, selecione entidades para o bloco.

2 Inicie o comando GBLOCO.

3 A caixa de diálogo **Grava Bloco p/ Arquivo** é aberta.

4 Clique no botão de procurar [...] ao lado do campo **Nome e caminho**.

A caixa de diálogo **Salvar bloco** é exibida.

5 Selecione uma pasta e digite um nome no campo **Nome do arquivo**, e clique no botão **Salvar**.

A caixa de diálogo **Salvar Bloco** é fechada.

6 **Inserir unidades**: define o valor da variável de sistema INSUNITS para o arquivo de bloco.

Por padrão, esse é o valor INSUNITS do desenho de origem.

7 O número de entidades pré-selecionadas na etapa 1 é indicado.

8 Para selecionar mais entidades para o novo bloco, faça o seguinte:
- Clique o botão de rádio **Bloco** e selecione um bloco na lista drop-down.
- Clique no botão de opção **Desenho inteiro**, para selecionar todas as entidades no espaço do modelo do desenho atual.
- Clique no botão de rádio **Objetos**, então clique no botão **Selecionar Entidades**.

  A caixa de diálogo **Grava Bloco p/ Arquivo** fecha temporariamente para permitir que você selecione entidades no desenho.

  Clique o botão-direito para parar de selecionar entidades.

9 Clique no botão **Escolher ponto** ( ) para definir o **Ponto base** do novo desenho

ou

Digite as coordenadas do ponto base no campo -X, -Y e -Z.

10 Defina o comportamento das entidades selecionadas depois que o bloco está criado:
- **Manter**: As entidades selecionadas permanecem no desenho.
- **Converter para bloco**: As entidades selecionadas são substituídas pelo recém-criado bloco.
- **Excluir**: As entidades selecionadas são excluídas.

11 Clique no botão **OK** para criar o arquivo de desenho.

Nota: Se a opção **Excluir** na etapa 7 estiver definida, o comando OOPS traz de volta as entidades apagadas bloco.

12.2 Using Blockify



The Blockify tool enables you to automatically convert repetitive sets of entities to block definitions. Select a set of 2D entities or 3D solids and BricsCAD searches the drawing for identical sets of entities and automatically replaces them by block references.

For more information about this command, visit the BLOCKIFY Command Reference article.

12.2.1 Blockify from a selected input

1 Launch the BLOCKIFY command.

1 **You are prompted**: Select source entities or [match Equal solids/match existing Blocks/Settings/ selection options (?)] <match Equal solids>:

2 Select the entities you want to convert into a block and press Enter.

3 Entities selected: xx

You are prompted: Select source entities or [Settings/selection options (?)]:

4 Select more entities or press Enter or right click to stop selecting.

5 **You are prompted**: Select block insertion point or [use Default point] <use Default point>:

6 Identify the insertion point for the block.

7 **You are prompted**: Select search space or [Entire drawing/selection options (?)] <Entire drawing>:

8 Press Enter or right click to search the entire drawing.

9 **You are prompted**: Specify block name or <ENTER to use default name>:

10 Type a name for the block, then press Enter.

11 BricsCAD reports:

Duration of finding similar groups: x.xx sec

Number of block inserts: xx

Nota: Depending on the value of the BLOCKIFYMODE system variable, some prompts are suppressed and defaults are used for the insertion point, search space and/or block name.

12.2.2 Existing block definitions

One of the aims of the BLOCKIFY command is to automatically add and improve the structure of a model. If the model already contains block definitions, the existing block definitions are checked.

- **Use a selected input set of entities**: When one of the existing block definitions matches the BLOCKIFY input set, no definition will be created. All instances of the input will be replaced by block references to the existing block definition.
- **Match existing blocks**: The entire drawing, or selection set, is searched for matching geometry in existing block definitions. As such, repeated geometry corresponding to a block definition is replaced by block references to existing definition(s). This option allows to recreate block instances that were exploded accidentally.

12.2.3 Combining duplicate block definitions

Identical block definitions can be combined into a single one and optionally purge the duplicate using the OVERKILL command.

1 Launch the OVERKILL command.

1 **You are prompted**: Select entities or [Combine duplicate block definitions/selection options (?)]:

2 Choose the **Combine duplicate block definitions** option.

The **Combine duplicate block definitions** dialog box appears.

3 (option) Select the properties to ignore.

4 (option) Tick the **Purge duplicate definitions** option.

5 Click the **OK** button.

BricsCAD reports:

- Number of duplicate block definitions found
- Names of duplicate block definitions
- Purged block definitions

12.3 Editar blocos

Você pode acessar o Editor de Blocos com o comando EDITARBLOCO, ou clicar duas-vezes no bloco que deseja editar. Ao usar o Editor de Bloco, o BricsCAD exibe as entidades do bloco em uma sessão temporária de desenho. Inclui todas as ferramentas necessárias para criar e editar uma definição de Bloco. Para sair da sessão de edição de bloco, use o comando FECHARBLOCO e escolha se deseja salvar ou descartar as alterações. Se você selecionar a ferramenta SALVAR enquanto estiver na sessão de editar o bloco, o BricsCAD salvará as alterações na definição de bloco e vai sair do Editor de Bloco.

Blocos também podem ser editados usando o comando EDITARREF.

12.4 Inserir blocos

O comando INSERIR permite inserir blocos através de uma caixa de diálogo.

O comando -INSERIR insere blocos, solicitando pela barra de Comando. BricsCAD pesquisa o nome do bloco entre as definições de bloco no desenho atual. Se o bloco não for encontrado no desenho atual, os caminhos definidos pela variável do sistema SRCHPATH serão pesquisados. Se o bloco não for encontrado, o BricsCAD responde "Não foi possível encontrar o arquivo <blockname>".

O comando INSERTALIGNED insere uma entidade de bloco, com alinhamento fácil em entidades existentes. O comando INSERTALIGNED é semelhante ao comando INSERIR, mas durante o posicionamento o bloco se alinha dinamicamente com linhas, polilinhas, arcos ou círculos existentes. O bloco se alinha com a entidade à qual o cursor se encaixa. Se nenhuma entidade for encaixada, o bloco será alinhado com o eixo-X do WCS.

O comando INSERIRM insere um bloco como uma matriz retangular; combina os comandos -INSERIR e MATRIZ (abreviação de "inserção múltipla").

Você pode optar por selecionar um bloco existente ou inserir um desenho inteiro como bloco. Quando você insere um desenho, uma nova definição de bloco é criada no desenho atual. Se você alterar o arquivo de desenho original, essas alterações não terão efeito no desenho atual, a menos que você redefina o bloco, pela reinserção do desenho alterado. Quando um bloco é inserido em um desenho, ele é tratado como uma entidade isolada.

Notas:

- Quando inserir um desenho como um bloco, a origem é definida pela variável do sistema INSBASE do desenho inserido. A variável é definida pelo comando BASE.
- Se um bloco contém atributos, você será solicitado a preencher o texto para cada atributo, pela barra de Comando.
- Se a variável de sistema DRAGOPEN é definida em zero (Desl), desenhos podem ser inseridos como um bloco, arrastando-os a partir da caixa de diálogo do **Windows Explorer** para o desenho atual; Se DRAGOPEN = 1 (Liga), em vez disso, o desenho será aberto.
- As variáveis do sistema INSUNITS e INSUNITSSCALING controlam o dimensionamento automático ao inserir blocos ou anexar RefEx's quando o valor de INSUNITS é diferente no bloco ou RefEx e no desenho de destino.
- Se o valor de INSUNITS no desenho de origem for 0 (Não especificado), o valor da variável de sistema INSUNITSDEFSOURCE será usado.
- Se o valor de INSUNITS no desenho de destino for 0 (Não especificado), o valor da variável de sistema INSUNITSDEFTARGET será usado.
- Os valores de INSUNITSDEFSOURCE e INSUNITSDEFTARGET são salvos no registro e, portanto, aplicam-se a todos os desenhos nos quais o valor da variável de sistema INSUNITS não é especificado (sem unidade).

- Quando você digita o nome de um bloco no prompt 'Bloco para inserir:', o BricsCAD vai procurar o bloco nas definições de bloco no desenho atual. Se o bloco não for encontrado lá, os caminhos definidos pela variável de sistema SRCHPATH serão pesquisados. Se o bloco não for encontrado, o BricsCAD responde "Não foi possível encontrar o arquivo <blockname>".

Na caixa de diálogo **Inserir Bloco**:

- O campo **Unidade** em **Unidade do Bloco** é definido pela variável do sistema INSUNITS no desenho de origem ou pela propriedade **Unidade do Bloco** de uma definição de bloco (consulte Criar Blocos).
- O campo **Fator** expressa a relação entre o valor da variável INSUNITS no desenho de origem e o desenho de destino. Ex., se INSUNITS for Milímetros na origem e no destino, o valor do campo **Fator** é 0.1.

12.4.1 Usando o comando INSERIR

1 Inicie o comando INSERIR.

1 A caixa de diálogo **Inserir Bloco** é exibida.

2 Siga um dos procedimentos:

- Selecione uma definição de bloco existente na lista **Nome**.

  O nome do bloco selecionado é exibido no campo **Nome**.

- Clique o botão **Procurar** e selecione um arquivo de desenho.

  O campo de caminho mostra a localização do desenho selecionado.

3 (opção) Verifique a opção **Curvas de referência do usuário para inserir**.

4 Defina os modos **Ponto de Inserção**, **Escala** e **Rotação**.

As opções são:

- Especifique as configurações na caixa de diálogo **Inserir Bloco**.
- **Especificar na tela**: Você será solicitado durante o processo de inserção.

5 (opção) Marque a opção **Explodir**, para explodir o bloco após a inserção.

6 Clique o botão **OK**.

Você será solicitado de acordo com as opções selecionadas.

12.4.2 Para inserir um bloco usando a caixa de diálogo Explorer do Desenho

Inserir blocos do desenho atual.

1 Inicie o comando EXPBLOCKS.

1 A caixa de diálogo **Explorer do Desenho — Blocos** é exibida.

2 (opção) Defina o modo de visualização: **Detalhe** ou **Ícone**.

3 (opção) Selecione um bloco, clique o botão-direito e escolha **Opções...** no menu de contexto.

A caixa de diálogo **Opções do Explorer do Desenho** aparece.

Defina as opções de inserção e clique no botão **OK**.

4 Clique duas vezes no ícone de bloco ou no número quando estiver no modo **Vista Detalhe**.

A caixa de diálogo **Explorer do Desenho** é fechada para permitir que você insira o bloco no desenho.

5 Especificar o ponto de inserção.

- Dependendo das opções de inserção atuais, você é solicitado a especificar a escala e/ou rotação.
- Quando a opção **Alinhar** estiver selecionada, você deverá especificar a posição do bloco em relação à entidade linear selecionada.

  Você é solicitado: Entre o ponto de controle de espelhamento do bloco:

A caixa de diálogo **Explorer do Desenho** reabre.

Inserir blocos de outro desenho.

1 Abra a caixa de diálogo **Explorer do Desenho**.

2 No painel **Desenhos**, clique na aba **Pastas**.

3 (Opção) Se nenhuma pasta ainda estiver disponível na árvore **Pastas Locais**, ou se quiser adicionar uma pasta, clique no botão **Adicionar pasta local...**.

A caixa de diálogo **Escolher uma pasta** é exibida.

Navegue até a pasta que deseja adicionar e clique no botão **Selecionar Pasta**.

A pasta e suas subpastas são adicionadas na árvore **Pastas Locais**.

4 Selecione um desenho na árvore **Pastas Locais**.

5 Expanda o desenho, e selecione **Blocos**.

Todos os blocos no desenho selecionado são exibidos no painel **Blocos**.

6 Prossiga da etapa 2 do procedimento anterior.

12.4.3 Para inserir um bloco alinhado com uma entidade

1 Certifique-se de que os Snaps de Entidade apropriados estejam ativos.

2 Inicie o comando INSERTALIGNED.

3 **Você é solicitado**: Bloco para inserir (~ para abrir o diálogo de arquivo) ou [? para listar blocos no desenho] <bloco atual>:

4 Siga um dos procedimentos:

- Pressione Enter para aceitar o bloco atualmente selecionado.
- Digite o nome do bloco para inserir.
- Pressione a tecla ? para ver uma lista dos blocos disponíveis.

Você é solicitado: Ponto de inserção para o bloco ou [Escala/Escala X/Escala Y/Escala Z/Rotacionar/Vários blocos]:

5 (opção) Escolha a opção **Vários blocos** ou especifique uma escala fixa.

6 Especifique um ponto.

7 O bloco exibe o ponto de snap da entidade selecionada, alinhado com a entidade linear.

Você é solicitado: Entre o ponto de controle de espelhamento do bloco:

8 Mova o cursor para selecionar a posição desejada, e clique para inserir o bloco.

Nota: O bloco está alinhado da seguinte forma:

O bloco é rotacionado de modo que seu eixo-Z local seja paralelo ao eixo-Z do UCS ativo, e seu eixo-X local deve ser paralelo com a tangente à entidade na posição do ponto de snap. Quando você clica em um ponto em uma entidade, o bloco é alinhado com a entidade. Então, movendo o ponteiro do mouse ao redor do ponto de inserção, você pode escolher como o bloco deve ser espelhado. O ponto de origem do bloco está na entidade.

12.4.4 Para inserir várias instâncias de um bloco em uma matriz retangular

1 Inicie o comando INSERIRM.

1 **Você é solicitado**: Bloco para inserir (~ para abrir o diálogo de arquivo) ou [? para listar blocos no desenho] <bloco atual>:

2 Siga um dos procedimentos:

- Pressione Enter para aceitar o bloco atualmente selecionado.
- Digite o nome do bloco para inserir.
- Pressione a tecla ? para ver uma lista dos blocos disponíveis.

Você é solicitado: Ponto de inserção para o bloco ou [Escala/Escala X/Escala Y/Escala Z/Rotacionar/Vários blocos]:

3 (opção) Escolha a opção **Vários blocos** ou especifique uma escala fixa.

4 Especifique um ponto.

5 **Você é solicitado**: Angulo de rotação para o bloco <0°0'0">:

6 Pressione Enter para aceitar o ângulo de rotação padrão ou especificar um ângulo.

7 **Você é solicitado**: Número de linhas na matriz <1>:

8 Especifique um número.

9 **Você é solicitado**: Número de colunas < 1>:

10 Especifique um número.

11 **Você é solicitado**: Distância vertical entre linhas, ou retângulo de espaçamento:

12 Siga um dos procedimentos:

- Especificar um valor.
- **Você é solicitado**: Distância horizontal entre colunas:

  Especificar um valor.
- Clique em dois pontos para especificar um retângulo, que define as distâncias de espaçamento horizontal e vertical.

O bloco é inserido como uma entidade **Bloco InserirM**.

Notas:

- Se você escolher **Múltiplos blocos** na etapa 3, você será solicitado a criar várias instâncias da matriz de blocos.
- Não é possível explodir uma entidade do tipo **Bloco InserirM**.

- Você pode editar as propriedades de uma entidade **Bloco InserirM** no painel **Propriedades**.

12.5 Explodir Blocos

Se a propriedade de um bloco estiver definida **Explodível**, você pode explodir um bloco inserido para suas entidades componentes originais. Explodir um bloco afeta apenas essa única instância (cópia) do bloco. A definição original do bloco permanece no desenho, e você ainda pode inserir cópias adicionais do bloco original. Se você explodir um bloco que contenha atributos, os atributos serão perdidos, mas as definições do atributo original permanecerão.

A variável do sistema EXPLMODE controla se é possível explodir blocos não-uniformemente dimensionados.

12.5.1 Para explodir um bloco

1 Inicie o comando EXPLODIR.

1 **Você é solicitado**: Selecione entidades para explodir [opções de seleção (?)]:

2 Selecione um ou mais blocos.

3 Entidades no conjunto: x

Você é solicitado: Selecione entidades para explodir [opções de seleção (?)]:

4 Selecione mais entidades ou pressione Enter para parar de selecionar.

5 Os blocos selecionados são explodidos.

Nota: O ato de explodir dissocia as entidades componentes para o seu próximo nível (mais simples) de complexidade: blocos ou polilinhas aninhados em um bloco tornam-se novamente blocos ou polilinhas originais.

12.5.2 Para definir a propriedade Explodível de um bloco

A propriedade **Explodível** de um bloco é definida ao criar um bloco usando o comando BLOCO.

Para editar a propriedade, faça o seguinte:

1 Inicie o comando EXPBLOCKS.

1 A caixa de diálogo **Explorer do Desenho — Blocos** é exibida.

2 Escolha o modo de **Vista Detalhe**.

3 Clique na caixa de seleção **Explodir** para alternar a propriedade **Explodivel**.

12.6 blocos paramétricos 2D

Se você vem de um ambiente de projeto em AutoCAD®, talvez esteja familiarizado com os Blocos Dinâmicos. Você também pode estar familiarizado com restrições paramétricas 2D. São duas funcionalidades diferentes que se sobrepõem um pouco.

Em vez de oferecer a funcionalidade do bloco dinâmico, o BricsCAD concentra seu foco em parametrização. Essa permite que você crie definições de bloco tanto em 2D como em 3D usando o mesmo conjunto de ferramentas e fluxo de trabalho. Se você nunca trabalhou com restrições

paramétricas, elas podem soar assustadoras. Mas são bastante intuitivas e lógicas. E, incrivelmente poderosas!

12.6.1 Procedimento geral para criar blocos paramétricos 2D

Crie a geometria.

Crie restrições geométricas.

Crie restrições e parâmetros dimensionais.

Edite restrições e parâmetros dimensionais no painel **Parâmetros e Restrições** ou no painel **Navegador de Mecânica**.

Opcionalmente crie estados de visibilidade para mostrar/ocultar a geometria, usando o comando ESTADOSDEVISIBILIDADE.

Salve o desenho como um bloco de biblioteca, usando o comando CRIARBLOCOBIBLIOTECA.

Nota: O comando PARAMETRIZAR2D cria automaticamente restrições geométricas e dimensionais.

12.6.2 Inserir um bloco paramétrico

Você pode inserir blocos paramétricos no desenho usando os comandos INSERIR ou BMINSERT, ou pelo painel **Biblioteca**.

12.6.3 Configurar os valores dos parâmetros

Selecione o bloco paramétrico, e altere os valores dos parâmetros expostos em **Parâmetros** no painel **Propriedades**.

12.7 Dynamic 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!

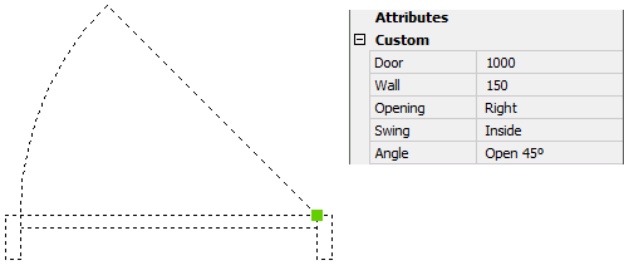
BricsCAD partly supports dynamic blocks, you can:

- insert dynamic blocks from your existing dynamic blocks libraries.
- edit the custom properties in the **Properties** panel.
- grip edit dynamic blocks.

12.7.1 Editing dynamic blocks

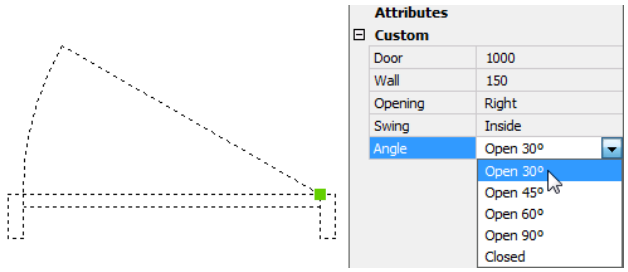
1 Select the dynamic block in the drawing.

1 The properties of the selected block are displayed in the **Properties** panel.



2 Under **Attributes** > **Custom**: edit the properties needed.

The dynamic block is updated automatically.



12.7.2 Grip editing dynamic blocks

1 Select the dynamic block in the drawing.

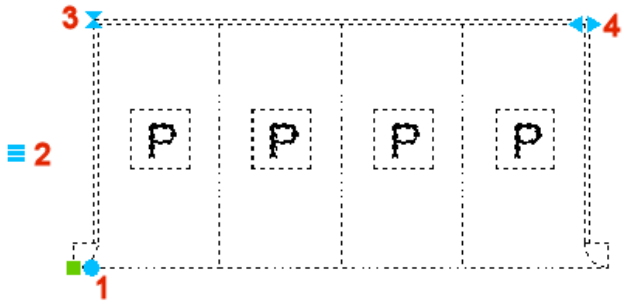
1 The following grip types are displayed:

- Point
- XY
- Linear
- Polar
- Rotation
- Flip
- Visibility

2 Drag a grip to edit the block.

3 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.



12.8 Definir atributos

O comando DEFATRIB define atributos por meio de uma caixa de diálogo.

O comando -DEFATRIB define atributos no prompt de Comando.

12.8.1 Procedimento geral para definir atributos

- Especificar as características do atributo, incluindo seu nome, aviso e um valor padrão.
- Especifique as coordenadas de inserção.
- Defina as propriedades do texto.
- (opção) Defina os sinalizadores de atributo.

12.8.2 Para definir um atributo

1 Siga um dos procedimentos:

- Clique no botão **Atributos** > **Definir Atributos...** ( ) na barra **Ferramentas**.
- Escolha **Definir Atributos...** no menu **Ferramentas**.
- Inicie o comando DEFATRIB.

A caixa de diálogo **Definir Atributo** aparece.

2 Definir o Atributo:

- Digite um nome no campo **Etiq**.

  A Etiqueta identifica cada ocorrência de um atributo no desenho. O nome que você digita no campo **Etiq** pode ser qualquer combinação de caracteres, exceto espaços. Letras minúsculas são convertidas automaticamente para maiúsculas.

- Digite um aviso no campo **Prompt** (Aviso).

  O prompt de atributo aparece quando você insere um bloco que contém o atributo. Se um prompt for omitido, a Etiqueta será usada no lugar do prompt.

- Siga um dos procedimentos:

• Digite um valor padrão no campo de texto **Padrão**.

  Para atributos variáveis, o valor padrão é substituído pelo valor real quando você insere um bloco que contém o atributo.

• Clique o botão **Inserir Campo** ( ) para atribuir um valor de campo ao atributo (ver 'Usar conjunto de folhas e propriedades da folha num bloco de titulo').

- Se o sinalizador **Múltiplas linhas** está marcado, o campo de texto **Padrão** é esmaecido.
- Clique no botão ao lado do campo de **texto Padrão**.

  A caixa de diálogo **Definir Atributo** fecha temporariamente para permitir que você crie uma entidade TextoM.

- Siga um dos procedimentos:
- Especificar as coordenadas X, Y e Z para o ponto de inserção dos atributos.
- Clique no botão Escolher Ponto ( ) para especificar o ponto de inserção no desenho.

  Você é solicitado: Selecione o ponto de inserção.

  A caixa de diálogo **Definir Atributo** fecha temporariamente.

- (opção) Defina o sinalizador de atributo **invisível**.

  Os atributos com o sinalizador **Invisível** não são exibidos, e nem impressos.

  Se a variável ATTMODE (Modo de Exibição de Atributo) estiver definida como **2**, todos os atributos são exibidos, incluindo atributos **Ocultos**.

- (opção) Defina o sinalizador de atributo **Constante**.

  Atributos com o sinalizador definido **Constante** não podem ser editados.

- (opção) Defina o sinalizador **Verificar** atributo.

  Os atributos com o sinalizador **Verificar** definido devem ser validados explicitamente quando um bloco contendo esses atributos for inserido.

- (opção) Defina o sinalizador de atributo **Predefinido**.

  Ao inserir um bloco, você não é solicitado a definir atributos dos quais o sinalizador **Predefinido** está ajustado. Você ainda pode editar os atributos depois.

- (Opção) Defina o sinalizador de atributo **Bloquear Posição** para bloquear a posição do atributo dentro do bloco.

  Atributos desbloqueados podem ser movidos em relação ao restante do bloco usando edição por alças, e atributos de múltiplas linhas podem ser redimensionados.

- Defina as propriedades do **Texto**: **Estilo de Texto**, **Justificação**, **Anotativa**, **Altura do texto** e **Rotação**.

3 Clique no botão **OK** para criar o atributo no desenho.

  A caixa de diálogo **Definir Atributo** é fechada.

12.8.3 Usando propriedades de conjunto de folhas em um bloco de título

1 Criar o conjunto de folhas.

2 Definir o conjunto de folhas personalizado e as propriedades da folha.

3 Abra uma folha (layout) do conjunto de folhas.

4 Criar as entidades de origem do bloco de título no layout de Paper Space:

- Criar bordas, trabalhos de linha e textos fixos.
- Crie atributos com campos que se referem ao conjunto de folhas, subconjunto e propriedades

da folha.

Consulte o comando CONJUNTODEFOLHAS para saber mais sobre as propriedades de conjunto de folhas, subconjunto e das folhas.

Propriedades personalizadas de conjunto de folhas:

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

Propriedades da folha e propriedades personalizadas da folha:

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

5 Criar um bloco a partir das entidades de origem do bloco de título.

6 Copiar o bloco para um desenho em uma de suas pastas de blocos.

7 Excluir as entidades de origem do bloco de título.

O bloco de título pode ser inserido nas outras folhas do conjunto de folhas e em qualquer outro conjunto de folhas em que existam o mesmo conjunto de folhas personalizado e as propriedades da folha.
Todos os campos que se referem ao conjunto de folhas padrão ou personalizado, ao subconjunto ou às propriedades da planilha são preenchidos automaticamente. Se nenhum valor está disponível, um campo exibe hifens (----).

12.9 Trabalhar com atributos

Um atributo é uma entidade específica que você pode salvar como parte de uma definição de bloco. Atributos consistem em dados baseados em texto. Você pode usar atributos para rastrear itens como número da peça e preços. Atributos possuem valores fixos ou variáveis. Quando você insere um bloco contendo atributos, o programa adiciona os valores fixos ao desenho, junto com o bloco, e você é solicitado a fornecer os demais valores de variáveis.

Depois de inserir blocos contendo atributos, você pode extrair as informações do atributo para um arquivo separado, e então usar essas informações em uma planilha ou banco de dados para produzir uma lista de peças ou lista de materiais. Você também pode usar informações de atributos para rastrear o número de vezes que um determinado bloco é inserido em um desenho.

Atributos podem estar visíveis ou ocultos. Atributos ocultos não são exibidos e nem plotados, mas a informação ainda assim está armazenada no desenho e será gravada em um arquivo quando você extrai-lo.

Comando DEFATRIB - define atributos por meio de uma caixa de diálogo.

-DEFATRIB comando - define atributos através da barra de Comando.

Comando VISATRIB - alterna a exibição do texto do atributo.

Comando EDITARATRIB - edita valores e propriedades de atributos.

Comando -EXTRATRIB - extrai atributos de desenhos para arquivos de dados através da barra de Comando.

Comando ATTREDEF - redefine um bloco e atualiza os atributos associados.

Comando SINCATRIB - sincroniza definições de atributo em todas as referências de bloco de uma definição de bloco especificada.

O comando GERATRIB (abreviação de "gerenciador de atributos de bloco") edita todos os aspectos dos atributos em uma definição de bloco, e opcionalmente aplica as alterações a todos os blocos que tenham o mesmo nome no desenho.

Comando EDITARATRIBA - edita valores de atributos e a maioria das propriedades.

12.9.1 Procedimento geral para trabalhar com atributos

1 Definir os atributos

2 Criar um bloco.

Você pode anexar atributos a um bloco. Inclua os atributos quando o programa solicitar que você selecione as entidades a ser incluídas na definição de bloco. Se um bloco contiver atributos, o programa vai solicitar cada vez que você inserir o bloco, para que possa especificar valores diferentes para os atributos sempre que for inserido em um novo desenho.

1 Inserir o bloco em um desenho.

2 (opção) Editar os atributos em um bloco.

3 Extrair informações de atributo

12.10 Editar definições de atributo

As propriedades do Atributo podem ser editadas no painel **Propriedades**.

O comando EDITARATRIB edita valores de atributos e propriedades.

O comando GERATRIB (abreviação de "gerenciador de atributos de bloco") edita todos os aspectos dos atributos em uma definição de bloco, e opcionalmente aplica as alterações a todos os blocos que tenham o mesmo nome no desenho.

Nota: Os atributos em uma definição de bloco podem ser editados no painel **Propriedades** ao usar o comando EDITARBLOCO para editar o bloco.

12.11 Editar atributos de bloco

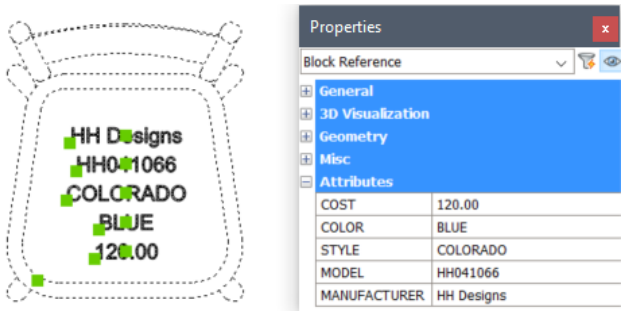
Os atributos de bloco podem ser editados no painel **Propriedades**.

1 Selecione um bloco que contém atributos.

1 O valor atual dos atributos é exibido no painel **Propriedades**.

2 Selecione o atributo que deseja editar.

3 O campo de configurações do atributo selecionado está ativo.



4 Digite um novo valor, e pressione Enter ou selecione outro atributo.

12.12 Inserir blocos com atributos

Ao colocar um bloco que contém atributos, você será solicitado a definir os atributos na barra de Comando, ou através de uma caixa de diálogo, dependendo do valor da variável de sistema ATTDIA:

- Liga: os atributos são exibidos em uma caixa de diálogo.
- Desl: você será solicitado na barra de Comando para definir os atributos.

12.12.1 Para inserir um bloco com atributos

1 Siga um dos procedimentos:

- Clique na ferramenta **Inserir Bloco** ( ) na barra **Inserir** ou no painel **Inserir/Blocos** da Faixa de opções.
- Escolher **Inserir Bloco...** no menu **Inserir**.
- Inicie o comando INSERIR.

A caixa de diálogo **Inserir Bloco** é exibida.

2 Selecione o bloco, e clique no botão **OK**.

Você é solicitado: Ponto de inserção para o bloco ou [Escala/Escala X/Escala Y/Escala Z/Rotacionar/Vários blocos]:

3 Especifique um ponto.

4 Todas as entidades de bloco são colocadas no desenho, exceto os atributos.

5 Dependendo do valor da variável de sistema ATTDIA:
- **Você é solicitado:** <Aviso do Atributo> <Default Text 1>:
- A caixa de diálogo **Editar Atributos** é exibida.

6 Siga um dos procedimentos:
- Na barra de Comando, para cada atributo:
  - Pressione Enter ou clique o botão-direito para aceitar o texto padrão.
  - Digite um novo texto na barra de Comando, e pressione Enter e defina o atributo a seguir.
- No diálogo **Editar Atributos**:
  - Selecione um atributo na lista.
  - O valor atual é exibido no campo **Valor**.
  - Digite um novo valor no campo **Valor**.
  - Pressione **OK** quando todos os atributos estão definidos corretamente.

12.13 Extrair informações de atributo

O comando EXTRATRIB copia dados de atributos para um arquivo de texto, através de uma caixa de diálogo.

O comando -EXTRATRIB copia dados de atributos para um arquivo de texto via linha de Comando.

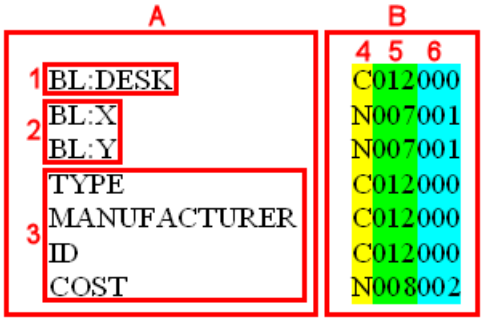
O arquivo de texto pode a ser importado em um banco de dados ou numa planilha.

Você pode salvar o arquivo em qualquer um dos seguintes formatos:

- **Comma Delimited Format** (CDF): Contém uma linha para cada instância de um bloco, com campos de atributo individuais separados por vírgulas. Os campos de string de caracteres são envolvidos por aspa única (apóstrofo). Você deve especificar um arquivo template ao extrair para um arquivo CDF.
- **Space Delimited Format** (SDF): Contém uma linha para cada instância de um bloco. Cada campo de atributo tem um comprimento fixo; não há separadores ou delimitadores de sequência de caracteres. Você deve especificar um arquivo template ao extrair para um arquivo SDF.
- **Drawing Exchange Format** (DXF): Cria um subconjunto de um arquivo padrão DXF (um arquivo *.dxx) contendo todas as informações sobre cada bloco, incluindo os pontos de inserção, ângulos de rotação, e valores de atributo. Nenhum arquivo template é necessário.

Antes de extrair atributos para um arquivo CDF ou SDF, você deve criar um arquivo como template. O arquivo template é um arquivo de texto ASCII que especifica os campos de dados de atributo a ser gravados no arquivo de extração. Cada linha do arquivo template especifica um campo de atributo.
O BricsCAD reconhece 15 campos diferentes, que contêm elementos como o nome do bloco, as coordenadas X, Y e Z de seu ponto de inserção, a camada na qual ele está inserido, etc. Você pode incluir quaisquer desses campos. O arquivo template deve incluir pelo menos um nome de atributo.

Cada linha no arquivo template deve começar com o nome do campo. O nome do bloco e os valores do ponto de inserção devem começar com **BL:**. O próximo caractere não vazio deve ser um C (indicando um campo de cadeia de caracteres) ou um N (indicando um campo numérico). Esse caractere é seguido por três dígitos que indicam a largura do campo (em caracteres). Os três dígitos finais indicam o número de casas decimais (para campos numéricos). No caso de campos de caracteres, os últimos três dígitos devem ser zeros (000). Um arquivo template típico é semelhante ao mostrado aqui:



A. Nome do campo

B. Formato do campo

1. Nome do bloco (deve ser precedido por **BL:**)
2. Campos de coordenadas (devem ser precedidos por **BL:**)
3. Etiqueta de atributo
4. Indica o tipo de dado do campo: **C** para campos de caracteres, **N** para campos numéricos
5. Largura do campo (número de caracteres)
6. Número de casas decimais para campos numéricos, **000** para campos de caracteres

12.13.1 Para extrair informação de atributo

1 Siga um dos procedimentos:
   - Clique no botão **Extrair Atributos** na barra **Ferramentas**.
   - Escolha **Extrair Atributos** no menu **Ferramentas**.
   - Inicie o comando EXTRATRIB.

   A caixa de diálogo **Extrair Atributos** é exibida.

2 Clique no botão **Selecionar**.

   A caixa de diálogo **Extrair Atributos** fecha temporariamente.

3 Selecione os blocos, e pressione Enter ou clique o botão-direito do mouse.

4 A caixa de diálogo **Extrair Atributos** é reaberta.

5 Selecione o formato de arquivo de texto: **DXF** (drawing exchange format), **CDF** (comma delimited format) **ou SDF** (space delimited format).

6 Selecionar o **arquivo Template**.

7 Especifique o nome do **arquivo de Saída**.

8 Clique no botão **Extrair**.

Você é solicitado: <número> registros extraídos.

12.14 Gerenciando RefEx's

O comando EXPREFEX abre a seção **Referências Externas** da caixa de diálogo **Explorer do Desenho**.

O comando REFEX abre o painel **Anexos**.

A caixa de diálogo **Explorer do Desenho - Referências Externas** permite:

- Anexar um desenho como uma referência externa ( ).
- Desanexar uma referência externa ( ).
- Recarregar uma referência externa ( ).
- Descarregar uma referência externa ( ).
- Juntar uma referência externa ( ).
- Inserir uma referência externa ( ).
- Modificar o tipo de anexação: **Anexar** ou **Sobrepor**.
- Modificar o **Caminho Salvo** de um RefEx.
- Modificar o **Cam. Encontrado** de um RefEx.

12.14.1 Para desanexar referências externas

1 Selecione o(s) RefEx que você deseja desanexar.

1 Mantenha pressionada a tecla Ctrl para selecionar vários RefEx's, ou para desselecionar um RefEx selecionado.

2 Clique no botão **Desanexar RefEx** ( ) na barra **Detalhes**.

Os RefEx's selecionados são desanexados. Você não é solicitado a confirmar o desanexação.

12.14.2 Para descarregar referências externas

Siga um dos procedimentos:

- Clique na coluna **Carregado** de um RefEx carregado.
- Selecione o(s) RefEx, e clique no botão **Descarregar RefEx** ( ) na barra **Detalhes**, ou clique na coluna **Carregado** de um dos RefEx's selecionados.

12.14.3 Para recarregar referências externas

Siga um dos procedimentos:

- Clique na coluna **Carregado** de um RefEx descarregado.
- Selecione o(s) RefEx, e então clique no botão **Recarregar RefEx** ( ) na barra **Detalhes**, ou clique na coluna **Carregado** de um dos RefEx's selecionados.

12.15 Anexar uma referencia externa

O comando ANEXAREFEX anexa um desenho como Referência Externa (RefEx). A RefEx anexada aparece como uma inserção no desenho atual. Como blocos, Refexs anexados podem estar aninhados. O desenho anexado representa a versão salva mais recentemente.

12.15.1 Para anexar um arquivo de referência externa

1 Siga um dos procedimentos:

- Clique no botão **Anexar RefEx...** () na barra **Inserir** ou no painel **Inserir / Referências** da Faixa de opções.
- Escolha **Anexar RefEx...** no menu **Inserir**.
- Clique no botão **Anexar RefEx** () na barra **Detalhes** na caixa de diálogo **Explorer do Desenho / Refexs**.
- Clique no botão **Anexar Dwg...** () no painel **Anexos**.
- Inicie o comando ANEXAREFEX.

A caixa de diálogo **Selecionar Arquivo de Referência** é exibida.

2 Selecione o desenho que você deseja anexar e pressione o botão **Abrir**.

A caixa de diálogo **Anexar Referência Externa** é exibida.

3 O caminho completo para o desenho selecionado é exibido no campo **Caminho**.

(opção) Selecione o **Tipo de Caminho**: **Completo**, **Relativo** ou **Sem caminho**.

4 Escolha o **Tipo de Referência Externa**:

- **Anexo**: Se o desenho atual é anexado a outro desenho, como uma Refex, um anexo se torna uma Refex aninhada.
- **Sobrepor**: Um arquivo sobreposto é ignorado quando o desenho ao qual ele está anexado for por sua vez anexado a um outro desenho, como referência externa.

5 Escolha se deseja especificar o **Ponto de Inserção** na tela ou nos campos **X**, **Y**, **Z**.

6 Escolha se deseja especificar a **Escala** na tela ou nos campos **X**, **Y**, **Z**.

Marque a opção **Escala Uniforme** para especificar a escala apenas no campo **X**.

7 Escolha se deseja especificar o ângulo de **Rotação** na tela ou no campo **Angulo**.

8 Clique no botão **OK** para anexar o RefEx.

Dependendo das opções nas etapas 5, 6 e 7, você será solicitado a especificar o ponto de inserção, a escala e/ou o ângulo de rotação.

Notas:

- A variável de sistema XDWGFADECTL (controle de fade do banco de dados RefEx) define o nível de esmaecimento para as referências externas.
- O ponto de origem de uma referência externa é controlado pela variável de sistema INSBASE (ponto base de inserção). Use o comando BASE para definir a variável de sistema INSBASE no desenho de referência externa.

- O campo **Unidade** em **Unidade do Bloco** é definido pela variável de sistema INSUNIT (Unidades de Inserção) na referência externa. O campo **Fator** expressa a relação entre o valor da variável de sistema INSUNIT no RefEx e o desenho-pai.

12.16 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.

The XREF command (short for "external reference") attaches DWG files to the current drawing through the **Drawing Explorer**.

The -XREF command attaches DWG files to the current drawing through the Command line.

The XATTACH command (short for "external attach") attaches externally-referenced drawings through a dialog box.

The XCLIP command (short for "Xref clip") clips externally-referenced drawings, and adjusts front and rear clipping planes.

The XDWGFADECTL system variable controls the fading of external references. Values between 0 (no fading) and 90 are accepted. The fading factor affects the screen display only, not the printing.

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**.

The XREFNOTIFY system variable controls whether warnings for missing or modified Xrefs are issued.

12.17 Editar blocos e referencias externas

O comando EDITARREF (abreviação de "editar referência") edita referências de blocos e desenhos referenciados externamente (RefEx's) por meio de uma caixa de diálogo. O que o EDITARREF faz é conhecido como **edição no-local de referência**. O comando funciona em uma porção da referência ou no desenho inteiro, ou em todas as entidades do bloco.

O comando -EDITARREF edita referências de bloco e desenhos externamente referenciados, através da barra de Comando.

O comando CONJREF adiciona e remove entidades da referência que está sendo editada.

O comando FECHARREF fecha o editor de referência.

O comando EDITARBLOCO edita as entidades que formam blocos em uma sessão temporária de desenho chamada **Editor de Bloco**, que usa o sistema de coordenadas local do bloco.

O comando -EDITARBLOCO solicita um nome de bloco na linha de Comando, e abre o ambiente do **Editor de Bloco**.

A variável de sistema XFADECTL define o nível de esmaecimento para as entidades que não estão trabalhando na sessão de EditarRef atual. Valores entre 0 (sem esmaecer) e 90 são aceitos.

12.17.1 Para editar Refex's

1 Siga um dos procedimentos:

- Clique duas vezes no RefEx.
- Clique no botão **EDITARREF** () na barra de ferramentas **EditarRef**.
- Inicie o comando EDITARREF.

Você é solicitado: Selecione a referência:

Clique no RefEx ou no bloco.

A caixa de diálogo **Editar Referência** é exibida.

A caixa de diálogo ajuda você a identificar visualmente a referência para editar e controlar como a referência é selecionada.

- **Caminho**: Localização da referência selecionada; se a referência for um bloco nenhum caminho é exibido.
- **Nome de Referência**: O nome da referência atualmente selecionada e todas as demais referências, que estejam aninhadas dentro da referência selecionada.
- **Visualização** (Prévia): Exibe a referência como ela foi salva pela última vez no desenho. Observe que a imagem de visualização não é atualizada quando as alterações são salvas de volta na referência.

2 (opção) Se várias referências aninhadas forem exibidas na árvore **Nome de referência**, selecione uma referência a ser modificada.

Clique no botão expandir/recolher para mostrar/ocultar referências aninhadas.

Você pode editar apenas uma referência por vez.

3 Escolha uma das seguintes para **Selecionar entidades aninhadas...**:

- **Automaticamente**: Todas as entidades e blocos aninhados na referência selecionada são incluídas na sessão de edição da referência.
- **Solicitar**: Depois de fechar a caixa de diálogo **Editar Referência**, você será solicitado a selecionar as entradas e os blocos aninhados na referência que deseja editar. Somente essas entidades serão extraídas temporariamente e disponibilizadas para modificação dentro do

contexto do desenho atual.

Esse conjunto de entidades de trabalho temporário pode ser editado e salvo novamente para atualizar a referência que está sendo editada. Para distinguir 'desenho atual' de 'conjunto de edição', o BricsCAD exibe todas as entidades pertencentes ao desenho atual como esmaecidas. O nível de esmaecimento é controlado por meio da variável de configurações XFADECTL, que tem um valor padrão de **50**.

4 Verifique as **Configurações**:

- **Habilitar nomes de layer e símbolo exclusivos**:
  - Se for selecionado, objetos nomeados nos RefEx's são alterados, adicionando um prefixo **$#$**.
  - Se estiver desmarcada, nomes de camadas e outros objetos nomeados permanecem os mesmos como no desenho original referenciado.
- **Mostrar as definições de atributo para edição**: Controla se as definições de atributo em blocos são extraídas e exibidas durante a edição da referência.
- **Bloquear entidades não no conj. de trabalho**: As entidades na referência que está sendo editada que não estão incluídas no conjunto de trabalho temporário não podem ser manipuladas, nem as entidades no desenho hospedeiro ou outras RefEx's. Esse comportamento é semelhante aos objetos em camadas bloqueadas.

5 Clique no botão **OK** para começar a editar o conjunto de trabalho.

A caixa de diálogo **Editar Referência** é fechada.

6 (opção) Clique no botão **Adicionar ao EditarRef** () para adicionar entidades ao conjunto de trabalho.

As entidades selecionadas serão removidas do desenho hospedeiro e adicionadas à referência que está sendo editada, quando o conjunto de trabalho é salvo novamente.

7 (opção) Clique no botão **Remover de EditarRef** () para remover entidades do conjunto de trabalho.

As entidades selecionadas serão removidas da referência que está sendo editada quando a lista de trabalho for salva de volta. Essas entidades são adicionadas ao desenho hospedeiro.

8 Siga um dos procedimentos:

- Clique o botão **FecharRef e Salvar** () para salvar as alterações de volta para o desenho origem de RefEx ou a definição de bloco no desenho atual.
- Clique no botão **FecharRef e descartar** () para interromper a sessão de EditarRef sem salvar. O desenho de origem ou a definição do bloco são deixados inalterados.

12.18 Trabalhar com grupos

O comando GRUPO cria e modifica grupos nomeados de entidades, em uma caixa de diálogo.

O comando -GRUPO cria e modifica grupos nomeados de entidades, usando a linha de Comando.

Trabalhar com grupos é uma técnica para manipular um conjunto de vários objetos como se fosse um único objeto. Um grupo também pode ser considerado como uma 'Seleção Nomeada'. Você pode escolher um único objeto e todo o conjunto é selecionado como se fosse um bloco, ou você pode editar objetos

individuais sem afetar o resto do grupo e sem dividir o grupo. Você pode usar grupos aninhados, se necessário, e também uma entidade isolada pode ser membro de grupos diferentes.

Nota: Certifique-se de que o bitcode 1 da variável de sistema PICKSTYLE esteja definido (valor = 1 ou 3) para habilitar a seleção de grupos.

12.18.1 Para criar um grupo

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Grupo...** () na barra **Ferramentas** ou no painel **Home / Grupos** da Faixa de opções.
- Escolha **Grupo...** no menu **Ferramentas**.
- Inicie o comando GRUPO.

A caixa de diálogo **Agrupar Entidades** é exibida:

2 Siga um dos procedimentos:

- Digite um nome no campo **Nome**.
- Ative a opção **Sem Nome** para criar um grupo não nomeado.

3 (opcional) Digite uma descrição no campo **Descrição**.

4 (opcional) Marque / desmarque a opção **Selecionável**.

5 Clique no botão **Selecionar entidades e criar grupo**.

A caixa de diálogo **Agrupar Entidades** fecha temporariamente para permitir que você selecione entidades.

6 Selecione as entidades que deseja incluir no novo grupo.

7 Clique o botão-direito do mouse para concluir a seleção de entidades.

8 A caixa de diálogo **Agrupar Entidades** é reaberta.

O grupo recém-criado é adicionado.

9 Clique o botão **OK** para fechar a caixa de diálogo **Agrupar Entidades**.

12.18.2 Para modificar um grupo

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Grupo...** () na barra **Ferramentas** ou no painel **Home / Grupos** na Faixa de opções.
- Escolha **Grupo...** no menu **Ferramentas**.
- Inicie o comando GRUPO.

A caixa de diálogo **Agrupar Entidades** é exibida.

2 Selecione o grupo que você deseja modificar.

3 (opção) Dê um duplo-clique no campo **Nome** para editar o nome do grupo.

4 (opção) Clique duas vezes no campo **Descrição** para editar a descrição do grupo.

5 (opção) Clique na coluna **Selecionável** para ligar ou desligar a opção **Selecionável**.

6 (opção) Clique no botão **Adicionar objetos ao Grupo** para adicionar novas entidades ao grupo.

A caixa de diálogo **Agrupar Entidades** fecha temporariamente para permitir que você selecione as novas entidades do grupo.

7 (opção) Clique no botão **Remover objetos do Grupo** para remover entidades do grupo.

A caixa de diálogo **Agrupar Entidades** fecha temporariamente para permitir que você selecione as entidades.

8 Clique o botão **OK** para fechar a caixa de diálogo **Agrupar Entidades**.

12.18.3 Para desagrupar entidades

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Grupo...** ( ) na barra **Ferramentas** ou no painel **Home / Grupos** da faixa de opções.
- Escolha **Grupo...** no menu **Ferramentas**.
- Inicie o comando GRUPO.

A caixa de diálogo **Agrupar Entidades** é exibida.

2 Selecione o grupo (ou grupos) que deseja excluir.

3 Clique no botão **Desagrupar Grupos selecionados**.

Os grupos selecionados são removidos.

Nota: Quando você desagrupa entidades, o grupo é excluído, mas as entidades permanecem no desenho.

12.18.4 Para alterar a ordem das entidades

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Grupo...** ( ) na barra **Ferramentas** ou no painel **Home / Grupos** da Faixa de opções.
- Escolha **Grupo...** no menu **Ferramentas**.
- Inicie o comando GRUPO.

A caixa de diálogo **Agrupar Entidades** é exibida.

2 Clique no botão **Reordenar entidades**.

A caixa de diálogo **Ordenar Grupo** é exibida:

3 (opção) Clique no botão **Destacar** para ver a posição atual de cada entidade.

A caixa de diálogo **Agrupar Objetos** é aberta.

4 (opção) Clique o botão **Ordem Inversa** para inverter a ordem de todas as entidades no grupo.

5 (opção) Para alterar a ordem de uma entidade isolada faça o seguinte:

- No campo **Remover da posição**, digite a posição atual da entidade que você deseja mover.
- No campo **Entre novo número de posição para a entidade**, digite a nova posição.

6 (opção) Digite o número de objetos você deseja reordenar no campo **Número de entidades**.

7 Clique no botão **Reordenar**.

Uma caixa de mensagem confirma que o grupo foi reordenado.

12.19 Trabalhando com subjacências

Arquivos PDF podem ser anexados como uma subjacência a um arquivo de desenho. Assim como arquivos de imagem raster e referências externas (RefEx's), uma subjacência PDF não faz parte do desenho, mas está a este vinculada. O caminho para a subjacência é salvo no desenho-pai. Você pode editar o caminho para garantir que o PDF correto seja encontrado. Por padrão, a pasta do desenho-pai é pesquisada em primeiro lugar. Se a subjacência PDF não for encontrada lá, a pasta do caminho salvo será pesquisada. Se o PDF subjacente também não for encontrado lá, o BricsCAD informará uma 'Referência ausente ou inválida' no desenho.

O comando ANEXARPDF permite ajustar as configurações de esmaecer, contraste e monocromático de uma subjacência PDF, pela barra de Comando.

Os comandos ANEXARPDF e -ANEXARPDF inserem um arquivo PDF como uma subjacência no desenho atual.

O comando CAMADASPDF permite controlar a exibição das camadas em uma subjacência PDF.

O comando CORTARPDF recorta a exibição de um PDF. Você pode escolher entre um limite poligonal ou retangular. Cada instância do mesmo arquivo PDF subjacente pode ter um limite diferente.

A variável de sistema PDFOSNAP permite capturar a geometria num PDF inserido se esta variável de sistema estiver **Ligada**.

A variável de sistema PDFNOTIFY controla se os avisos para arquivos de subjacência PDF ausentes ou modificados são emitidos.

A variável de sistema XNOTIFYTIME define a frequência de verificação dos recursos modificados em recursos externamente referenciados: RefEx's, imagens e anexos em PDF se XREFNOTIFY, IMAGENOTIFY e/ou PDFNOTIFY estiver **Ligado**.

Se o valor da variável de sistema FRAME = 3, a variável de sistema PDFFRAME controla a visibilidade dos quadros PDF. As opções são:

- 0: Ocultar quadros PDF.
- 1: Exibir e plotar quadros PDF.
- 2: Exibir, mas não plotar quadros do PDF

12.19.1 Controlando o armazenamento em cache de subjacências PDF

Um cache de imagem persistente de resolução múltipla é utilizado para exibir underlays PDF anexados, permitindo operações (muito) rápidas de zoom e pan. A resolução em cache mais alta é de 5000 x 5000 pixels. Ainda assim, ao ampliar muito próximo, a exibição do underlay PDF ficará pixelada. Portanto, um modo híbrido pode ser usado que muda para a geração em tempo-real de exibição de underlay PDF nítida ao ampliar muito próximo. A geração inicial do cache de imagem pode levar alguns segundos, a partir de então o processamento fica (muito) rápido e permanece assim nas sessões subsequentes.

A variável de sistema PDFCACHE controla como o armazenamento em cache de PDF é aplicado.

- 0 = não use cache de PDF, sempre use geração em tempo-real da imagem da subjacência PDF. (esse modo era o padrão nas versões anteriores do BricsCAD).
- 1 = usar cache de PDF, alterna para geração em tempo-real quando fizer um zoom muito próximo.
- 2 = sempre usar cache de PDF (padrão).

12.19.2 Anexar uma subjacência PDF

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Anexar PDF** ( ) na barra **Inserir** ou no painel **Inserir / PDFs** na Faixa de opções.
- Clique no botão **Novo** ( ) na barra **Detalhes** no diálogo **Explorer do Desenho - Subjacências PDF**.
- Digite **AnexarPDF** na linha de Comando, e pressione Enter.

O diálogo **Selecionar Arquivo Subjacência PDF** é aberto.

2 Selecione o arquivo e, em seguida, clique duas vezes no arquivo ou clique no botão **Abrir** na caixa de diálogo **Selecionar Arquivo de Subjacência PDF**.

A caixa de diálogo **Anexar Subjacência PDF** é exibida.

3 No caso de um PDF com múltiplas páginas, clique na seta para baixo, no campo **Nome**, e selecione a página do PDF que deseja carregar.

4 Clique no botão de opção **Caminho** e selecione:

- **Caminho completo**: O caminho completo será pesquisado quando o desenho é carregado. Se a sobreposição não for encontrada lá, a pasta do desenho-pai será pesquisada. Se o arquivo da subjacência também não for encontrado, a mensagem "Referência ausente ou inválida" será exibida no ponto de inserção da subjacência PDF.
- **Caminho relativo**: O caminho relativo, em relação à pasta do desenho-pai, será pesquisado quando o desenho é carregado. Se a sobreposição não for encontrada lá, a pasta do desenho-pai será pesquisada. Se o arquivo de sobreposição também não for encontrado, "Referência ausente ou inválida" será exibida no ponto de inserção da subjacência PDF.
- **Sem caminho**: Somente a pasta do desenho-pai é pesquisada quando o desenho é carregado. Se o arquivo de sobreposição não for encontrado aqui, "Referência ausente ou inválida" será exibido no ponto de inserção da subjacência PDF.

5 Escolha se você deseja especificar ou não **Ponto de Inserção**, **Escala** e o ângulo de **Rotação** na tela.

6 Clique o botão **OK**.

A caixa de diálogo **Anexar Subjacência PDF** é fechada.

7 Dependendo das opções de inserção escolhidas na etapa anterior, **você será solicitado** a especificar o **Ponto de Inserção**, **Escala** e/ou ângulo de **Rotação**.

12.19.3 Inserir uma subjacência PDF

Se você deseja inserir um segundo exemplar de um arquivo PDF que já esteja anexado, ou uma outra página do mesmo arquivo, faça o seguinte:

1 Inicie o comando EXPPDFS.

1 A caixa de diálogo **Explorer do Desenho - Subjacências PDF** é exibida.

2 Selecione o arquivo PDF.

3 Siga um dos procedimentos:

- Clique o botão-direito do mouse e escolha **Inserir** no menu de contexto.
- Escolha **Inserir** no menu **Editar** na caixa de diálogo **Explorer do Desenho - Subjacências PDF**.

A caixa de diálogo **Explorador de Desenho - Subjacências PDF** é fechada.

4 Continue com as etapas 3 a 7 do procedimento anterior.

12.19.4 Controlar a exibição de uma subjacência PDF

1 Clique no quadro da subjacência PDF.

1 O quadro da subjacência PDF se destaca.

As propriedades da subjacência são exibidas no painel **Propriedades**.

2 Em **Diversos**, clique em **Mostrar subjacência** na barra **Propriedades** e selecione **Sim** ou **Não**.

12.19.5 Definir a camada de exibição em uma subjacência PDF

1 Inicie o comando CAMADASPDF.

1 **Você é solicitado**: Selecione subjacência PDF.

2 Clique no quadro da subjacência PDF.

3 O quadro da subjacência PDF se destaca.

A caixa de diálogo **Camadas da Subjacência** é exibida.

4 O ícone na coluna **Liga** indica o estado atual das camadas; **Liga** (💡) ou **Desl** (💡).

5 Clique o ícone na coluna **Liga** para alternar (ligar/desligar) a exibição de uma camada.

6 (opção) Clique no campo **Pesquisar camada**, e digite um nome de camada para pesquisar uma camada.

Use caracteres curinga (? ou *) para limitar o número de camadas na lista.

7 (opção) Clique no botão **Nome da referência** para selecionar outra subjacência PDF.

8 Clique no botão **OK** para parar.

12.19.6 Recortar uma subjacência PDF

1 Inicie o comando CORTARPDF.

1 **Você é solicitado**: Selecione a subjacência PDF:

2 Clique no quadro da subjacência PDF.

3 O quadro da subjacência PDF se destaca.

Você é solicitado: Entre a opção de recorte de PDF [Liga/Desl/Excluir/Novo] <Novo>:

4 Pressione Enter para aceitar a opção de comando padrão **Novo**.

Você é solicitado: Digite o tipo de recorte de PDF [Poligonal/Retangular] <Retangular>:

5 Siga um dos procedimentos:
- Pressione Enter para aceitar a opção padrão de comando **Retangular** para definir um limite retangular de recorte.
- Digite P, e pressione Enter para definir um limite poligonal de recorte.

6 Definir o limite de recorte

7 Os pontos definidores devem estar dentro do quadro da subjacência PDF. Se você clicar fora da subjacência PDF o ponto será colocado no quadro da subjacência PDF.

12.19.7 Alternar a exibição da parte recortada de uma subjacência PDF

1 Clique no quadro da subjacência PDF.

1 O quadro da subjacência PDF se destaca.

As propriedades da subjacência são exibidas no painel **Propriedades**.

2 Em **Diversos**, clique em **Mostrar recortado** na barra **Propriedades**, e selecione **Sim** ou **Não**.

12.19.8 Excluir um limite de recorte

1 Inicie o comando CORTARPDF.

1 **Você é solicitado**: Selecione subjacência PDF.

2 Clique no quadro da subjacência PDF.

3 O quadro da subjacência PDF se destaca.

Você é solicitado: Entre a opção de recorte de PDF [Liga/Desl/Excluir/Novo] <Novo>:

4 Digite o atalho na barra de Comando e pressione Enter.

5 O limite de recorte é excluído, e o arquivo PDF completo é exibido.

12.19.9 Ajustar as configurações de exibição de uma subjacência PDF

O comando AJUSTARPDF permite ajustar as configurações para **Desvanecer**, **Contraste** e **Monocromático** de uma subjacência PDF na barra de Comando.

Selecione a subjacência PDF e, em seguida, ajustar as configurações de exibição sob **Ajustar Subjacência** no painel **Propriedades**.

12.20 Componentes e inserções de componentes

Um componente é um grupo nomeado de entidades, que pode ser inserido em um desenho com o comando BmInsert. Os componentes podem ser paramétricos: por exemplo, janelas e portas no BricsCAD BIM, e componentes mecânicos padrão no BricsCAD Mechanical.

Qualquer arquivo *.dwg pode ser usado como um componente. O mesmo componente pode ser inserido uma ou várias vezes em um ou mais outros componentes. Quando o componente é modificado, todas

as inserções do componente refletirão automaticamente as alterações. Os componentes não podem ser auto-referenciados.

Uma inserção de componente é uma entidade nomeada. O nome padrão de uma inserção de componente é composto pelo nome do componente e pelo número de série da inserção. Internamente, uma inserção de componente é representada como uma referência a um bloco anônimo ou como uma referência externa.

Propriedades de componentes inseridos, incluindo seus parâmetros podem ser vistos e editados no **Navegador de Mecânica** ou no painel **Propriedades**.

12.20.1 Componentes externos e locais

Os componentes podem ser locais ou externos. Um componente local é armazenado no arquivo *.dwg com a montagem, como um bloco anônimo. Um componente externo é definido em um arquivo *.dwg separado e representado por um RefEx ou como uma cópia local armazenada como um bloco anônimo. Quando o arquivo *.dwg correspondente é modificado, todas as inserções de componentes externos podem ser atualizadas usando o comando BMUPDATE ou o menu de contexto do **Navegador de Mecânica**. Se você usar componentes externos em sua montagem, lembre-se sempre de transferir todos os arquivos relacionados quando quiser compartilhar seu modelo.

Use o comando ETRANSMIT para criar um pacote de um arquivo de desenho e todas as suas dependências, como referências externas, imagens, arquivos de fonte, arquivos de configuração de plotagem, tabelas de estilo de plotagem e arquivos de mapa de fontes.

Você sempre pode converter um componente Local para Externo, e vice-versa.

Para editar um componente local, ele primeiro precisa ser convertido em um componente externo usando o comando BMEXTERNALIZE. Como alternativa, você pode usar o comando BMOPENCOPY para abrir uma cópia de um componente, editar e salvá-lo. Em seguida, use o comando BMREPLACE para substituir o original pela cópia.

Você pode escolher o tipo de inserção padrão para o componente: **Local** ou **Externo**. Para alterar o tipo, selecione o nó raiz no **Navegador de Mecânica** e selecione **Local** ou **Externo** em **Inserir como propriedades**.

Dependendo do valor de BMAUTOUPDATE, as inserções de componentes externos são atualizadas automaticamente ao abrir o documento da montagem, ou manualmente com o comando BMUPDATE. Você pode alterar esse valor na caixa de diálogo **Configurações**. As opções são:

- Atualiza somente quando o comando BMUPDATE for usado.
- Atualiza automaticamente ao abrir o arquivo.

12.20.2 Converter componentes

O comando BMEXTERNALIZE converte componentes locais em componentes externos.

O comando BMLOCALIZE converte componentes externos em componentes locais.

Siga um dos procedimentos:

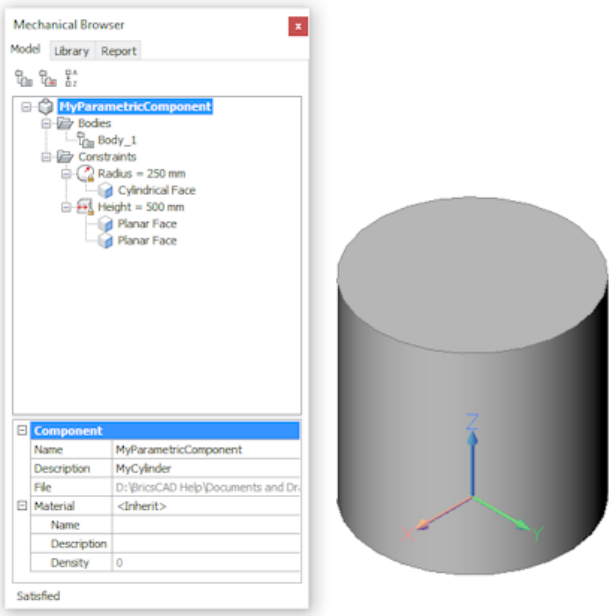
- Clique em **Trocar Componente para Externo** ( ) ou **Alternar componente para Local** na aba **Montagem** da Faixa de opções. As ferramentas também estão disponíveis na barra de ferramentas **Montagem** e no menu **Montagem**.
- Clique o botão-direito sobre o componente inserido no **Navegador de Mecânica** e selecione **Trocar para Externo** ou **Trocar para Local** no menu de contexto.
- Passe o cursor do mouse sobre uma inserção de componente em uma janela de desenho e selecione **Trocar para Externo** ( ) ou **Alternar para Local** no grupo de comandos **Projeto de Montagem** no Quad.

Quando você converte um componente externo em um local, o arquivo *.dwg com a definição do componente não é excluído, mas alterações futuras no componente externo não irão afetar a montagem.

Você sempre pode distinguir entre inserções de componentes locais ( ) e inserções de componentes externos ( ) pelos seus ícones no **Navegador de Mecânica**.

12.20.3 Componentes Paramétricos

Você pode fazer alterações paramétricas dos componentes inseridos no nível da montagem. Um componente é considerado paramétrico se contiver pelo menos uma restrição dimensional 2D ou 3D ou uma matriz associativa com uma expressão.



Notas:

- O BricsCAD mantém automaticamente a associatividade entre a definição de um componente paramétrico e suas inserções. Assim, você pode abrir o componente paramétrico para edição, modificar sua geometria e salvá-lo. Para ver o resultado, execute o comando BmUpdate para atualizar a estrutura mecânica da sua montagem.

- Se você precisa voltar ao valor padrão do parâmetro de um componente, basta editar o campo **Expressão** e excluir o seu conteúdo.

Você pode controlar se um parâmetro de componente específico estará disponível para edição em um desenho que contenha as inserções desses componentes (no nível da montagem). Para fazer isso, selecione um parâmetro no **Navegador de Mecânica** e altere o valor do campo **Exposto** na grade de propriedades do parâmetro.

As opções são:

- **Desl**: Oculta os parâmetros na montagem.
- **Liga**: Torna os parâmetros visíveis na montagem.
- **Auto**: Oculta todos os parâmetros cujos valores são controlados por expressões, e expõe todos os outros parâmetros.

Você pode associar unidades com um parâmetro. Se você atribuir um valor, diretamente ou por meio de uma expressão, a um parâmetro de componente no nível da montagem, este vai converter automaticamente das unidades de desenho da montagem para as unidades do parâmetro. Os valores atribuídos a parâmetros sem-unidade não são convertidos.

Por padrão, todos os parâmetros são sem-unidade, exceto parâmetros de restrições dimensionais, e parâmetros que são usados como expressões para restrições dimensionais. O BricsCAD atribui automaticamente unidades de desenho a esses parâmetros.

12.20.4 Para editar os parâmetros de uma inserção de componente

Quando um componente paramétrico é inserido em um modelo, você pode editar os parâmetros tanto no **Navegador de Mecânica** ou no painel **Propriedades**.

A inserção do componente é atualizada imediatamente de acordo com o novo valor de um parâmetro. Observe que isso não modifica a definição do componente paramétrico, que é um arquivo de desenho separado. Como resultado, múltiplas inserções do mesmo componente paramétrico podem existir no mesmo modelo, com valores diferentes para cada um de seus parâmetros:

Para editar um parâmetro no Navegador de Mecânica

1 Expanda o nó **Parâmetros** do componente.

2 Selecionar o parâmetro.

3 Na seção **Parâmetro**, digite um novo valor no campo **Expressão**.

Para editar um parâmetro na Barra Propriedades.

1 Selecione o componente no desenho.

2 Expanda o grupo de configurações **Parâmetros**.

3 Digite um novo valor para um dos parâmetros disponíveis.

Notas:

- No painel **Propriedades**, o valor atual é exibido. Se esse valor for controlado por meio de uma expressão, a expressão será substituída pelo valor recém-inserido.

- Se vários componentes paramétricos forem selecionados, os parâmetros compartilhados poderão ser editados simultaneamente. Os campos de configurações dos parâmetros que atualmente têm valores diferentes são expressos como: *varia*.

12.20.5 Para substituir componentes

O comando BMREPLACE substitui as inserções de componentes. Somente inserções de nível superior (top-level) podem ser substituídas.

Nota: Valores dos parâmetros que são compartilhados entre o componente de substituição e o componente substituído são copiados automaticamente.

12.20.6 Componentes, blocos, ou referências externas?

Os componentes são baseados em blocos e referências externas (RefEx's), mas o comportamento dos componentes é diferente. Um componente externo está vinculado a um arquivo, como um RefEx, mas um componente externo pode ser representado por um RefEx (BMINSERT neste caso age como um ANEXAREFEX) ou por um bloco anônimo local (BMINSERT age como INSERIR).

Componentes podem ser tanto peças isoladas como montagens:

- Peça (isolada): composta apenas de entidades; sem inserções de subcomponentes.
- Montagem: composta por entidades e inserções de subcomponentes. Mesmo quando os subcomponentes são removidos, o status da montagem é mantido.

Quando um componente externo é uma peça e seus parâmetros não são modificados, o componente é inserido como uma RefEx. Isso está listado na seção **Referências Externas** da caixa de diálogo **Explorer do Desenho**. As camadas RefEx têm um prefixo 'nome_arquivo'.

Quando um componente externo é uma montagem, este é inserido como um bloco anônimo local. Todas as suas camadas são mescladas com as camadas do documento principal e nenhum prefixo é adicionado ao nome da camada.

12.20.7 Converter blocos e referências externas

O comando BMMECH converte o desenho atual em um componente mecânico. Se o desenho já é um componente mecânico, o BMMECH não faz nada.

Referências de bloco são convertidas em inserções de componentes locais enquanto as referências externas são convertidas em inserções de componentes externos.

O comando BMUNMECH converte o componente mecânico atual em um desenho simples. Componentes mecânicos são convertidos de volta em blocos e referências externas. O comando se aplica a desenhos que são somente um componente mecânico.

12.20.8 Recursos baseados em componentes

Um recurso é um constituinte físico de uma peça com significado de engenharia. Exemplos de recursos são: furos, nervuras, ranhuras, cavidades, formas. Se aplicado a um sólido 3D, um recurso remove e/ou adiciona geometria. Para criar seus próprios recursos, use a tecnologia de extensão de componentes do BricsCAD.

Um componente estendido tem camadas BC_SUBTRACT e/ou BC_UNITE. Quando você usa o comando BMINSERT para inserir esse componente na face de um sólido 3D, o sólido de destino é modificado da seguinte forma:

- Todos os sólidos 3D localizados na camada BCSUBTRACT do componente são subtraídos do sólido 3D de destino;
- Todos os sólidos 3D localizados na camada BC_UNITE do componente são unidos com o sólido 3D destino.

Todas as faces criadas após operações Booleanas formam um recurso baseado em componentes. Eles são atualizados quando você move a inserção do componente ou quando você modifica qualquer parâmetro. Quando a inserção é excluída, o sólido de destino é reconstituído da modificação. Ajustar os sólidos 3D de destino com o comando BMLINK.

Alguns exemplos de recursos paramétricos baseados em componentes estão incluídos no BricsCAD. Você pode usá-los como exemplos para criar seus próprios recursos.

12.20.9 Atribuir materiais físicos aos componentes

Você pode atribuir um material físico a um componente mecânico. Se um componente mecânico não tiver um material atribuído, este herdará o material de seu componente-pai. Se o componente parental não tiver um material atribuído, o material do componente mais próximo, será usado. Como resultado, o mesmo material pode ser atribuído a todos os componentes que têm um 'pai' comum.

1 Selecionar o componente-raiz no **Navegador de Mecânica**.

2 Selecione a propriedade **Material** e pressione o botão **Procurar** (...).

3 Na caixa de diálogo **Materiais Físicos**, selecione um material na lista **No Projeto** ou **Na biblioteca**.

Os materiais que são escolhidos **Na Biblioteca** são automaticamente copiados para o banco de dados **No Projeto**.

4 Clique no botão **OK** para atribuir o material selecionado ao componente.

Para remover o material do componente:

1 Selecionar o componente-raiz no **Navegador de Mecânica**.

2 Selecione a propriedade **Material** e pressione o botão **Limpar** (X).

Inserindo um componente com um material físico:

Quando você insere um componente mecânico em um modelo, todos os seus materiais são copiados para a biblioteca de materiais do modelo. Se um material com o mesmo nome já existe no modelo de destino, esse material será usado. Como resultado, as propriedades do material no modelo de destino são usadas. Isso substitui as propriedades do material do componente inserido.

Nota: Materiais que forem copiados para a biblioteca de materiais do documento não serão apagados em operações de **Desfazer**.

13. Gerenciar imagens

13.1 Managing images

The EXPIMAGES command displays the Drawing Explorer – Images dialog box.

The IMAGE command opens the Attachments panel.

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

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; sets the value of the IMAGEFRAME system variable.

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.

You can manage images in the **Drawing Explorer - Images** dialog box and in the **Attachments** panel.

Here you can:

- Attach an image () or ().
- Load / Unload inserted images.
- Detach images ().
- Edit the path to the image.
- See a preview of the attached images (**Drawing Explorer** dialog box only).

To open the **Drawing Explorer - Images** dialog box, do one of the following:

- Choose **Drawing Explorer** > **Images** in the **Tools** menu.
- Select **Images** in the **Open Drawings** sub-window of the **Drawing Explorer** dialog box.
- Launch the EXPIMAGES command.

To open the **Attachments** panel, do one of the following:

- Click the **Image Management** tool button () in the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGE command.

13.1.1 Attaching an image

1 Do one of the following:

- Click the **New** tool button () in the **Drawing Explorer - Images** dialog box.
- Click the **Attach Raster Image** tool button () in the **Insert** toolbar or the **Insert / Images** ribbon panel.

- Launch the IMAGEATTACH command.

The **Select Image File** dialog box appears.

2 Select the image file, then double-click or click the **Open** button.

The **Attach Raster Image** dialog box appears.

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 Specify an **insertion point** or select the **Specify on-screen** check box.

5 Specify a **scale** or select the **Specify on-screen** check box.

6 Specify a **rotation** angle or select the **Specify on-screen** check box.

7 Click the **OK** button.

8 If none of the **Specify on-screen** options are 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.

13.1.2 Loading and unloading images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily hide it. Unloaded images are still inserted in the drawing, but they are no longer displayed. If the **Imageframe** property is ON, the frame around the image is still displayed.

To load / unload a single image

Click the **Load** column of the image in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

To load / unload multiple images

1 Hold down the Ctrl key, then select the images in the **Attachments** panel or the **Drawing Explorer – Images** dialog box. Click a selected image to de-select.

2 Click the **Load** column of a selected image.

13.1.3 Detaching images

Since an image is first attached, then inserted in the drawing, you can delete an inserted image, in the drawing without detaching it.

To detach an image:

1 Select the image(s) in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

Hold down the Ctrl key to select multiple images; click a selected image to de-select.

2 Do one of the following:

- Click the Delete or Detach (✕) tool button.
- Right-click, then choose **Delete** or **Detach** in the context menu.

13.2 Image utilities

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The TRANSPARENCY command toggles the transparency of images that have a transparent alpha channel. The alpha channel is a special channel that handles transparency. When an image has an alpha channel on it, you can adjust the image's opacity levels and make bits translucent or totally see-through.

Nota: When the image frames do not appear (IMAGEFRAME = 0), you must use a window selection method to select an image.

13.2.1 Clipping an image

1 Do one of the following:

- Click the **Clip Image** tool button () on the **Image** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGECLIP command.

You are prompted: Select image:

2 Select the image.

3 **You are prompted**: Enter image clipping option [ON/OFf/New boundary] <New boundary>:

4 Press Enter to accept the **New Boundary** default option.

5 **You are prompted**: [Select polyline/Polygonal/Rectangular] <Rectangular>:

Press Enter to accept the **Rectangular** default option.

6 **You are prompted**: Specify first corner.

7 Click a point to define the first corner of the clipping rectangle.

8 The clipping rectangle is dynamically displayed.

You are prompted: Specify opposite corner:

9 Click a point to define the clipping rectangle.

10 The image is clipped.

13.2.2 Modifying the Image Frame setting

Using the Image Frame tool (IMAGEFRAME command):

1 Do one of the following:

- Click the **Image Frame** tool () on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGEFRAME command.

You are prompted: Enter image frame setting <current value>:

2 Do one of the following:

- Type 0 to hide image frames.
- Type 1 to display and plot image frames.

- Type 2 to display but not print image frames.

Editing the IMAGEFRAME system variable:

1 Open the **Settings** dialog box.

2 Go to **Drawing / Underlays**.

3 Set the FRAME system variable = 3: Use individual settings.

4 Select the IMAGEFRAME system variable.

5 The options are:

- 0: Hide image frames.
- 1: Display and plot image frames.
- 2: Display but do not plot image frames.

13.2.3 Setting the image transparency

1 Do one of the following:

- Click the **Image Transparency** tool button ( ) on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the TRANSPARENCY command.

You are prompted: Select images.

2 Select the images you want to set the **Transparency** property of.

You are prompted: Enter transparency mode [ON/OFF]<current mode>:

3 Do one of the following:

- Press Enter to accept the current mode.
- Choose **On** to switch the **Transparency** property of the selected images on.
- Choose **Off** to switch the **Transparency** property of the selected images off.



Transparency of the camera image: OFF (left) and ON (right)

13.2.4 Setting the display properties of an image

1 Select an image.

2 The properties of the image appear in the **Properties** panel.

3 Go to the **Misc** properties category.

4 The following properties can be set **Yes** or **No**:

- **Show Image**: shows or hides the image.
- **Show clipped**: sets the clipping boundary on/off.
- **Clip inverted**: inverts the clipping boundary.
- **Background transparency**: sets the transparency on/off.

14. Anotar

14.1 Dimensions

14.1.1 Visão geral das ferramentas de Dimensionamento

Ferramentas de dimensionamento podem ser encontradas em:

- Na barra de ferramentas **Dimensões**.
- No menu **Dimensões**.
- No painel da Faixa de opções **Anotar / Dimensões**.
- Na aba **Desenhar** do Quad **Sem Seleção**.

Ícone	Ferramenta	Comando
	Linear	COTALINEAR
	Alinhado	COTAALIN
	Arco	COTAARCO
	Raio	COTARAIOS
	Diâmetro	COTADIAMETRO
		COTADESASSOCIAR
		REASSOCIARCOTA
	Angular	COTAANG
	Base	COTALINBASE

Ícone	Ferramenta	Comando
	Continuar	COTASERIE
	Ordenada	COTAORDEN
	Rotacionada	COTALINEAR
	Oblíqua	EDITARCOTA + O
	Multichamada	LINCHMULT
	Editar Multi-chamada	EDITARLINCHMULT
	Multi-chamada adicionar chamadas	AIMLEADEREDITADD
	Multi-chamada remover linhas de chamada	AIMLEADEREDITREMOVE
	Tolerância	TOLERANCIA
	Linhas de Centro	COTACENTRO
	Editar Texto de Dimensão	EDITARCOTA + E
	Rotacionar Texto da Cota	EDITARCOTA + R
	Reposicionar Texto de Dimensão	EDTXTCOTA

Ícone	Ferramenta	Comando
	Restaurar Posição do Texto	EDITARCOTA
	Aplicar Estilo	-ESTILOCOTA + A
	Salvar Estilo	-ESTILOCOTA + A
	Restaurar Estilo	-ESTILOCOTA
	Status das Variáveis de Dim	-ESTILOCOTA + A
	LinhaChamada	LINHACHAMCOTA
	Chamada Rápida	LINHACHAMR

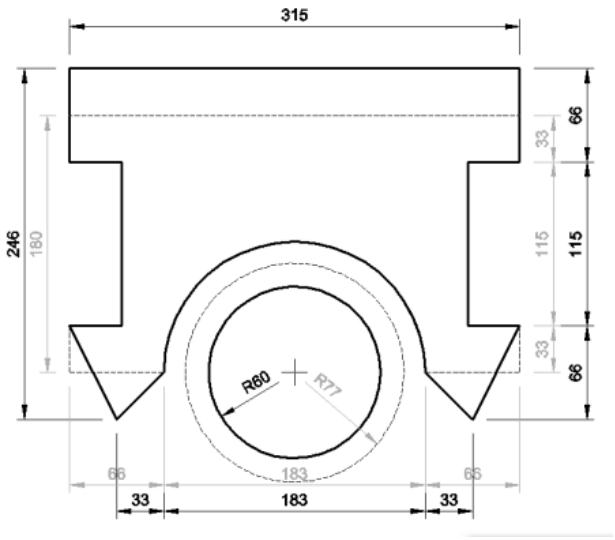
14.1.2 Dimensoes associativas

As dimensões associativas são atualizadas automaticamente quando a entidade dimensionada é movida ou modificada.

O BricsCAD cria dimensões associativas se o valor da variável de sistema DIMASSOC for 2 (padrão).

A propriedade associativa das dimensões também se aplica a dimensões que são colocadas no Paper Space e estão associadas a entidades do Model Space.

Se necessário, você pode usar o comando COTAREGEN para atualizar todas as dimensões associativas.



Os seguintes Snaps de Entidade podem ser usados para criar dimensões associativas: **Extremidade**, **PtoMédio**, **Centro**, **Perpendicular**, **Quadrante**, **Inserção**, **Ponto**, **Interseção** e **Interseção aparente**.

O comando COTADESASSOCIAR remove a associatividade das entidades de dimensão selecionadas.

O comando REASSOCIARCOTA reassocia ou associa dimensões a entidades ou pontos em entidades.

A variável do sistema GENERATEASSOCVIEWS permite a criação de dimensões associativas para vistas geradas pelos comandos VISTABASE e VISTACORTE e desenhos gerados pelo comando BIMCORTEATUALIZAR.

14.1.3 Configuração de Dimensões

Configurações de dimensão são salvas em um estilo de dimensão. Entidades de dimensão são criadas usando as configurações no estilo de dimensão atual.

O comando ESTILOCOTA cria e modifica estilos de dimensão, por meio do **Explorer do Desenho**.

O comando -ESTILOCOTA cria e modifica estilos de dimensão pela linha de Comando.

Um estilo de cotas consiste em seis grupos de configurações

- **Linhas e Setas**
- **Texto**
- **Ajustar**
- **Unid. Primárias**
- **Unid. Alternativas**
- **Tolerâncias**

14.1.3.1 Para listar o status atual das variáveis de dimensão.

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Status da Variável de Dimensão** ( ) na barra de ferramentas **Dimensões**.
- Escolha **Status da Variável de Dimensão** no menu **Dimensões**.

Uma lista de todas as variáveis de dimensão é exibida na barra de Comando.

2 Pressione a tecla de função **F2** para abrir a janela **Histórico de Prompt BricsCAD**.

3 Pressione Enter para continuar a listagem.

4 Pressione a tecla de função **F2** para fechar a janela **Histórico de Prompt BricsCAD**.

14.1.4 Working with dimension styles

The DIMSTYLE command opens the **Drawing Explorer – Dimension Styles** dialog box.

Here you can:

- Create a new dimension style
- Create dimension child styles
- Edit a dimension style
- Define dimension style overrides
- Rename a dimension style
- Set a dimension style current
- Copy dimension styles between drawings
- Compare dimension styles
- Delete a dimension style

14.1.4.1 To set a dimension style current

First method:

1 Right click the dimension style field in the **Status bar**.

A context menu appears.

2 Select the dimension style in the context menu.

Second method:

1 Launch the DIMSTYLE command.

1 The current dimension style is marked in the **Current** column.

2 Click the current column of the dimension style.

14.1.4.2 To create a new dimension style

1 Launch the DIMSTYLE command.

2 Select an existing dimension style, which will be used as a template for the new style.

3 Do one of the following:

- Click the **New** button.
- Place the cursor on an existing dimension style, then right click and choose **New** in the context menu.

4 Select the **Name** field and type a name for the new dimension style.

5 Define the settings for the new dimension style.

Nota: 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.

14.1.4.3 Creating 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.

To create a dimension child style, do the following:

1 Select the parent dimension style, then right click and choose a **New child style** in the context menu.

A context menu appears.



2 Select a dimension entity type in the context menu.

3 The name of the child style is: <parent style>:<dimension entity type>

4 Select the child style and define the properties that need to be different from the parent style for the selected dimension entity type.

14.1.4.4 To edit a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Adjust the dimension settings.

4 Close the **Drawing Explorer** dialog box.

14.1.4.5 To define dimension style overrides

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 The first line of the dimension style list reads: <overrides on dimstylename>

4 Define the overrides.

5 Close the **Drawing Explorer** dialog box.

Nota:

- 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 to save the overrides.

- Setting another style current will discard the existing dimension style overrides.

14.1.4.6 To rename a dimension style

1 Launch the DIMSTYLE command.

2 Select the dimension style.

3 Right click, then choose **Rename** in the context menu.

4 Type a new name in the **Name** field, then press enter.

14.1.4.7 To copy dimension styles between drawings

1 Open both the source and the target drawing.

2 Launch the DIMSTYLE command.

3 Open drawings are listed in the **Open drawings** list on the **Drawing Explorer** dialog box.

4 Select **Dimension styles** of the source drawing in the **Open drawings** list.

5 Select the dimension style(s) to be copied.

6 Hold down the Ctrl key to select multiple dimension styles.

7 Place the cursor over one of the selected dimension styles, then press and hold the left mouse button and drag the selection onto the target drawing, then release the left mouse button.

14.1.4.8 To compare dimension styles

1 Launch the DIMSTYLE command.

2 Select the dimension styles.

3 Click the first dimension style, then hold down the Ctrl key to select the next dimension style(s).

The properties of the selected styles display. The settings that are different, are highlighted.

4 (option) Hold down the Ctrl key to add or remove a dimension style from the comparison table.

14.1.5 Using the dimension style tools

The dimension style tools on the **Dimensions** toolbar and the **Dimensions** ribbon panel allow to:

- 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.

14.1.5.1 To apply the current dimension style

1 Do one of the following:

- Click the **Apply Style** tool ().
- Choose **Apply Style** in the **Dimensions** menu.
- Launch the -DIMSTYLE command, then type **a** or **apply** and press Enter.

2 **You are prompted**: Select dimensions to apply the current style.

3 Select the dimensions, then press Enter.

Nota: 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.

14.1.5.2 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.
- Launch the -DIMSTYLE command, 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 If the name already exists, the Command line prompts you to redefine it or not.

14.1.5.3 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.
- Launch the -DIMSTYLE command, 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.

14.1.5.4 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.
- Launch the -DIMSTYLE command, 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. Press the F2 function key to close the **Prompt History** window.

14.1.6 Creating linear dimensions

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.

The following commands create linear dimension entities:

- DIMLINEAR: Creates horizontal and vertical linear dimensions with respect to the current coordinate system (World or UCS).

- DIMALIGNED: Creates dimensions parallel to (aligned with) lines and polyline segments. The command also works with polyline arc segments, arcs, and circles, but dimensions their diameters (see DIMDIAMETER command).
- DIMBASELINE: Places multiple linear and angular dimensions, all starting at the same base point of an existing dimension. This command works only when at least one other dimension already exists in the drawing.
- DIMCONTINUE: Places a chain of dimensions by continuing linear or angular dimensions from the endpoint of the previous dimension or another existing dimension.
- DIMARC: Places dimensions that measure the lengths of arcs and polyline arc segments.

14.1.6.1 To create a horizontal or vertical linear dimension

1 Launch the DIMLINEAR command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

The dimension line is dynamically displayed.

Move the cursor in to create a horizontal dimension, or horizontally to create a vertical dimension.

3 Specify a point to position the dimension line.

14.1.6.2 To create an aligned linear dimension

1 Launch the DIMALIGNED command.

1 **You are prompted**: Origin of first extension line <Select entity>:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

The dimension line is dynamically displayed.

3 Specify a point to position the dimension line.

14.1.6.3 To create a rotated linear dimension

1 Do one of the following:

- Click the **Rotated** tool button on the **Dimensions** toolbar or ribbon panel.
- Choose **Rotated** in the **Dimensions** menu.

You are prompted: Origin of first extension line <Select entity>:Do one of the following:

2 Do one of the following:

- Specify the origin of the first extension line, then specify the origin of the second extension line.
- Press Enter, then select the line or linear segment of a polyline you want to dimension.

You are prompted: Angle of dimension line <0>:

3 Do one of the following:
   - Type the rotation angle in the Command line, then press Enter.
   - Enter the rotation angle by specifying two points.

   The dimension line is dynamically displayed.

4 Specify a point to position the dimension line.

14.1.6.4 To create baseline dimensions

1 Launch the DIMBASELINE command.

1 **You are prompted**: Baseline: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

2 Do one of the following:
   - Go to step 2 to accept the baseline dimension which is dynamically displayed, starting from the most recently added linear or angular dimension.
   - Press Enter, then select the starting dimension entity.

3 Specify a point to create the first baseline dimension.

4 Keep specifying points or press Enter to stop.

Nota:

- 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 DIMDLI system variable (Dim baseline spacing property in the dimension style).

14.1.6.5 To create continued dimensions

1 Launch the DIMCONTINUE command.

1 **You are prompted**: Continue: Origin of next extension line or [Select starting dimension/Undo] <Select starting dimension>:

2 Do one of the following:
   - Go to step 2 to accept the continued dimension which is dynamically displayed, starting from the most recently added linear or angular dimension.
   - Press Enter, then select the starting dimension entity.

3 Specify a point to create the first continued dimension.

4 Keep specifying points or press Enter to stop.

Nota:

- 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.

14.1.6.6 To create an arc length dimension

1 Launch the DIMARC command:

1 **You are prompted**: Select arc or polyline arc segment:

2 Select the arc or polyline arc segment.

3 The arc length dimension is dynamically displayed.

4 Specify a point to position the dimension line.

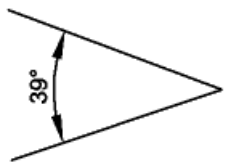
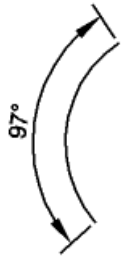
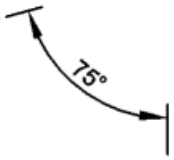
14.1.7 Criar dimensões angulares

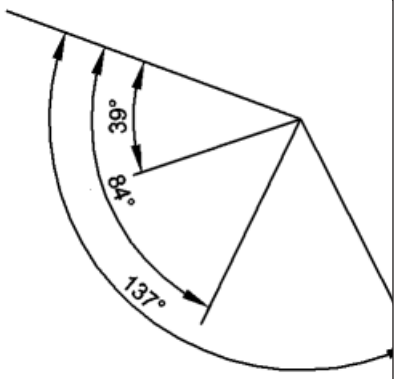
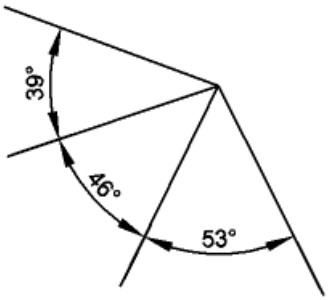
O comando COTAANG cria uma dimensão angular.

As dimensões angulares anotam o ângulo medido entre duas linhas, dois segmentos lineares de uma polilinha, ou o ângulo de um arco circular. Você também pode dimensionar um ângulo selecionando um vértice angular e duas extremidades.

A partir de uma dimensão angular existente, você pode adicionar uma dimensão de linha de base (acumulada) ou uma dimensão contínua.

- Uma dimensão de linha de base angular insere uma dimensão adicional a partir de uma primeira origem comum de linha de extensão.
- Uma dimensão angular contínua continua uma dimensão angular a partir da segunda linha de extensão, de uma dimensão angular anterior.

		
dimensão angular medida entre duas linhas	dimensão angular de um arco	dimensão angular definida por um vértice e dois pontos de extremidade

		
dimensões angulares em linha de base	dimensões angulares contínuas	

14.1.7.1 Para dimensionar um ângulo entre duas linhas

1 Inicie o comando COTAANG.

1 **Você é solicitado**: Selecione linha, arco ou círculo [opções de seleção (?)] <Pressione ENTER para especificar o ângulo>:

2 Selecione um segmento de linha ou polilinha.

3 **Você é solicitado**: Outra linha para dimensão angular [opções de seleção (?)] :

4 Selecione uma segunda linha, ou um segmento linear de polilinha.

5 A dimensão angular é exibida dinamicamente.

Você é solicitado: Localização do arco de dimensão [Angulo/Texto]:

6 Especifique um ponto para posicionar a dimensão.

7 Mova o cursor para dimensionar o ângulo interno entre as linhas ou seu suplemento.

14.1.7.2 Para dimensionar um ângulo circundado por um arco

1 Inicie o comando COTAANG.

1 **Você é solicitado**: Selecione linha, arco ou círculo [opções de seleção (?)] <Pressione ENTER para especificar o ângulo>:

2 Selecione um arco.

3 A dimensão angular é exibida dinamicamente.

Você é solicitado: Localização do arco de dimensão [Angulo/Texto]:

4 Especifique um ponto para posicionar a dimensão.

14.1.7.3 Para criar um dimensão angular definida por um vértice e duas extremidades

1 Inicie o comando COTAANG.

1 **Você é solicitado**: Selecione linha, arco ou círculo [opções de seleção (?)] <Pressione ENTER para especificar o ângulo>:

2 Pressione Enter.

3 **Você é solicitado**: Vértice do ângulo:

4 Especifique um ponto.

5 **Você é solicitado**: Primeiro lado do ângulo:

6 Especifique um ponto.

7 **Você é solicitado**: Outro lado do ângulo:

8 Especifique um ponto.

9 A dimensão angular é exibida dinamicamente.

Você é solicitado: Localização do arco de dimensão [Angulo/Texto]:

10 Especifique um ponto para posicionar a dimensão.

11 Mova o cursor para dimensionar o ângulo interno ou ângulo externo.

14.1.8 Criar dimensões diametrais e radiais

Os comandos COTACENTRO e COTADIAMETRO criam dimensões diamétricas e radiais para anotar os raios e diâmetros de círculos e arcos circulares.

14.1.8.1 Para criar uma dimensão de diâmetro

1 Inicie o comando COTADIAMETRO.

1 **Você é solicitado**: Selecione arco ou círculo para dimensionar [opções de seleção (?)] :

2 Selecione um arco ou um círculo.

3 A dimensão diamétrica é exibida dinamicamente.

4 Especifique um ponto para posicionar a entidade de dimensão.

14.1.8.2 Para criar uma dimensão de Raio

1 Inicie o comando COTARAIOS.

1 **Você é solicitado**: Selecione arco ou círculo para dimensionar [opções de seleção (?)] :

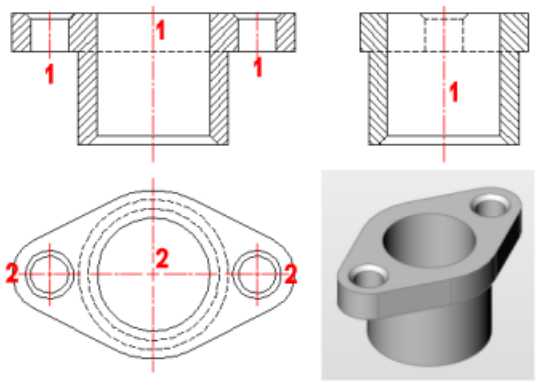
2 Selecione um arco ou um círculo.

3 A dimensão radial é exibida dinamicamente.

4 Especifique um ponto para posicionar a entidade de dimensão.

14.1.9 Centerlines and center marks

Centerlines and center marks are drawing reference entities to indicate axes of symmetry and 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.



1 Centerline

2 Center mark

The CENTERLINE command creates a centerline entity associated with two selected lines.

The CENTERMARK command creates a center mark entity associated with a selected circle or arc.

The CENTERREASSOCIATE command recreates the association between a centerline entity and two selected lines, or between a center mark and a selected circle or arc.

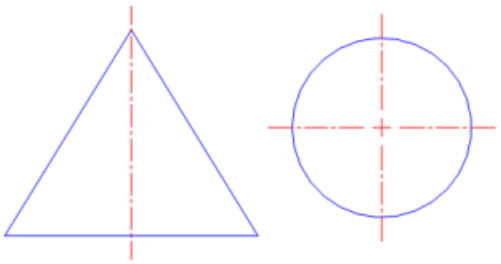
The CENTERDISASSOCIATE command breaks the association of centerline and a center mark entities.

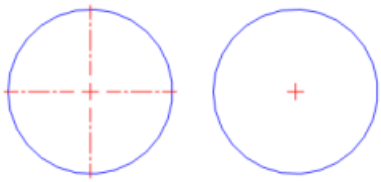
The CENTERRESET command resets centerline and center mark entities.

14.1.9.1 System variables for center entities

The properties and geometry of centerline and center mark entities are controlled through system variables.

Name	Title	Description
CENTERCROSSGAP	Center mark cross gap	Defines the gap between the center mark and its centerlines.  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.

Name	Title	Description
CENTERCROSSSIZE	Center mark cross size	Defines the size of the associative center mark. 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. Type a . (dot) to create center marks with zero length lines.
CENTEREXE	Centerline extensions length	Defines the length of the centerline extensions. The value is expressed in drawing units. 
CENTERLAYER	Default layer for center mark or centerline	Specifies the default layer for new centerlines and center marks. Type a name. If such layer not yet exists, the layer will be created when a centerline or centemark is created. Type a . (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 a new linetype name or press Enter to accept the current name. Type a . (dot) for continuous lines.
CENTERLTSCALE	Linetype scale for center mark or centerline	Specifies the linetype scale for newly created centerlines and center marks.

Name	Title	Description
CENTERMARKEXE	Automatic extension for center mark or centerline	Determines whether centerlines extend (left) or not (right) for new center marks. 

14.1.9.2 Editing center entities

The properties of center entities can be edited in the **Properties** panel.

14.1.10 Creating ordinate dimensions

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current coordinate system (WCS or 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 detects whether the point is x- or y-ordinate based depending on the 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.

14.1.10.1 To create an ordinate dimension

1 Set the origin of the coordinate system at the reference point for the ordinate dimensions.

2 Launch the DIMORDINATE command.

3 **You are prompted**: Select point for ordinate dimension:

4 Specify a point.

5 The ordinate dimension is dynamically displayed.

You are prompted: Leader endpoint [Xdatum/Ydatum/Text/Angle]:

6 Click to position the ordinate dimension.

14.1.11 Editing dimensions

You can edit dimension entities:

- Using grips.
- Edit the dimension text.
- Rotate the dimension text.
- Reposition the dimension text.
- Restore the text position.

- Make the extension lines oblique.

14.1.11.1 Using grips to edit a linear dimension

1 Select the dimension entity.

1 5 grips display:

2 origin point grips (1)

2 dimension line grips (2)

1 dimension text grip (3)



2 (option) Drag the origin point grips to adjust the dimension origin points.

3 The dimension text is adjusted automatically.

4 (option) Drag dimension line grips to move the dimension line.

5 (option) Drag the text grip to move the dimension text.

Nota:

- Depending on the **Text movement** setting (DIMTMOVE system variable), 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.

14.1.11.2 Editing the dimension text

Dimension text can be edited:

- in the MText editor:
  - using the DDEDIT command
  - double clicking the dimension entity
- in the **Text override** field in the **Properties** panel.
- using the **Edit Text** option of the DIMEDIT command.

Nota:

The following tags can be used to format the dimension text:

- **\L** (backslash upper case L) starts underlined text, **\l** (backslash lower case L) to stop underlined text
- **<>** to add the default dimension text (= measured distance)
- **[]** to add the alternate units, when alternate units are currently switched off in the dimension style.
- **%%d** to add the degree symbol (°)
- **%%c** to add the diameter symbol (Ø)

- **%%p** to add the plus/minus symbol (±)

e.g. if you type **%%c\L<> cm** in the **Text override** field, dimension text displays as:

Ø24,43cm

14.1.11.3 To rotate the dimension text

1 Do one of the following:

- Click the **Rotate Dimension Text** tool ( ) in the **Dimensions** toolbar or ribbon panel.
- Choose **Rotate Dimension Text** in the **Dimensions** menu.
- Launch the DIMEDIT command then choose the **Rotate text** option.

You are prompted: Angle of dimension text:

2 Type a value, then press Enter.

3 **You are prompted**: Select entities to rotate text [selection options (?)]:

4 Select the dimensions and press enter.

14.1.11.4 To reposition the dimension text

1 Do one of the following:

- Click the **Reposition Dimension Text** tool ( ) in the **Dimensions** toolbar or the ribbon panel.
- Choose **Reposition Dimension Text** in the **Dimensions** menu.
- Launch the DIMTEDIT command.

You are prompted: Select dimensions to reposition text:

2 Select the dimension.

3 The dimension text moves dynamically.

4 Click to reposition the dimension text.

14.1.11.5 To restore the text position

1 Do one of the following:

- Click the **Restore Text Position** tool ( )
- Choose **Restore Text Position** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Restore text** option.

You are prompted: Select dimensions to restore text to default position [selection options (?)]:

2 Select the dimensions, then press Enter.

14.1.11.6 To make the extension lines oblique

1 Do one of the following:

- Click the **Make Oblique** tool ( ) in the **Dimensions** toolbar or ribbon panel.
- Choose **Make Oblique** in the **Dimensions** menu.
- Launch the DIMEDIT command and choose the **Oblique lines** option.

You are prompted: Select linear dimensions to make oblique [selection options (?)]:

2 Select the dimension entities, then press Enter.

3 **You are prompted**: Enter obliquing angle:

4 Type a value, then press Enter.

Nota: To restore non-rotated extension lines, execute the above procedure and type 0 (zero) at the 'Enter obliquing angle' prompt.

14.1.12 Criar linhas de chamada e anotações

Uma linha de chamada é composta por uma ponta de seta, uma linha, polilinha ou spline, e uma anotação.

O procedimento geral para inserir uma anotação é:

- Especifique a posição da ponta da seta.
- Criar os segmentos de linha de chamada.
- Digitar o texto da anotação.

14.1.12.1 Para Criar uma chamada e uma anotação

1 Siga um dos procedimentos:

- Clique no botão da ferramenta **Chamada** () na barra **Dimensões** ou no painel de Faixa de opções.
- Escolha **Chamada** no menu **Dimensões**.
- Abra o comando LINHACHAM ou LINHACHAMCOTA.

Você é solicitado: Iniciar a chamada

2 Especifique a posição da ponta da seta.

3 **Você é solicitado**: Próximo ponto.

4 Especifique o ponto final do primeiro segmento da chamada.

5 **Você é solicitado**: Para apontar [Formatar/Desfazer/Anotação] <Anotação>:

6 Adicione mais segmentos à chamada, e pressione Enter ou clique o botão-direito para parar de adicionar segmentos.

7 **Você é solicitado**: Primeira linha do texto de anotação/ <opções>:

8 Digite a primeira linha do texto de anotação, e pressione Enter.

9 **Você é solicitado**: Próxima linha do texto da anotação:

10 Siga um dos procedimentos:

- Pressione Enter ou clique o botão-direito para criar a chamada e a anotação.
- Continue adicionando linhas para o texto da anotação, e pressione Enter ou clique o botão-direito para criar a chamada e a anotação.

14.1.12.2 Para criar apenas uma chamada

1 Siga um dos procedimentos:

- Clique no botão **Chamada** () na barra **Dimensões** ou no painel da Faixa de opções.

- Escolha **Chamada** no menu **Dimensões**.
- Abra o comando LINHACHAM ou LINHACHAMCOTA.

Você é solicitado: Iniciar a chamada

2 Especifique a posição da ponta da seta.

3 **Você é solicitado**: Próximo ponto.

4 Especifique o ponto final do primeiro segmento da chamada.

5 **Você é solicitado**: Para apontar [Formatar/Desfazer/Anotação] <Anotação>:

6 Adicione mais segmentos à chamada, e pressione Enter ou clique o botão-direito para parar de adicionar segmentos.

7 **Você é solicitado**: Primeira linha do texto de anotação/ <opções>:

8 Pressione Enter.

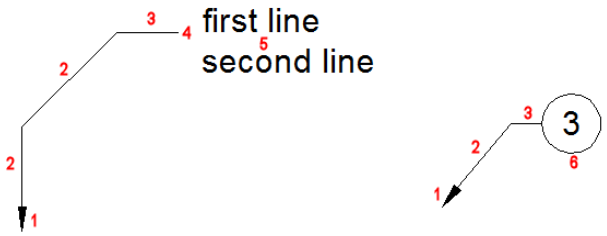
9 **Você é solicitado**: Opções do texto de dimensão [Bloco/Copiar/Nenhum/Tolerância/TextoM] <TextoM>:

10 Escolha a opção **Nenhum**.

14.1.13 Creating multileaders

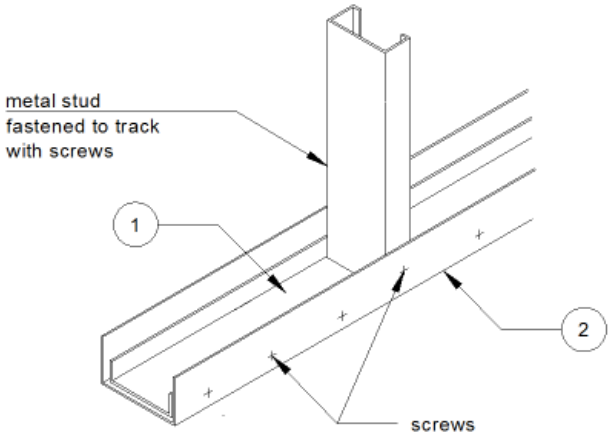
A multileader consists of:

- 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)



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.



14.1.13.1 To create a multileader

1 (option) Launch the MLEADERSTYLE command, then set the current multileader style.

2 Do one of the following:

- Click the **Multileader** tool button ( ) on the **Dimensions** toolbar or ribbon panel.
- Choose **Multileader** in the **Dimensions** menu.
- Launch the MLEADER command.

You are prompted: Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:

3 (option) Select a placement option:

- arrowhead first
- landing first
- content first.

The most recently used placement option is repeated.

4 The default sequence is: arrowhead - landing - content.

5 (option) Choose **Options** 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>:

Specify the option(s), then choose **Exit** to proceed.

6 Specify a point to define the leader arrowhead location.

7 **You are prompted**: Specify leader landing location:

8 Specify a point to define the leader landing location.

9 Depending on the content type:

- **MText**: The MText command is launched.
- **Block**: A block is inserted.
- **None**: The command is concluded.

14.1.13.2 Adding vertices to a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Vertex** tool ( ).

A new vertex is added dynamically.

6 Specify the position of the vertex.

14.1.13.3 Removing vertices from a leader line

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to add a vertex to, then hold down the Ctrl key.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu expand the **Modify** tab.

5 Click the **Remove Vertex** tool ( ).

The vertex is removed.

14.1.13.4 Adding leader lines to a multileader

1 If necessary press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the multileader entity.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Add Leader** tool ( ).

A new leader line is dynamically displayed.

6 Specify the origin of the new leader line*.

7 Another new leader line is dynamically displayed.

8 Do one of the following:

- Specify the origin of the new leader line.
- Right click or press Enter to stop adding leader lines.

(*) When the Maximum leader points property of the multileader style is higher than 2, right click to stop adding vertices for the new leader.

14.1.13.5 Removing leader lines from a multileader

1 If necessary, press the **F12** function key to activate the **Quad** cursor menu.

2 Pause the cursor over the leader line you want to remove.

3 The **Quad** cursor menu is displayed.

4 Move the cursor to the **Quad** cursor menu and expand the **Modify** tab.

5 Click the **Remove Leader** tool ( ).

The leader line is removed.

14.1.14 Adicionar tolerancias geométricas

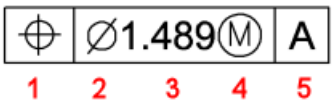
Tolerâncias geométricas especificam a variação máxima permitida na forma ou na posição da geometria verdadeira. A tolerância geométrica é, em essência, a zona de tolerância dentro da qual uma superfície ou eixo de furo ou cilindro pode estar, o que resulta no recurso resultante aceitável para a função adequada e intercambiabilidade.

BricsCAD desenha as tolerâncias geométricas utilizando um quadro de controle do recurso, que é um retângulo dividido em compartimentos.

Cada quadro de controle do recurso consiste em pelo menos dois compartimentos. O primeiro compartimento contém um símbolo de tolerância geométrica que indica a característica geométrica à qual a tolerância é aplicada, como localização, orientação, ou forma. Por exemplo, uma tolerância à forma pode indicar a planariedade ou arredondamento de uma superfície.

O segundo compartimento contém o valor da tolerância. Quando apropriado, o valor de tolerância é precedido por um símbolo de diâmetro e seguido por um símbolo da condição do material. As condições do material se aplicam a recursos que podem variar em tamanho.

O valor da tolerância pode então ser seguido por letras de referência de dados primárias, secundárias e terciárias, juntamente com as condições do material de cada dado. As letras de referência de dados são geralmente usadas como tolerâncias de referência para um de até três planos perpendiculares a partir dos quais uma medição é feita, embora as letras de referência de dados também possam indicar um ponto ou eixo exato.



1 Símbolo característico da geometria

2 Símbolo de diâmetro

3 Valor da tolerância

4 Símbolo da condição do material

5 Referência de dados

Quando duas tolerâncias se aplicam à mesma geometria, você também pode adicionar uma tolerância composta que consiste em um valor de tolerância primário seguido por um valor de tolerância secundário. Para tornar uma tolerância ainda mais específica, esta também pode conter uma tolerância projetada que consiste em um valor de altura seguido por um símbolo de tolerância projetado. Por exemplo, você pode usar uma tolerância projetada para indicar a perpendicularidade de uma peça incorporada.

14.1.14.1 Criar um quadro de tolerância geométrica

1 Siga um dos procedimentos:

- Clique no botão **Tolerância** ( ) na barra **Dimensões** ou no painel da Faixa de opções.

- Clique em **Tolerância** no menu **Dimensões**.
- Digite **Tolerancia** na linha de Comando, e pressione Enter.

Abre-se a caixa de diálogo Tolerância Geométrica.

2 Componha o quadro de tolerância geométrica na caixa de diálogo **Tolerância Geométrica**.

3 Clique o botão **OK**.

A caixa de diálogo **Tolerância Geométrica** é fechada.

4 Especifique um ponto para inserir o quadro de tolerância geométrica no desenho.

14.1.14.2 Editar um quadro de tolerância geométrica

1 Clique duas vezes no quadro de tolerância geométrica.

1 A caixa de diálogo **Tolerância Geométrica** é aberta.

2 Editor de config. do plotter:

3 Clique o botão **OK**.

A caixa de diálogo **Tolerância Geométrica** é fechada.

O conteúdo do quadro de tolerância geométrica é atualizado.

14.2 Texts

14.2.1 Text system variables

System variables can be set using the **Settings** dialog box.

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 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.

Variable Name	Variable Title	Description
TEXTANGLE	Text angle	Stores the angle of the last added text entity.
TEXTFILL	Text fill	Determines whether True Type fonts show as either filled or outlined. 
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.

14.2.2 Trabalhar com estilos de texto

Quando você adicionar texto a um desenho, este usa o estilo de texto atual, o que determina a fonte, tamanho, ângulo, orientação e outras propriedades da nova entidade texto.

Cada desenho tem um estilo de texto padrão, chamado **Padrão**, que inicialmente usa a fonte **Arial**. Você não pode excluir ou renomear o estilo **Padrão**, mas pode modificá-lo.

Se você alterar a **fonte** ou a **orientação** de um estilo existente, todas as entidades de texto existentes criadas usando esse estilo serão atualizadas automaticamente para refletir a alteração. Alterar qualquer outra propriedade não tem efeito sobre o texto existente.

Você pode criar e usar um número ilimitado de estilos adicionais de texto.

Estilos de texto anotativos criam entidades de texto anotativas. Em um estilo de texto anotativo, a **Altura** define a altura do texto no Paper Space; a altura do texto no Model Space depende da escala de anotação atual, conforme definido pelas variáveis do sistema CANNOSCALE e CANNOSCALEVALUE.

14.2.2.1 Para abrir a caixa de diálogo Explorer Estilo de Texto

Siga um dos procedimentos:

- No menu **Ferramentas** > **Explorer do Desenho**, escolha **Estilos de Texto...**
- Clique no campo **Estilo** na barra **Status**.
- Inicie o comando ESTILO.

14.2.2.2 Para criar um estilo de texto

1 Abra a caixa de diálogo **Explorer Estilo de Texto**.

2 Siga um dos procedimentos:

- Clique no botão **Nova** () na seção **Detalhes** da caixa de diálogo **Explorer do Desenho**.
- Selecione um estilo existente, então clique o botão-direito e escolha **Novo** no menu de contexto.

Um estilo de texto **NovoEstilo** é criado, herdando as propriedades do estilo atualmente selecionado.

3 Digite um nome no campo **Nome do Estilo de Texto**, substituindo o nome padrão **NovoEstilo**.

4 Defina as propriedades do estilo de texto.

5 Feche a caixa de diálogo **Explorer Estilo de Texto**.

14.2.2.3 Para editar um estilo de texto

1 Abra a caixa de diálogo **Explorer Estilo de Texto**.

2 Selecione o estilo de texto.

3 Defina as propriedades do estilo de texto.

4 Feche a caixa de diálogo **Explorer Estilo de Texto**.

14.2.2.4 Para definir um estilo de texto atual

Siga um dos procedimentos:

- Clique o botão-direito no campo **Estilo** na barra **Status**, e clique no estilo de texto.
- Abra a caixa de diálogo **Explorer Estilo de Texto**.

  O estilo de texto atual é indicado na coluna **Atual**.

  Siga um dos procedimentos:

  - Clique no campo **Atual** do estilo de texto.
  - Clique o botão-direito no estilo de texto, e escolha **Definir Atual** no menu de contexto.

14.2.2.5 Para excluir um estilo de texto

1 Abra a caixa de diálogo **Explorer Estilo de Texto**.

2 Selecione o estilo de texto.

3 Siga um dos procedimentos:

- Clique o botão **Excluir** (✕).
- Selecione o estilo de texto, então escolha **Excluir** no menu de contexto.

Se o estilo de texto não é usado no desenho ou em um estilo de dimensão, o estilo de texto é excluído, ou de outra forma exibe uma caixa de alerta.

4 Siga um dos procedimentos:

- Clique no botão **Excluir** para excluir o estilo de texto e todas as entidades que usam este estilo.
- Clique no botão **Alterar...** para excluir o estilo de texto, e atribuir um estilo de texto diferente para as entidades que atualmente usam o estilo selecionado.
- Clique no botão **Cancelar** para cancelar o procedimento de exclusão.

Nota: Não é possível excluir o estilo do texto atual.

14.2.3 Criar texto

O comando TEXTO coloca linhas simples de texto no desenho; opcionalmente, avalia expressões LISP.

O comando TEXTOM insere textos formatados, textos de parágrafo, em uma caixa delimitadora que limita a extensão do texto.

14.2.3.1 Para criar texto de linha única

1 Siga um dos procedimentos:

- Clique no botão **Texto** (A) na barra **Desenhar** ou no painel **Anotar / Texto** da Faixa de opções.
- Escolha **Texto** no menu **Desenhar**.
- Inicie o comando TEXTO.

2 **Você é solicitado**: Ponto inicial do texto ou [Usar estilo definido/Alinhar na linha/Ajustar entre pontos/ Centralizar horizontalmente/Médio (hor/vert)/Justificar à direita/Opções de justificação...]:

3 Especifique um ponto.

4 **Você é solicitado**: Altura do texto <altura atual>:

5 Siga um dos procedimentos:

- Pressione Enter ou clique o botão-direito para aceitar o texto padrão.
- Digite um valor.
- Clique em um ponto para definir graficamente a altura.

Você é solicitado: Angulo de rotação do texto <ângulo atual>

6 Siga um dos procedimentos:

- Pressione Enter ou clique o botão-direito para aceitar o atual ângulo de rotação.
- Digite um valor.
- Clique em um ponto para definir graficamente o ângulo de rotação.

7 Digite o texto, e pressione Enter.

8 Um novo texto vazio é criado.

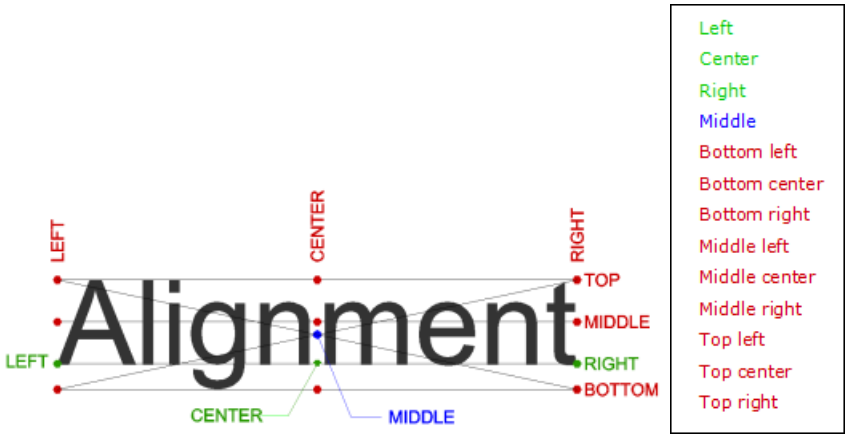
9 Siga um dos procedimentos:

- Digite o texto.
- Pressione Enter para parar.

10 (opção) Continue adicionando textos e pressione Enter duas vezes para parar.

Nota: Quando você já tiver criado o texto na sessão atual do BricsCAD, a entidade de texto mais recente é realçada quando você repete o comando TEXTO. Pressione Enter ou clique o botão-direito para criar um novo texto abaixo do texto mais recente, usando a mesma altura de texto e ângulo de rotação.

Opções de justificação de texto em linha única:



14.2.3.2 Para criar texto de múltiplas linhas

1 Siga um dos procedimentos:

- Clique o botão **Texto Multi-linhas** (A) na barra **Desenhar** ou no painel **Anotar /Texto** da Faixa de opções.
- Escolha **Texto Multi-linhas** no menu **Desenhar**.
- Inicie o comando TEXTOM.

Você é solicitado: Primeiro canto da caixa de texto:

2 Especifique o ponto de inserção da entidade de texto de múltiplas linhas.

3 **Você é solicitado**: Justificação/Rotação/Estilo/Altura/Direção/Largura/ <Canto oposto para bloco de texto>:

4 Siga um dos procedimentos:

- Especificar o canto oposto da caixa de texto para definir a largura máxima do texto de múltiplas linhas.
- Se **Dimensões Dinâmicas** estiver ativa, a altura e a largura do campo de texto são exibidas dinamicamente. Digite a altura e a largura nos campos de dimensão dinâmica. Pressione a tecla TAB para alternar; pressione Enter para confirmar.
- Digite @ na barra de Comando, e pressione Enter para texto não fracionado (largura zero) (= largura de linha ilimitada).

A barra **Formatação de texto** é aberta logo acima do bloco de texto.

5 Digite seu texto.

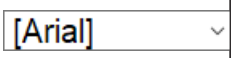
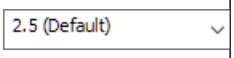
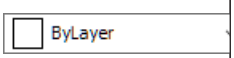
6 A largura máxima da linha é definida pela largura do bloco de texto, conforme definido na etapa 3.

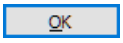
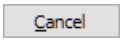
Pressione Enter para iniciar um novo parágrafo.

7 Para concluir o comando e fechar o diálogo **Formatação de texto** faça um dos seguintes:

- Clique o botão **OK**
- Pressione Ctrl+Enter.
- Clique fora da entidade TextoM.

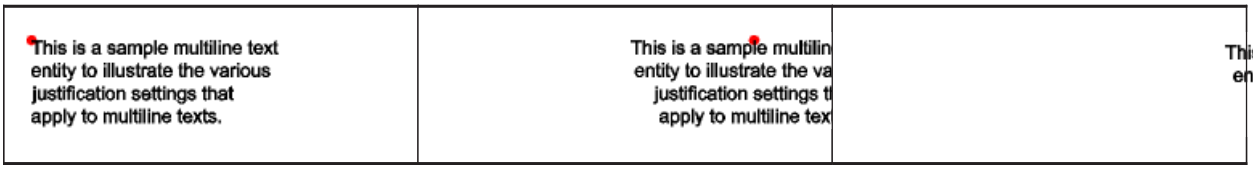
Ferramentas para formatação de texto e configurações

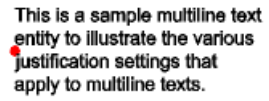
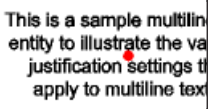
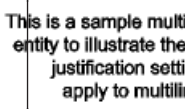
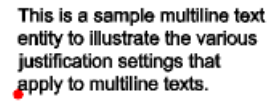
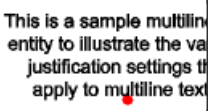
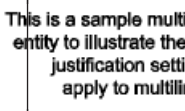
Botão ou configuração	Nome	Descrição
	Estilo de texto	Define o estilo de texto
	Fonte do Texto	Define a fonte do texto
	Altura do Texto	Define a altura do texto
	Negrito	Cria texto em negrito
	Itálico	Cria texto em itálico
	Sublinhado	Cria texto sublinhado
	Overscore	Cria texto com barra na parte superior
	Maiúsculas	Converte o texto selecionado em maiúsculas
	Minúsculas	Converte o texto selecionado em minúsculas
	Anotativa	Alterna a propriedade Anotativa
	Espaçam Linhas	Define o espaçamento de linhas
	Colunas	Organiza o texto em colunas
	Cor	Define a cor

Botão ou configuração	Nome	Descrição
	Angulo Oblíquo	Define o ângulo oblíquo
	Fator de Largura	Define o fator de largura
	Espaçamento entre Caracteres	Define o espaçamento entre caracteres
	Desfazer	Desfaz as ações recentes
	Refazer	Refaz as ações anteriormente desfeitas
	Fração Empilhada	Cria frações empilhadas
	Caractere Especial	Insere caracteres especiais
	Campo	Insere um campo variável
	Justificar	Define a justificação do texto.
	Máscara de Fundo	Adiciona um fundo colorido
	Régua	Alterna a exibição da régua
	OK	Fecha a barra **Formatação de texto** e aplica as alterações
	Cancelar	Fecha a barra **Formatação de texto** e deixa a entidade TextoM inalterada

Justificação de texto de múltiplas linhas

O ponto vermelho indica o ponto de inserção da entidade texto de Multi-linhas.



Superior esquerdo	Superior centro	Top right
		
Meio esquerdo	Médio centro	Médio direito
		
Esquerda inferior	Centro inferior	Bottom right

14.2.3.3 Incluir símbolos e caracteres especiais de texto

Você pode usar códigos de controle e sequências de caracteres unicode para incluir caracteres especiais em entidades de texto.

Para incluir códigos de controle, ao digitar texto, digite dois símbolos de porcentagem (%%) seguidos pelo código ou caractere de controle especial. Um único sinal de porcentagem é tratado como um caractere normal de texto.

Caracteres especiais de texto

Código de Controle	Sequência Unicode	Resultado
%%d	\U+00B0	Desenha um símbolo de grau (°)
%%p	\U+00B1	Desenha um símbolo de mais/menos (±)
%%c	\U+2205	Desenha um símbolo de diâmetro (Ø)
%%k	\U+0336	Começa a riscar o texto. Repita o código para parar.
%%u	\U+005F	Começa a sublinhar o texto. Repita o código para parar.
%%o	\U+203E	Começa a superescritar o texto. Repita o código para parar.

Símbolos e sequências Unicode

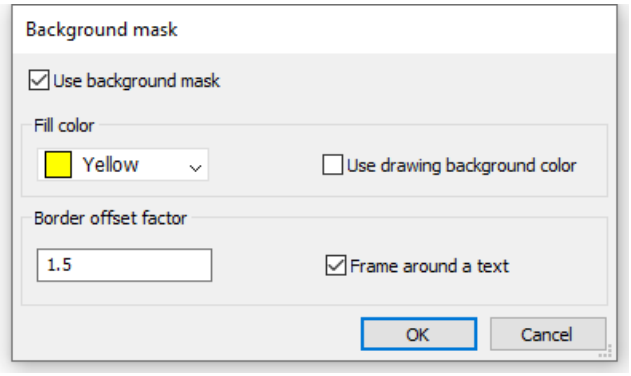
Nome	Símbolo (*)	Sequência Unicode
Quase igual	≈	\U+2248
Angulo	∠	\U+2220
Linha de centro	℄	\U+2104
Delta	Δ	\U+0394
Fase elétrica	ϕ	\U+0278
Identidade	≡	\U+2261
Desigual	≠	\U+2260
Omega	Ω	\U+03A9
Quadrática	²	\U+00B2
Cubo	³	\U+00B3

(*) Se for suportado pela fonte do texto.

14.2.3.4 Para adicionar uma máscara de fundo

1 Ao criar ou editar uma entidade de texto multi-linhas, clique no botão **Máscara de Fundo** ( ) na barra **Formatação de Texto**.

A caixa de diálogo **Máscara de Fundo** é exibida:



2 Marque a opção **Use a máscara de fundo**.

3 Siga um dos procedimentos:

- Escolher uma cor na lista **Cor de preenchimento**.

- Escolha **Selecionar cor...** na lista suspensa **Cor de preenchimento** e escolha uma cor na caixa de diálogo **Selecionar cor**.
- Marque a opção **Use a cor de fundo do desenho**.

4 Defina o **Fator de deslocamento da borda**, que define a margem ao redor do texto para o plano de fundo. O valor é baseado na altura do texto e deve estar no intervalo 1-5. Um fator de 1.0 ajusta exatamente à entidade de texto de múltiplas linhas.

5 (opção) Marque a opção **Quadro ao redor de um texto** para criar um quadro de texto.

6 Clique em **OK** para criar a máscara de fundo.

14.2.4 Editar texto

Você pode editar e modificar uma entidade de texto como faria com qualquer outra entidade do desenho. Ou seja, você pode excluir, mover, rotacionar, e escalar texto.

Você pode editar as propriedades do texto no painel **Propriedades**.

A variável de sistema TEXTED controla o comportamento de edição de texto para entidades Texto:

- 0 = editor no-local
- 1 = diálogo em pop-up
- 2 = editor no-local com entrada repetida

A variável de sistema MTEXTED permite usar um editor de texto externo para editar entidades TextoM.

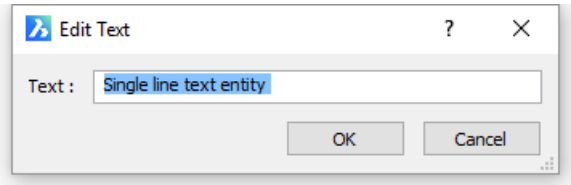
A variável de sistema MTEXTFIXED controla se a vista é em Zoom, se é rotacionada e/ou deslocada em Pan, para se ajustar ao TextoM editado.

14.2.4.1 Para editar uma entidade Texto

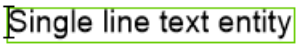
1 Clique duas vezes na entidade de texto.

2 Dependendo do valor da variável de sistema TEXTED:

3 A caixa de diálogo **Editar Texto** é exibida.



ou o texto é editado 'no-local'.



4 Edite o texto, e pressione Enter.

5 Siga um dos procedimentos:

- Clique em outra entidade de texto para editar.
- Pressione Enter para parar de editar entidades de texto.

14.2.4.2 Para editar uma entidade de TextoM

1 Clique duas vezes na entidade TextoM.

2 A barra de ferramentas **Formatação de Texto** é exibida.

3 Edite o texto e clique no botão **OK** ou clique fora do bloco de texto.

4 Siga um dos procedimentos:

- Clique em outra entidade TextoM para editar
- Pressione Enter para parar de editar entidades TextoM.

14.2.4.3 Para editar uma seleção de entidades TextoM

1 Selecione as entidades TextoM.

2 No painel **Propriedades**, selecione **Conteúdo**.

3 Pressione o botão **Procura** (...) no lado direito do campo **Conteúdo**.

4 O diálogo **Formatação de Texto** abre para a primeira entidade TextoM na seleção.

5 Edite o TextoM

6 Pressione o botão **OK** ou clique fora do bloco de texto.

A próxima entidade TextoM na seleção pode ser editada agora.

7 Repita as etapas 5 e 6 até que o último TextoM esteja editado.

14.2.4.4 Para um editor de texto externo para TextoM's

1 Inicie o comando AJUSTES para abrir a caixa de diálogo **Configurações**.

2 Sob **Desenho** > **Desenho** > **Criação da** > **Entidade** > **Textos** > **Texto multilinha**, selecione a variável do sistema MTEXTED.

3 Digite o nome do arquivo e o caminho completo do editor de texto externo, na barra de Comando e pressione Enter.

4 ex., C:\Program Files\Windows NT\Accessories\Wordpad.EXE

Nota: Para restaurar o editor de texto integrado, exclua o conteúdo da variável de sistema MTEXTED.

14.2.5 Exploding text

The TXTEXP command explodes Text and Mtext entities into 2D polylines.

14.2.5.1 To explode text

1 Select the text(s)

2 Do one of the following:

- Choose **Explode** in the **Edit** tab of the **Quad**.
- Launch the TXTEXP command.

Nota:

- The solid fill of certain TTF fonts is removed when exploding such texts, leaving the text outlines only.
- Use the JOIN command to create closed polylines from the exploded text outlines.

- The **Explode** tool in the **Edit** tab of the **Quad** launches the TXTEXP command. The EXPLODE command cannot explode text entities.

14.2.6 Localizar e substituir texto

O comando LOCALIZAR permite especificar o texto que deseja encontrar. Então você pode encontrar ou substituir o texto. Opcionalmente, você pode ampliar em zoom as várias instâncias das entidades de texto que contêm o texto de pesquisa.

14.2.6.1 Para definir as opções Localizar e Substituir

1 Lance o comando LOCALIZAR.

1 A caixa de diálogo **Localizar e Substituir** é exibida.

2 Clique o botão **Opções**.

3 Marque as opções na caixa de diálogo **Opções de Localizar e Substituir**.

4 Clique o botão **OK**.

14.2.6.2 Para localizar texto

1 Lance o comando LOCALIZAR.

1 A caixa de diálogo **Localizar e Substituir** é exibida.

2 (opção) Definir as opções de **Localizar e Substituir**.

3 (opção) Clique no botão **Selecionar Entidades** () para compor um conjunto de seleção.

Se nenhuma entidade for selecionada, todo o desenho será pesquisado.

- O diálogo **Localizar e Substituir** é fechado temporariamente para permitir a você selecionar entidades.
- Clique o botão-direito do mouse para concluir a seleção de entidades.

O campo **Pesquisar em** lê **Seleção Atual** em vez do **Desenho Inteiro**.

4 Digite o texto de pesquisa no campo **Localizar Sequência de Texto**.

5 Clique no botão **Localizar**.

6 O conteúdo da primeira entidade de texto que contém a string de pesquisa é exibido no painel **Resultados da pesquisa**.

7 (opção) Clique o botão **Zoom Para**.

O desenho é ampliado em zoom na entidade de texto.

8 Siga um dos procedimentos:

9 Repita as etapas 5 e 6 para localizar mais casos da mesma sequência pesquisada.

10 Repita as etapas 2 a 6 para localizar outra sequência de texto.

11 Pressione a tecla Esc ou clique no botão **Fechar** para parar.

14.2.6.3 Para substituir texto

1 Lance o comando LOCALIZAR.

1 A caixa de diálogo **Localizar e Substituir** é exibida.

2 (opção) Definir as opções de **Localizar e Substituir**.

3 (opção) Clique no botão **Selecionar Entidades** ( ) para compor um conjunto de seleção.

Se nenhuma entidade for selecionada, todo o desenho será pesquisado.

- O diálogo **Localizar e Substituir** é fechado temporariamente para permitir a você selecionar entidades.
- Clique o botão-direito do mouse para concluir a seleção de entidades.

O campo **Pesquisar em** lê **Seleção Atual** em vez do **Desenho Inteiro**.

4 Digite o texto de pesquisa no campo **Localizar Sequência de Texto**.

5 Digite o texto de substituição no campo **Substituir com**, e pressione Enter.

Todas as entidades de texto que contêm a string de pesquisa são listadas.

6 Siga um dos procedimentos:

- Clique em um item na lista para destacar a entidade de texto no desenho e, em seguida, clique no botão **Substituir**.
- Clique no botão **Substituir Tudo**.

14.2.7 Verificar a ortografia

O verificador ortográfico confere a ortografia do texto no desenho atual. Você pode verificar a ortografia de uma ou mais entidades de texto selecionadas, ou o desenho inteiro.

Durante uma verificação ortográfica, o comando ORTOGRAF compara as palavras no desenho ou no atual conjunto de seleção, com as palavras no atual dicionário principal e no atual dicionário personalizado. Se uma palavra não for encontrada em nenhum dos dicionários, você poderá selecionar a ortografia correta no campo de sugestões ou adicionar a palavra ao dicionário personalizado atual. Dicionários personalizados podem ser usados para palavras específicas da disciplina.

14.2.7.1 Configurar dicionário

Os dicionários a ser usados pelo comando ORTOGRAF estão definidos pelas variáveis de sistema DCTMAIN (dicionário principal), e DCTCUST (dicionário personalizado).

Nota: Nós só entregamos o dicionário de Inglês por causa do tipo de licença nos arquivos do dicionário: a versão em Inglês pode ser distribuída livremente, enquanto todos os outros idiomas não são gratuitos. No entanto, você pode baixar outros idiomas e usá-los gratuitamente (consulte Para adicionar um dicionário principal abaixo).

14.2.7.2 Para abrir a caixa de diálogo Verificação ortográfica

Siga um dos procedimentos:

- Escolha **Verificar Ortografia** no menu **Ferramentas**.
- Clique na ferramenta **Verificar Ortografia** no painel **Anotar / Texto** da Faixa de opções.

- Inicie o comando ORTOGRAF.

14.2.7.3 Para verificar a ortografia

1 Abra a caixa de diálogo **Verificação Ortográfica**.

2 (opção) Clique no botão **Selecionar** ( ) para compor um conjunto de seleção.

Se nenhuma entidade for selecionada, todo o desenho será pesquisado.

- A caixa de diálogo **Verificação Ortográfica** fecha temporariamente para permitir que você selecione entidades.
- Clique o botão-direito do mouse para concluir a seleção de entidades.

O campo **Onde verificar** lê **Seleção Atual** em vez de **Desenho Inteiro**.

3 (opção) Altere os dicionários atuais (veja abaixo).

4 Clique no botão **Iniciar**.

A primeira palavra possivelmente com erros ortográficos é exibida na seção **Palavra atual**.

5 O conteúdo da entidade de texto na qual a **Palavra atual** é encontrada é mostrado no campo **Contexto**.

6 Siga um dos procedimentos:

- Clique no botão **Ignorar** para deixar inalterada esta instância da **Palavra atual**.
- Clique no botão **Ignorar Tudo** para deixar inalteradas todas as instâncias da **Palavra atual**.
- Aceite a palavra no campo **Sugestões** ou selecione uma palavra na lista de **Sugestões**, e clique o botão **Alterar** para substituir a cópia da **Palavra atual** pela palavra presente no campo **Sugestões**.
- Aceite a palavra no campo **Sugestões** ou selecione uma palavra na lista de **Sugestões**, e clique no botão **Alterar Tudo** para substituir todas as cópias da **Palavra atual** pela palavra no campo **Sugestões**.
- Clique no botão **Adicionar** para adicionar a **Palavra atual** ao atual dicionário personalizado.
- Clique no botão **Pesquisar** para listar palavras semelhantes à palavra no campo **Sugestões**.

7 Repita a etapa anterior até que a mensagem **Verificação ortográfica está concluída** seja exibida.

14.2.7.4 Para alterar dicionários

1 Abra a caixa de diálogo **Verificação Ortográfica**.

2 Clique no botão **Alterar Dicionários...**.

A caixa de diálogo **Alterar dicionários** é exibida.

3 Clique o botão de lista **Dicionário principal** para alterar o dicionário principal.

4 Selecione um novo dicionário principal.

5 (opção) Clique no botão **Procurar...** para selecionar um novo **Dicionário personalizado**.

6 (opção) Digite uma palavra no campo **Palavras no dicionário personalizado** e clique no botão **Adicionar** para adicionar essa palavra ao seu dicionário personalizado.

7 Clique no botão **Aplicar & Fechar**.

Notas:

- Dicionários personalizados são mais bem salvos na pasta **Support** da Pasta-raiz **Roamable** (transportável).
- Se você quiser compartilhar o dicionário personalizado com o MS Word, aponte para o dicionário personalizado do MS Word na etapa 5 do procedimento acima. Por padrão, CUSTOM.DIC é o dicionário personalizado no MS Word, que pode ser encontrado sob *C:\Users\<user name>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

14.2.7.5 Para adicionar um novo dicionário principal

1 Abra a caixa de diálogo **Verificação Ortográfica**.

2 Clique no botão **Alterar Dicionários...**.

3 Clique no botão **Download...**.

Seu navegador abre: https://extensions.openoffice.org/dictionaries

4 Percorra a lista para o idioma de sua escolha e clique no link para abrir a página do idioma selecionado.

5 Clique no botão **Download extension** para iniciar o download do arquivo ***.oxt**.

6 Altere a extensão **oxt** para **zip**.

7 Extraia arquivos ***.aff** e ***.dic** do arquivo ZIP para a pasta **C:\Program Files\Bricsys\BricsCAD Vxx pt_BR\Support**.

O dicionário está agora disponível na lista **Dicionário principal** na caixa de diálogo **Alterar Dicionários**.

14.2.7.6 Para adicionar um novo dicionário personalizado

1 Abra um editor de texto como o **Bloco de Notas**.

2 (opção) Digite as palavras que você deseja usar em seu dicionário personalizado.

3 Cada palavra deve estar em uma linha separada.

4 Salve o arquivo na pasta **Support** da pasta raiz **Roamable** usando a extensão ***.cus** .

14.2.8 Working with fields

A field is a text entity which contains instructions to display the status or property of another entity or setting in the drawing. 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 command to learn more about default and custom drawing properties.

See the SHEETSET 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 system variable is ON, fields are displayed on a light gray background which does not print.

The FIELDEVAL system variable controls when fields are updated.

14.2.8.1 General procedure to create fields

1 Do one of the following:

- Click the **Insert Field** tool button ( ) on the **Insert** toolbar and the **Annotate / Text** ribbon panel.
- Choose **Insert Field...** in the **Insert** menu.
- Launch the FIELD command.
- Click the **Field** tool button ( ) on the **Text Formatting** toolbar when creating or editing Mtext or a table cell content.

The **Field** dialog box appears.

2 Under **Field Names**, expand a group, then choose a field name.

The settings for the selected field name appear.

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]:

Do the following:

- (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.
- Specify the insertion point of the field in the drawing.

An Mtext entity containing the field is created.

14.2.8.2 To display an entity property in a field

1 Launch the FIELD command.

1 The **Field** dialog box appears.

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.

5 The **Field** dialog box appears again.

6 Select the entity property in the **Property** list.

7 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.

8 Click the **OK** button.

9 Specify a point to place the field in the drawing.

14.2.8.3 To edit a field

1 Edit the table cell or Mtext entity that contains the field.

2 Double click the field.

3 The **Field** dialog box appears.

4 Adjust the properties of the field.

5 Click the **OK** button to apply the changes.

14.2.8.4 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.
   - Launch the UPDATEDFIELD command.

   You are prompted: Select field(s) to update:

2 Select the field(s) you want to update, then right click or press Enter.

3 The selected fields are updated.

Nota: If bitcode 16 of the FIELDEVAL system variable is set, executing the REGEN command updates all fields in the drawing.

14.2.8.5 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.

14.2.9 Trabalhar com tabelas

Uma tabela é uma entidade composta por linhas e colunas, parecida com uma planilha do Microsoft Excel. Uma entidade Tabela pode ser criada como uma tabela vazia ou você pode importar dados a partir de um arquivo *.CSV (Comma Separated Values=Valores Separados por Vírgula). A aparência do texto

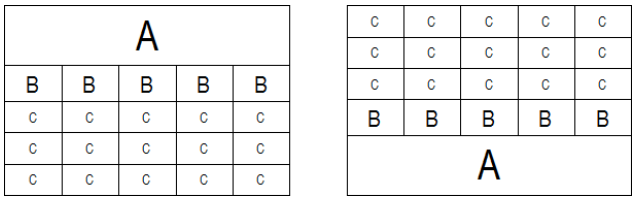
da tabela e das linhas de grade é controlada por um estilo de tabela. Os estilos de tabela são salvos no arquivo *.DWG.

Uma tabela padrão consiste em:

- uma célula **Título** (A)
- uma linha de células de **Cabeçalho** (B)
- uma ou mais linhas de células de **Dados** (C).

Uma tabela pode ter uma das duas direções:

- **Abaixo** (linha de título na parte superior)
- **Acima** (linha de título na parte inferior).



14.2.9.1 Para criar uma tabela vazia

1 Siga um dos procedimentos:

- Clique no botão **Tabela** ( ) na barra de ferramentas **Desenhar** ou no painel **Home / Anotações** da Faixa de opções.
- Escolher **Inserir Tabela...** no menu **Desenhar**.
- Inicie o comando TABELA.

A caixa de diálogo **Inserir Tabela** é exibida.

2 Escolha um Estilo de Tabela na lista **Estilo de Tabela**.

3 (opção) Defina o estilo de célula para a primeira, segunda e outras linhas.

4 Por padrão, o estilo de célula para a primeira linha é **Título**, o estilo de célula para a segunda linha é **Cabeçalho** e o estilo de célula para as outras linhas é **Dados**.

5 Em **Opções de Tabela**, selecione **Iniciar da Tabela Vazia**.

6 Em **Comportamento de Inserção**, siga um destes procedimentos:

- Escolha **Especificar Ponto de Inserção**, e especifique o número de colunas e a largura da coluna, e o número de linhas de dados e a altura da linha.
- Escolha **Especificar Janela**, e então faça o seguinte:
  - Especifique número de linhas de dados e de colunas.
  - Especifique a largura da coluna e altura da linha.
  - Especifique a largura da coluna e número de linhas de dados.
  - Especifique o número de colunas e a altura da linha.

7 Clique o botão **OK**.

Dependendo da sua escolha na etapa 5, **você será solicitado** a especificar um ponto de inserção ou

uma janela.

A tabela é criada e a barra **Formatação de Texto** é exibida (consulte Criar Textos).

8 Siga um dos procedimentos:
- Comece a adicionar texto na tabela.
- Clique fora da tabela para fechar a barra de ferramentas **Formatação de Texto**.

14.2.9.2 Para criar uma tabela a partir de dados

Quando importar dados de um arquivo *.CSV (arquivo de Valores Separados por Vírgula), por favor certifique-se que o caractere separador de lista em seu sistema é o mesmo usado no arquivo *.CSV (veja Definir o caractere separador de lista).

1 Siga um dos procedimentos:
- Clique no botão **Tabela** () na barra **Desenhar** ou no painel **Home / Anotações** na Faixa de opções.
- Escolher **Inserir Tabela...** no menu **Desenhar**.
- Inicie o comando TABELA.

A caixa de diálogo **Inserir Tabela** é exibida.

2 Em **Opções de Tabela**, selecione **De Dados**.

3 Clique no botão **Procurar** ().

4 A caixa de diálogo **Abrir** aparece.

Escolha um arquivo CSV e clique no botão **Abrir**.

5 Clique no botão **OK** na caixa de diálogo **Inserir Tabela**.

6 Clique em um ponto no desenho para colocar a tabela.

7 A tabela é criada e a barra **Formatação de Texto** é exibida (consulte Criar Textos).

8 Siga um dos procedimentos:
- Comece a editar o texto na tabela.
- Clique fora da tabela para fechar a barra de ferramentas **Formatação de Texto**.

9 (opção) Selecione a tabela e arraste as alças para ajustar as dimensões da tabela.

14.2.9.3 Para criar uma tabela a partir de uma planilha do Excel

1 Abra a planilha no **Microsoft Office Excel**.

2 Selecione as células a partir das quais deseja criar uma tabela.

3 Siga um destes procedimentos para copiar a seleção para a Area de transferência:
- Pressione **Ctrl + C**.
- Clique o botão direito do mouse e escolha **Copiar** no menu de contexto.

4 No BricsCAD, escolha **Colar especial...** no menu **Editar** ou inicie o comando COLARESPEC.

5 A caixa de diálogo **Colar Especial** é exibida.

6 Escolha **Planilha XML** na lista de opções **Como**.

7 Clique no desenho para inserir a tabela.

14.2.9.4 Editar uma tabela utilizando alças

Quando uma tabela é selecionada, múltiplas alças são mostradas:

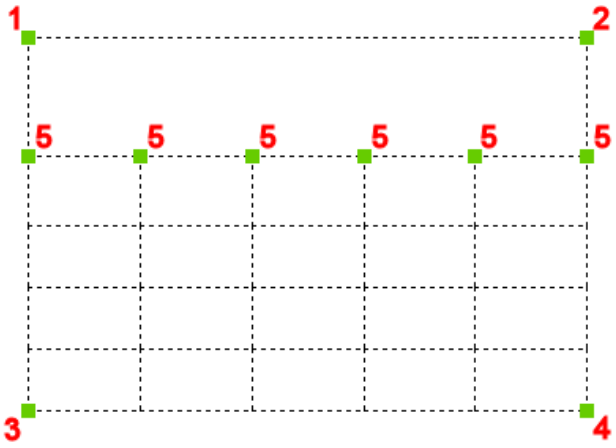
1 Move a tabela.

2 Modifica a largura uniformemente.

3 Modifica a altura uniformemente.

4 Modifica a largura e altura uniformemente.

5 Modifica a largura de uma coluna.



14.2.9.5 Editar uma tabela usando a barra de ferramentas de Tabela

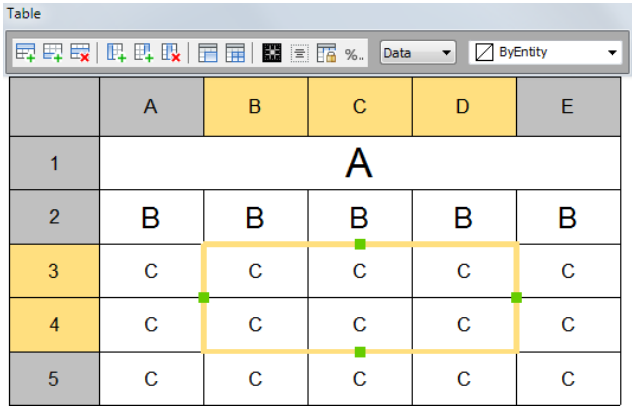
1 Para abrir a barra **Tabela**, faça o seguinte:

- Clique na célula que deseja editar.
- Pressione e mantenha pressionado o botão esquerdo do mouse, e defina um retângulo para selecionar várias células

2 As letras das colunas e os números das linhas são exibidos.

3 As colunas e as linhas da seleção são destacadas.

Um retângulo com quatro alças é desenhado ao redor da seleção.



4 (opção) Arraste as alças da seleção para modificar as dimensões.

5 (opção) Clique em uma ferramenta na barra **Tabela**.

6 (opção) Clique o botão-direito para abrir um menu de contexto:
- **Excluir Conteúdo da Célula**: exclui o texto nas células selecionadas.
- **Limpar Substituições de Célula**: redefine todas as propriedades de célula das células selecionadas, para as configurações de **Estilo de Célula**, tal como definido no **Estilo de Tabela**.
- **Limpar Substituições de Tabela**: redefine todas as propriedades de célula de todas as células, para as configurações de **Estilo de Célula** tal como definido no **Estilo de Tabela**.

7 (opção) Repita as etapas 3 e 4 para continuar a editar a tabela.

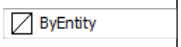
8 (opção) Dê um duplo clique em uma célula para editar o texto da célula.

9 (opção) Faça uma das seguintes:
- Clique em outra célula.
- Pressione e mantenha o botão esquerdo do mouse, então arraste o cursor para selecionar múltiplas células.

10 Clique fora da tabela para fechar a barra **Tabela**.

Ferramentas de Tabela

Ícone	Ferramenta	Descrição
	Inserir linha acima	Insere uma linha vazia acima da seleção.
	Inserir linha abaixo	Insere uma linha vazia abaixo da seleção.
	Excluir linha	Exclui linha(s) da seleção.
	Inserir coluna à esquerda	Insere uma coluna vazia à esquerda da seleção.
	Inserir coluna à direita	Insere uma coluna vazia à direita da seleção.
	Excluir coluna	Exclui coluna(s) da seleção.
	Mesclar	Mescla as células selecionadas.
	Desfazer mesclagem	Divide células mescladas anteriormente.
	Estilo da borda	Define o estilo de borda da seleção (ainda não implementado).

Ícone	Ferramenta	Descrição
	Alinhamento texto	Define o alinhamento do texto da seleção.
	Bloquear / Desbloquear	Bloqueia / Desbloqueia a seleção.
	Definir formato da célula	Define o formato da célula da seleção (ainda não implementado)
	Definir estilo da célula	Define o estilo das células selecionadas: **Título**, **Cabeçalho** ou **Dados**.
	Definir fundo da célula	Define a cor de fundo da seleção.

14.2.9.6 Editar uma tabela no painel Propriedades

1 Clique em uma das linhas da grade.

1 As propriedades da tabela são exibidas no painel **Propriedades**.

2 (opção) Aplicar um **Estilo de Tabela** diferente.

3 (opção) Alterar a direção da tabela.

4 (opção) Modificar a largura e/ou altura da tabela.

5 (opção) Modificar as margens vertical e/ou horizontal das células.

6 Pressione a tecla Esc para parar a edição.

14.2.9.7 Editar uma célula na barra de Propriedades

1 Clique na célula.

1 As propriedades da célula são exibidas no painel **Propriedades**.

2 (opção) Modificar a largura e/ou altura da célula.

3 A largura e a altura da coluna e/ou linha das células são alteradas.

4 (opção) Modificar a rotação do texto.

5 As opções são: 0°, 90°, 180° e 270°.

6 (opção) Editar o conteúdo da célula.

7 (opção) Escolha uma opção diferente para alinhamento do texto.

8 Pressione a tecla Esc para parar a edição.

14.2.9.8 Examinar fórmulas

Atualmente, fórmulas são parcialmente suportadas. Há fórmulas mais complexas que ainda não são suportadas, mas na maioria dos casos as fórmulas nas planilhas do Excel permanecerão funcionais

quando a planilha do Excel for importada como uma tabela. A edição e a criação de fórmulas por meio da caixa de diálogo de Campo ainda não são suportadas, mas a edição/criação 'in-loco' é possível. Por exemplo: se você digitar = SUM(A2:A4) em uma célula, esta será interpretada como fórmula, e a soma dos valores nas células A2 até A4 é calculada.

14.2.9.9 Para editar o conteúdo da tabela

1 Siga um dos procedimentos:
   - Duplo-clique na célula que você deseja editar.
   - Inicie o comando EDITARTABELA.
   - **Você é solicitado**: Escolha a célula da tabela para editar.

     Clique na célula que deseja editar.

   A barra de ferramentas **Formatação de Texto** é exibida acima da célula selecionada (consulte Criar textos).

2 Editar a célula.

3 (opção) Pressione Alt + Enter para criar uma segunda linha na mesma célula.

4 Para editar outra célula:
   - Pressione a tecla Tab para ir para a próxima célula, na mesma linha.
   - Pressione Shift + Tab para ir para a célula anterior, na mesma linha.
   - Pressione Enter para ir para a próxima célula, na mesma coluna.
   - Pressione Shift + Enter para ir para a célula anterior, na mesma linha.
   - Pressione uma das teclas de seta.

5 Clique fora da tabela para interromper a edição.

14.2.9.10 Para exportar uma tabela

Ao exportar dados para um arquivo *.CSV, (Comma Separated Values), o caractere de separador de lista do seu sistema é usado no arquivo *.CSV (consulte Definir o caractere separador de lista).

1 Inicie o comando EXPORTARTABELA.

1 **Você é solicitado**: Selecione a tabela para exportar:

2 Clique na tabela da qual você deseja exportar os dados.

3 A caixa de diálogo **Exportar Dados** aparece.

4 Selecione uma pasta para onde quer salvar os dados.

5 Digite um nome no campo **Nome de arquivo**.

6 Clique o botão **Salvar**.

   Os dados na tabela selecionada são salvos como um arquivo *.CSV.

14.2.9.11 Configurar o caractere separador de lista

1 Abra a caixa de diálogo **Configurações** do Windows.

2 Abra a categoria de configurações **Regiões**.

3 Clique em **Data, hora e configurações regionais adicionais**.

4 Clique em **Alterar formatos de data, hora ou número**.

5 Clique em **Configurações adicionais**.

6 Defina o caractere **Separador de lista**.

7 Clique em **OK** para confirmar.

14.3 Annotative Entities

14.3.1 Entidades anotativas

Anotações são todas entidades em seu desenho que não são um modelo ou uma geometria, como dimensões, textos, símbolos, etc. Um problema comum com anotações é que muitas vezes elas são dependentes da escala. Algumas anotações devem ser dimensionadas para a escala de plotagem, ou devem ser mostradas apenas em determinadas escalas.

Estilos anotativos e entidades mudam a maneira em que eles são exibidos (tamanho, localização e orientação) dependendo do fator de escala atual e da viewport.

As seguintes entidades podem ter a propriedade anotativa:

- texto e texto de multilinhas
- hachuras
- blocos e atributos
- dimensões e tolerâncias
- chamadas e multi-chamadas

Os seguintes estilos podem ter a propriedade anotativa:

- estilos de texto (ver o comando ESTILO)
- estilos de cota (ver comando ESTILOCOTA)
- estilos de multi-chamada (consulte o comando ESTILOLINCHMULT)

Notas:

- Entidades que são criadas usando um estilo anotativo, obtém automaticamente a atual escala de anotação atribuída.
- Quando um estilo anotativo é aplicado a uma entidade não-anotativa, a propriedade anotativa não é automaticamente atribuída.

14.3.1.1 Comandos

RESTAURARANOT: Redefine todas as representações de escala das entidades anotativas selecionadas para a localização de sua atual representação de escala.

ATUALIZANOT: Atualiza as entidades anotativas selecionadas para combinar com sua definição de estilo atual.

ESCALAOBJETO: Adiciona ou remove escalas suportadas para entidades anotativas através de uma caixa de diálogo.

-ESCALAOBJETO: Adiciona ou remove as escalas suportadas para entidades anotativas, pela linha de Comando.

EDITARLISTAESCALAS: Edita a lista de escala de um desenho, usada na seção **Escala plotagem** das caixas de diálogo **Imprimir** e **Configurar Página** e a propriedade **Escala padrão** de uma viewport do Paper Space. Define as escalas de anotação disponível para entidades anotativas.

-EDITARLISTAESCALAS: Edita a lista de escalas de um desenho, pela linha de Comando.

14.3.1.2 Variáveis de sistema

ANNOALLVISIBLE: Ativa/desativa a exibição de entidades anotativas que não suportam a atual escala de anotação.

ANNOTATIVEDWG: Especifica se um desenho vai ou não se comportar como um bloco anotativo quando estiver inserido dentro de outro desenho.

CANNOSCALE: Define o nome da escala de anotação atual para o espaço atual. Somente escalas existentes na lista de escalas atual são aceitas. A configuração CANNOSCALE é salva com Model Space e com cada viewport nos vários layouts. Quando você cria uma entidade anotativa, a escala anotativa atual é automaticamente aplicada.

CANNOSCALEVALUE: Exibe o valor da atual escala de anotação. Por exemplo, o valor da escala 1:2 é 0.5.

HPANNOTATIVE: Controla se novas hachuras serão anotativas.

MSLTSCALE: Se Ligado, os tipos de linha no Model Space são escalados para a atual escala de anotação. Um Regen é necessário quando você alterna o valor de MSLTSCALE. Essa variável é equivalente à variável PSLTSCALE (escala de tipo de linha do Paper Space) no Paper Space.

MSOLESCALE: Controla o tamanho de um objeto OLE que contém texto quando colado no Model Space. A variável de sistema MSOLESCALE afeta apenas o dimensionamento inicial. Objetos OLE existentes não são afetados se a variável for modificada.

SAVEFIDELITY: Controla se a exibição atual do desenho é preservada quando aberto em um programa que não oferece suporte a entidades anotativas.

SELECTIONANNODISPLAY: Controla se todas as representações de escala são exibidas quando uma entidade anotativa é selecionada.

14.3.1.3 Definir a atual escala de anotação

Para definir a atual escala de anotação, faça o seguinte:

- Clique no campo **Escala de anotação** sob **Misc** na barra **Propriedades**, e pressione a seta para baixo e selecione uma escala na lista.
- Clique o botão-direito no campo **Escala de Anotação** na barra **Status**, e selecione uma escala na lista. Se o campo **Escala de Anotação** não estiver disponível, clique na seta para baixo no lado direito da barra de **Status** e escolha **Escala de Anotação** na lista.

Nota: Somente escalas que existem na lista **Escala** do desenho estão disponíveis. Use o comando EDITARLISTAESCALAS para editar a lista **Escala**.

14.3.1.4 Para tornar uma entidade de anotação anotativa

Entidades de anotação que são criadas usando um estilo não-anotativo podem ser tornadas anotativas usando o procedimento a seguir:

1 Selecione a entidade de anotação.

2 No painel **Propriedades**, selecione a propriedade **Anotativo**.

3 Selecionar **Sim**.

A atual escala de anotação é aplicada para a entidade selecionada.

14.3.2 Annotation scales

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 you 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 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 system variable is active or when using the **Quad** cursor menu, an icon is displayed next to the highlighted entity, indicating whether a single annotation scale () applies to an annotative entity or multiple ().

14.3.2.1 To add or remove annotation scales

The following procedure allows to:

- Check which annotation scales currently apply.
- Add annotation scales.
- Remove annotations scales.

1 Select the annotative entity or entities.

2 In the **Properties** panel, select the **Annotative Scale** field.

3 Click the **Browse** button () of the **Annotative Scale** field.

The **Annotation Object Scale** dialog box appears.

4 (option) If multiple entities were selected, choose whether to list all scales for the selection, or only the common scales.

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 to the list.

6 Click the **OK** button to close the **Annotation Object Scale** dialog box.

The changes are applied to the selection set.

Nota: Only scales that exist in the **Scale** list of the drawing are available. Use the SCALELISTEDIT command to edit the **Scale** list.

14.3.3 Textos anotativos

Entidades de texto são criadas usando o estilo de texto atual:

- Se o estilo de cota for anotativo, as dimensões obtém automaticamente propriedade anotativas, e a atual escala de anotação é atribuída.
- Se o estilo de cota não é anotativo, você precisa fazer manualmente as dimensões anotativas.

Textos anotativos tem duas propriedades de altura:

- **Altura do texto no modelo**: a altura exibida na tela, no Model Space, no que diz respeito a atual escala de anotação.
- **Altura do texto no papel**: a altura no papel se impresso na escala de anotação suportada.

14.3.3.1 Para tornar uma entidade de texto anotativa

1 Selecione a entidade de texto.

1 As propriedades da entidade de texto selecionada são exibidas no painel **Propriedades**.

2 No painel **Propriedades**, selecione a propriedade **Anotativa**.

3 Escolha **Sim** na lista de opções.

A escala de anotação atual é atribuída.

O texto não é escalado: a altura atual do texto se torna a **Altura do texto no modelo**.

14.3.3.2 Para remover a propriedade anotativa

1 Selecione a entidade de texto.

1 As propriedades da entidade de texto selecionada são exibidas no painel **Propriedades**.

2 No painel **Propriedades**, selecione a propriedade **Anotativa**.

3 Escolha **Não** na lista de opções.

Se a entidade de texto suportar várias escalas de anotação, o texto será redimensionado para sua escala inicial.

14.3.4 Hachuras anotativas e preenchimentos em gradiente

A variável de sistema HPANNOTATIVE controla se novas entidades de hachura ou gradiente são anotativas.

O HPANNOTATIVE está Desligado por padrão. A variável do sistema não é salva e é redefinida para seu valor padrão quando o BricsCAD fecha.

14.3.4.1 Para definir a variável do sistema HPANNOTATIVE

Siga um dos procedimentos:

- Na caixa de diálogo **Configurações**, dob **Desenho** > **Criação de entidade** > **Hachuras**.
- Marque a opção **Anotativa** na caixa de diálogo **Hachura e Gradiente**.
- Escolha a opção **Anotativa** no prompt de Comando do comando -HACHURA.

14.3.4.2 Para tornar anotativas uma entidade hachura ou gradiente

1 Selecione a entidade de hachura ou de preenchimento em gradiente.

1 As propriedades da entidade selecionada são exibidas no painel **Propriedades**.

2 Selecione a propriedade **Anotativa**.

3 Escolha **Sim** na lista de opções.

A escala de anotação atual é aplicada à entidade de hachura ou gradiente selecionada, e a hachura é escalada de acordo.

14.3.4.3 Para remover a propriedade anotativa

1 Selecione o preenchimento de hachura ou gradiente.

1 As propriedades da entidade selecionada são exibidas no painel **Propriedades**.

2 Selecione a propriedade **Anotativa**.

3 Escolha **Não** na lista de opções.

14.3.5 Annotative blocks

The annotative property of a block is either set when creating the block definition using the BLOCK command, or next, in the **Drawing Explorer / Blocks** dialog.

Annotative blocks can have an optional property to always match the orientation of a layout, independent from the rotation of the paper space viewport content.

14.3.5.1 To make a block definition annotative

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box appears.

2 If necessary, choose **Detail View**.

Annotative block definitions have the **Annotative** icon () in the **Annotative** column.

3 Click in the **Annotative** field of a block definition to toggle the annotative property on/off.

14.3.5.2 To set the Match Orientation to Layout property

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer / Blocks** dialog box is displayed.

2 If necessary, choose **Detail View**.

3 Tick the **Match Orientation to Layout** checkbox of a block definition to set the property.

14.3.6 Dimensoes anotativas

Entidades de dimensão são criadas usando o atual estilo de cota:

- Se o estilo de cota for anotativo, as dimensões obtém automaticamente propriedade anotativa, e a atual escala de anotação é atribuída.
- Se o estilo de cota for não-anotativo, você precisa manualmente tornar as dimensões anotativas.

Notas:

- A configuração **Escala Geral de Dimensão** (variável de sistema DIMSCALE) não é editável para estilos de dimensão anotativa, e é negligenciada para dimensões anotativas.
- Recomenda-se definir Escala de **Escala Geral de Dimensão** = 1 para estilos de cota anotativa.

14.3.6.1 Para tornar uma entidade de texto anotativa

1 Selecione a entidade de dimensão.

1 As propriedades da entidade de dimensão selecionada são exibidas no painel **Propriedades**.

2 Selecione a propriedade **Anotativa**.

3 Escolha **Não** na lista de opções.

A entidade de dimensão é escalada em relação ao valor atual da variável de sistema DIMSCALE.

14.3.7 Chamadas anotativas

Multi-chamadas são criadas usando o estilo atual de Multi-chamada:

- Se o estilo Multi-chamada for anotativo, o LinchMult entidade obtém automaticamente a propriedade anotativa, e a escala de anotação atual é atribuída.
- Se o estilo de Multi-chamada não for anotativo, você precisará tornar manualmente a entidade Multi-chamada anotativa.

14.3.7.1 Para tornar uma entidade Multi-chamada anotativa

1 Selecione a entidade **LinchMult**.

As propriedades da entidade **LinchMult** selecionada são exibidas no painel **Propriedades**.

2 Selecione a propriedade **Anotativa**.

3 Escolha **Sim** na lista de opções.

A escala de anotação atual é atribuída.

A entidade **LinchMult** é escalada em relação à escala de anotação atual.

14.3.7.2 Para remover a propriedade anotativa

1 Selecione a entidade **LinchMult**.

As propriedades da entidade **LinchMult** selecionada são exibidas no painel **Propriedades**.

2 Selecione a propriedade **Anotativa**.

3 Escolha **Não** na lista de opções.

15. Personalização

15.1 Startup options

When starting up BricsCAD, it is possible to add arguments or switches. For example:

- The Command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" starts BricsCAD and loads the specified drawings.
- The Command line "bricscad.exe /P MyProfile" starts BricsCAD using the MyProfile user profile.

"/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 works. 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 uses the current profile (= the most recently used profile) which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, then 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).
/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 and on every new or opened drawing depending 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.
/Safemode	Prevents all third party plug-in code, enablers and user lisp files from loading in the current session. Starting in a clean environment can prevent potential crash causes. See Launching BricsCAD in safe mode.
/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.

Switch	Description
regserver unregserver	To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/".

Nota:

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile".
- Switches can be combined, for example **/P MyProfile /L /S C:\BricsCAD** starts 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.
- If GETSTARTED = ON when you launch BricsCAD, the **BricsCAD Launcher** dialog box appears, which lets you choose the workspace.

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

15.2 Arquivos de usuario BricsCAD

No BricsCAD os arquivos do usuário são armazenados em pastas específicas:

- **PlotConfig/Configuração de Plotter**: arquivos de configuração do plotter (*.pc3)
- **PlotStyles/Estilos de plotagem**: arquivos de tabela de plotagem por estilos (*.stb) e arquivos de tabela de plotagem por cores (*.ctb)
- **Suporte**:
  - arquivos de padrão de hachura (*.pat)
  - arquivos de estilo de linha (*.lin)
  - arquivos de alias (*.pgp)
  - arquivos de unidade (*.unt)
  - arquivos de interface de usuário personalizados (*.cui)
  - Arquivos de biblioteca BIM (*.bsyslib) consulte o comando BIMPROJETOINFO.

- Arquivos de configuração da árvore da estrutura (*.cst) consulte o comando PAINELESTRUTURA.
- Arquivos de configuração do painel da biblioteca (*.ccf) consulte o comando LIBRARYPANELOPEN.

- **Templates**: desenhos de modelo (*.dwt)

Plataforma	Pastas	Subpastas
Windows	C:\Users\<UserName>\AppData\Local \Bricsys \BricsCAD\<Version>\pt_BR	\Templates
Windows	C:\Users\<UserName>\AppData\Roaming \Bricsys\BricsCAD\<Version>\pt_BR	\PlotConfig \eStiloplot \Suporte
Mac	/Users/<UserName>/Library/Preferences/ Bricsys/BricsCAD/<Version>/pt_BR	/PlotConfig /eStiloplot /Suporte /Templates
Linux	/home/<UserName>/Bricsys/BricsCAD/ <Version>/pt_BR	/PlotConfig /eStiloplot /Suporte /Templates

15.2.1 O gerenciador de arquivos de usuário BricsCAD

Quando você atualiza, na primeira vez que você inicia o BricsCAD após a atualização, o conteúdo da subpasta **UserDataCache** da pasta de instalação do BricsCAD é comparado com o conteúdo das **pastas de arquivos do usuário**. Se você tiver arquivos personalizados nessas pastas e se um ou mais dos arquivos correspondentes na pasta **UserDataCache** forem mais recentes, a caixa de diálogo **Gerenciador de arquivos do usuário do BricsCAD** será exibida, perguntando se deseja manter ou substituir seus arquivos pela nova versão.

Siga um dos procedimentos:

1 Selecione um arquivo na lista, e clique no botão **Sim** para substituir ou clique no botão **Não** para manter seu arquivo.
   - Repita este procedimento para cada arquivo na lista.
   - Clique o botão **Sim para tudo** para substituir todos os seus arquivos com os arquivos mais recentes do BricsCAD.
   - Clique o botão **Parar de Copiar** para manter todos os seus arquivos.

2 Se você decidiu não atualizar um ou mais arquivos, uma janela de alerta será exibida, listando todos

os arquivos que você deseja manter e que, portanto, não serão atualizados. Você é perguntado se deve ou não ser lembrado na próxima vez que iniciar o BricsCAD.

3 Siga um dos procedimentos:

- Clique no botão **Sim** se quiser atualizar um desses arquivos mais tarde.

  Nesse caso, este procedimento é reiniciado na próxima vez que você iniciar o BricsCAD.

- Clique o botão **Não** se você não quer ser lembrado.

15.3 Loading and unloading CUI files

When you install BricsCAD, the default CUI files are copied to the **Support** folder (see BricsCAD User Files).

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: replaces the currently loaded main CUI file.
- Load partial CUI file: partially replaces the content to the currently loaded main CUI file.

15.3.1 Loading a main CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load main CUI file ...** .

The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The selected CUI file is loaded.

4 Close the **Customize** dialog.

15.3.2 Loading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load partial CUI file ...** .

The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

The content of the selected CUI file is appended: definitions which are different from the ones in the main CUI are used.

4 Close the **Customize** dialog.

Nota: 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.

15.3.3 Unloading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 Click the **Menus** tab.

3 Right-click the partial CUI name and choose **Unload <PartialCuiName>** in the context menu.

4 Close the **Customize** dialog.

15.3.4 Managing customization groups

1 Launch the MENULOAD command.

1 The **Customization groups** dialog opens.

2 (Option) To load a partial CUI file, click the browse button (...) to select a partial CUI file, then click the **Load** button.

3 (Option) To unload a partial CUI file, select the partial CUI file in the **Loaded Customization Groups** list, then click the **Unload** button.

4 Click the **Close** button to stop.

15.4 Creating a partial CUI file

1 Launch the CUSTOMIZE command.

2 Do one of the following:

- In the **File** menu, choose **Create new partial CUI file ...**
- Place the cursor on a Menu group, then right-click and choose **Create new partial CUI file ...** in the context menu.

The **Create a Customization File** dialog opens.

By default the new partial CUI file will be created in the **Support** folder.

3 (Option) Select a different folder to save the CUI file in.

4 Type a name for the new CUI file in the **File name** field.

5 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**, **Toolbars**, **Ribbon**, **Keyboards**, **Mouse**, **Tablet** and **Quad** tab and in the **Available Tools** pane of the different tabs.

15.5 Managing toolboxes

A toolbox contains a set of tools, which can be added to menus, toolbars, ribbon panels, Quad tabs and keyboard or mouse shortcuts. Toolboxes can be created in the main CUI or in a partial CUI.

15.5.1 Creating a toolbox in a menu group

1 In the **Customize** dialog box, select a menu group in the **Available Tools** panel:

- BricsCAD (= default main CUI)
- <PartialCuiName> (= partial CUI)

2 Right-click and choose **Append toolbox** in the context menu.

The **Add Toolbox** dialog box appears.

3 Type a name for the new toolbox in the **Name** field, then click the **OK** button.

The toolbox is added in the **Available Tools** tree.

15.5.2 Creating a tool in a toolbox

1 In the **Customize** dialog box, select a toolbox in the **Available Tools** panel.

2 Right-click and choose **Append tool** in the context menu.

The **Add tool** dialog box appears.

3 **Title**: type a name for the new tool.

4 **Help**: type a help string.

5 **Command**: type a command or a command macro*.

6 **Image**: Click the **Browse** button (…).

The **Tool Image** dialog box appears.

7 Do one of the following:

- Select **Builtin**, then choose one of the built-in icons.
- Select **Bitmap file**, first choose to select one image file or a small and a large image file, then click the **Browse** button (…) to select the image file(s).

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

* Syntax to create command macros:

- Start the macro with **^c^c**, the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- Type an underscore (_) in front of a command name if you want to use the macro in a non-English version of BRICSCAD.
- A period (.) in front of a command name ensures that the macro still works if this command is temporarily undefined.
- 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, e.g. to accept a default prompt option.
- A backslash (\) temporarily pauses the macro to allow user input, e.g. specifying a point.

In the following macro: ^c^c_-layer;m;InteriorWalls;;mline

- **^c^c** concludes a running command, if any.
- **-layer;** launches the -LAYER command.
- **m;** selects the **Make new current layer** option.
- **InteriorWalls** makes the InteriorWalls layer current and creates it first if necessary.
- **;;mline** concludes the -LAYER command and launches the MLINE command.

15.6 Creating toolbars

1 In the **Customize** dialog box, 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.

  The new toolbar will be added below the last existing toolbar.

- Select an existing toolbar in a menu group, then right-click and choose **Insert toolbar** in the context menu.

  The new toolbar will be added above the selected toolbar.

The **Add toolbar** dialog box appears.

3 Type a name in the **Title** field.

4 Click the **OK** button.

The toolbar is created.

5 Define the properties of the toolbar:

- **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 appears or not when the CUI file is loaded.
- **Rows**: Specifies the number of rows when its position is floating. Toolbars 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 Add items to the toolbar.

15.6.1 To add a flyout to a toolbar

1 In the **Customize** dialog box, click the **Toolbars** tab.

2 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.

3 The **Add flyout** dialog box appears.

4 Type a name in the **Title** field of the **Add flyout** dialog.

5 Click the **OK** button.

A new empty toolbar is created and inserted as a flyout in the selected toolbar.

6 Define the flyout properties of the toolbar.
   - **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.
   - **Target alias**: Identifies the referenced toolbar.
   - **Own Icon**:
   - **No**: The icon of the first tool or the icon of the most recently used tool is displayed in the parent toolbar.
   - **Yes**: The selected icon is displayed in the parent toolbar.

7 Define the properties of the toolbar when the flyout is teared off and inserted as a floating toolbar.

8 Add items to the toolbar.

9 Items added to the flyout are automatically added to the toolbar and vice versa.

15.7 Creating menus

Main menus sit in the menu bar at the top of the BricsCAD application window. Submenus show as a flyout of a main menu.

The MENUBAR system variable controls the display of menus.

15.7.1 Creating a main menu

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:
   - Click the **Main Menus** subgroup of a menu group, then right-click and choose **Append main menu** in the context menu.

     The new menu will be added below the last existing menu.
   - Select an existing menu in the **Main Menus** subgroup, then right-click and choose **Insert main menu** in the context menu.

     The new menu will be added above the selected menu.

   The **Add Main Menu** dialog box appears.

3 Type a name in the **Title**.

4 Click the **OK** button.

   The main menu is created.

5 Add items to the menu.

Nota: The new main menu is not visible in the menu bar until at least one menu item is added to it.

15.7.2 Adding a submenu to a main menu

1 In the **Customize** dialog, click the **Menus** tab.

   If collapsed, expand the menu group.

If collapsed, expand the **Main menus** group of the menu group.

2 Do one of the following:
   - Select a main menu, then right-click and choose **Append submenu** in the context menu.
     The new submenu will be added below the last existing submenu.
   - Select an existing submenu, then right-click and choose **Insert submenu** in the context menu.
     The new submenu will be added above the selected submenu.

   The **Add submenu** dialog box appears.

3 Type a name in the **Title** field.

4 Click the **OK** button.

   The submenu is created.

5 Add items to the submenu.

15.7.3 Adding a submenu to a submenu

1 In the **Customize** dialog, click the **Menus** tab.

2 Expand the submenu.

3 Select an item in the submenu, then right-click and choose **Insert submenu** in the context menu.

   The **Add submenu** dialog box appears.

4 4. Type a name in the **Title** field.

5 Click the **OK** button.

   The submenu is created.

6 Add items to the submenu.

15.8 Creating context menus

The availability of context aka shortcut menus is controlled through the SHORTCUTMENU system variable. There are three context menu modes:

- **Default mode**: Displays the **Default** context menu when no command is active and nothing is selected.
- **Edit mode**: Displays the **Edit** context menu when a selection set exists. This mode is context sensitive. If a specific shortcut menu for the selected entity or entities exists, its content is added to the **Edit** context menu. For example, the content of the **Multiline Text Object** context menu is available in the **Edit** context menu when an MText entity is selected.
- **Command mode**: Displays the **Command** context menu when a command is running.

Shortcut menus appear when you right-click. However:

- When bitcode 16 of the SHORTCUTMENU system variable is set, a right-click repeats the previous command and you need to hold down the right mouse button longer (= long right-click) to display shortcut menus.
- When bitcode 4 of the QUADDISPLAY system variable is set (= Show Quad on right-click), shortcut menus only show when bitcode 16 of the SHORTCUTMENU system variable is set.

Special shortcut menus are:

- **Grips**: Appears on a right-click when using grips to edit an entity.
- **Entity Snap**:
  - MBUTTONPAN = OFF: appears when you click the mouse wheel (middle mouse button).
  - MBUTTONPAN = ON: displays on Shift + Right-Click

15.8.1 Creating shortcut menus

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:
  - Click the **Context Menus** subgroup of a menu group, then right-click and choose **Append context menu** in the context menu.

    The new context menu will be added below the last existing context menu.
  - Select an existing context menu in the **Context Menus** subgroup, then right-click and choose **Insert context menu** in the context menu.

    The new context menu will be added above the selected context menu.

  The **Add Context Menu** dialog box appears.

3 Type a name in the **Menu title** field.

4 (option) To create a regular context menu:
  - Choose **As regular context menu**.
  - Click the **Add Context…** button.

    The **Add context alias** dialog box appears.
  - Select a context alias. The options are: CMCOMMAND, CMDEFAULT, CMEDIT, GRIPS and SNAP.
  - This controls when the context menu is available. See above.
  - (option) add another context alias.
  - Click the **OK** button to create the context menu.

5 (option) To create a context sensitive context menu:
  - Choose **As context menu on specific entities**.
  - Click the **Add Entity Types…** button.

    The **Add Entity Alias** dialog appears.
  - Tick the **One entity** and/or **Multiple entities** boxes.
  - Select one or more entities in the list, then click the **OK** button.
  - (option) Click the **Add Custom…** button add a custom alias.
  - Click the **OK** button to create the context menu.

6 Add tools to the context menu.

15.9 Creating keyboard shortcuts

1 In the **Customize** dialog box, click the **Keyboard** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append shortcut** in the context menu.

     The new shortcut will be added below the last shortcut.
   - Select an existing shortcut in a menu group, then right-click and choose **Insert shortcut** in the context menu.

     The new shortcut will be added above the selected shortcut.

   The **Add keyboard shortcut** dialog box appears.
3 Do one of the following:
   - Select **Add an existing tool**, then select an existing tool.
   - Select **Add a new tool**, then create a new tool.

     See the **Create a tool in a toolbox** procedure in **Managing Toolboxes**.
4 In the **Properties** grid of the new shortcut, select the **Key** field, then press the key combination you want to assign to the shortcut.

15.10 Customizing the Quad

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

- Right-click the **QUAD** field in the **Status** bar and choose **Customize Quad items...** in the context menu.
- Run the CUSTOMIZE command, then select the **quad** tab.

15.10.1 Creating a quad tab

1 Do one of the following:
   - Right-click a menu group and choose **Append quad tab** in the context menu.

     The quad tab will be added below the last existing quad tab in the menu group.
   - Right-click an existing quad tab and choose **Insert quad tab** in the context menu.

     The quad tab will be added above the selected quad tab.

   The **Add Quad Tab** dialog box appears.
2 Type a name in the **Label** field.

   The label appears in the Quad cursor menu.
3 Type a name in the **Title** field.

   The tab title must be unique in the menu group.
4 Click the **OK** button.
5 Add tools to the tab.

15.10.2 Copying a quad tab

You can insert a copy of a quad tab in the same menu group or to another menu group.

- To insert a copy of a quad tab, select the tab, then right-click and choose **Insert Copy** in the context menu.

The **Add Quad Tab** dialog box appears.

Specify a label and title (see above), then click the **OK** button.

- To insert a copy of quad tab in another menu group: drag-and-drop the quad tab onto the target menu group.

15.10.3 Setting the Entity Filter property of a tool

The Quad cursor menu is context sensitive. The available tools depend on the entity under the cursor or the current selection set. When no selection exists, the 'No Selection Quad' appears on a right-click if bitcode 4 of the QUADDISPLAY system variable is selected.

1 Select the tool.

1 The **Quad Button** properties are displayed in the bottom section of the **Customize** dialog box.

2 Select the **Entity Filter** property.

The default setting is NO_SELECTION.

3 Click the **Browse** button (…) at the right hand side of the field.

The **Edit Quad Button Filter** dialog box appears.

4 (option) Click the **Add Filter…** button.

The **Add quad button filter** dialog box appears.

Click the down arrow at the right hand side of the field and choose a filter:

- NO_SELECTION: available when nothing is selected.
- OBJECT_ANY: available when a single entity is selected.
- OBJECTS_ANY: available when multiple entities are selected.

Click the **OK** button.

5 (option) Repeat the previous step to add a filter or remove a filter.

6 (option) Click the **Add Entity Type(s)…** button.

The **Add Entity Alias** dialog box appears.

- Tick the **One entity** and/or **Multiple entities** boxes.
- Tick the box the entity type(s) the tool applies to.

Click the **OK** button.

Nota: It is not possible to modify the Entity Filter of a built-in Quad Button.

15.11 Adding tools to user interface controls

Tools can be drag-and-dropped from the toolboxes in the **Available Tools** section of the **Customize** dialog box onto menus, context menus, toolbars, rows in a ribbon panel, keyboard shortcuts and Quad tabs.

1 Select the target in the **Customize** dialog box.

2 In the **Available Tools** section, expand the source toolbox.

3 Select the tool and drag-and-drop it on the target item.

4 (option) Expand **Controls** in the **Available Tools** section, then drag-and-drop a control onto the target item.

5 (option) Drag-and-drop tools to reposition them or to move them to a different target item.

15.12 Personalizar a Faixa de opções

A Faixa de opções é organizada em uma série de abas, as quais por sua vez são compostas por painéis. Os painéis de Faixa são compostos por linhas com ferramentas, controles e sub-painéis.

As abas da Faixa de opções controlam quais os painéis que serão exibidos na Faixa de opções.

Para exibir uma aba na Faixa de opções, você deve adicionar a aba a uma área de trabalho.

15.12.1 Criar uma aba na Faixa de opções

1 Na caixa de diálogo **Personalizar**, clique na aba **Faixa de opções**.

2 Siga um dos procedimentos:

- Expanda (clique no '+') de um grupo de menus, clique o botão-direito em **Abas Faixa de opções** e escolha **Acrescentar aba na faixa de opções**, no menu de contexto.

  A nova aba da Faixa será adicionada abaixo da última aba da Faixa de opções existente.

- Selecione uma aba existente em um grupo de menu, então clique o botão-direito e escolha **Inserir aba na faixa** no menu de contexto.

  A nova aba será adicionada acima da aba selecionada da faixa.

A caixa de diálogo **Adicionar Aba na Faixa** de opções é exibida.

3 Definir a Aba da Faixa:

- **Rótulo**: Nome da aba onde será exibida na Faixa de opções.
- **Título**: Nome longo (explicativo) da aba.
- **ID**: Marca exclusiva para identificar a aba no grupo de menus. O ID é usado para fazer programaticamente referência à aba na Faixa de opções.

4 Clique no botão **OK** para criar a aba da Faixa de opções.

15.12.2 Criar um painel na Faixa de opções

1 Na caixa de diálogo **Personalizar**, clique na aba **Faixa de opções**.

2 Siga um dos procedimentos:

- Expanda um grupo de menus, e clique o botão-direito em **Painéis da faixa de opções** e escolha **Acrescentar painel na faixa de opções** no menu de contexto.

  O novo painel da faixa de opções será adicionado abaixo do último painel existente na faixa de opções.

- Selecione um painel da faixa de opções existente em um grupo de menus, então clique o botão-direito e escolha **Inserir painel na faixa de opções** no menu de

  O novo painel da faixa de opções será adicionado acima do painel selecionado na faixa de opções.

A caixa de diálogo **Adicionar Painel na Faixa** é exibida.

3 Definir o Painel da Faixa de opções:
   - **Rótulo**: Nome do painel, conforme exibido na faixa de opções.
   - **Título**: Nome longo do painel.
   - **ID**: Identificador exclusivo para identificar painel no grupo de menus. O ID é usado para fazer programaticamente referência à aba da faixa de opções.

4 Clique no botão **OK** para criar o painel na faixa de opções.

15.12.3 Adicionar um painel de faixa de opções a uma aba

1 Na caixa de diálogo **Personalizar**, clique na aba **Faixa de opções**.

2 Expanda as **Abas da faixa de opções** em um grupo de menu.

3 Siga um dos procedimentos:
   - Clique o botão-direito na aba da faixa de opções, e escolha **Acrescentar painel** no menu de contexto.

     O novo painel da faixa de opções será adicionado abaixo do último painel existente da faixa de opções.
   - Expanda uma aba da faixa de opções, então clique o botão-direito e escolha **Painel inserir** no menu de contexto

     O painel da faixa de opções será adicionado acima do painel selecionado.

   A caixa de diálogo **Selecionar painel da faixa** é exibida.

   Todos os painéis de faixa de opções existentes no grupo de menus são listados.

4 Selecione um painel na lista e clique duas vezes ou clique no botão **OK**.

   O painel é adicionado à aba.

15.12.4 Adicionar uma linha a um painel

1 Na caixa de diálogo **Personalizar**, clique na aba **Faixa de opções**.

2 Expanda **Painéis da faixa de opções** em um grupo de menus.

3 Selecione um painel, e faça um dos seguintes:
   - Clique o botão-direito e escolha **Acrescentar linha** no menu de contexto.

     A linha será adicionada abaixo da última linha existente no painel selecionado.
   - Selecione uma linha existente, e clique o botão-direito e escolha **Inserir linha na faixa** no menu de contexto

     A linha será adicionada acima da linha selecionada.

15.12.5 Adicionar itens a uma linha

Quando você clica o botão-direito em uma linha, as seguintes opções estão disponíveis em um menu de contexto:

- **Inserir painel em linha**: Adiciona um sub-painel contendo três linhas de sub-painel, por padrão. Você pode então adicionar um ou mais itens a essas linhas e adicionar/remover linhas.
- **Inserir botão de comando**: Consulte **Adição de ferramentas aos controles da interface do usuário**.

- **Inserir botão de divisão**: Adiciona um botão ao qual você pode adicionar vários itens. Somente o primeiro item ou o item usado mais recentemente é exibido.
- **Inserir botão de alternância**: Adiciona um botão para alternar uma configuração ou uma opção. Por exemplo, snaps de entidade ou prioridades de seleção. Um botão de alternância indica o estado da configuração ou da opção.
- **Inserir separador**: Adiciona um espaço entre as linhas.

15.12.6 Definir as propriedades de um item

Selecione o item, então defina as propriedades na grade de propriedades na seção inferior da caixa de diálogo **Personalizar**.

15.12.7 Reposicionar itens

Para reposicionar qualquer item em um painel ou linha: selecione o item, pressione e segure o botão esquerdo do mouse e mova o item para a posição desejada. Solte o botão do mouse para que este fique posicionado.

15.12.8 Excluindo itens

Selecione o item, clique o botão-direito e escolha **Excluir** no menu de contexto.

15.13 Tool palettes

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.

15.13.1 Opening the Tool Palettes panel

Do one of the following:

- Hover over a toolpanel (Windows) or an empty toolpanel area (Windows and Linux), then right-click and choose **Palettes** > **Tool Palettes** in the context menu.
- Launch the TOOLPALETTES command.

15.13.2 Setting the view options

1 Right-click an empty space of the icon area on the **Tool Palettes** panel and choose **View Options...** in the context menu.

The **View Options** dialog appears.

2 Drag the **Image Size** slider to adjust the image size.

3 Set the viewstyle: **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.

15.13.3 Customizing tool palettes

Nota: The default folder for tool palettes is defined by the TOOLPALETTEPATH system variable.

15.13.4 Opening the Customize dialog box

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

2 Choose **Customize Palettes** in the context menu.

The **Customize** dialog box appears.

15.13.5 Importing a tool palette

1 In the **Palettes** pane, right-click and choose **Import** in the context menu.

The **Import Palette** dialog box appears.

2 Select a folder, then select a palette file (*.atc, *.btc or *.xtp).

3 Click the **Open** button or double-click to import the palette.

The palette is added in the **Palettes** list.

4 Click the **Close** button to close the **Customize** dialog box.

15.13.6 Creating a palette group

1 Place the cursor on the **Palette Groups** root folder name.

2 Right-click and choose **New Group** in the context menu.

A new group folder is created.

3 Type a name for the new group.

15.13.7 Creating a palette subgroup

1 Place the cursor on an existing group name.

2 Right-click and choose **New Palette** in the context menu.

A new group folder is added as a subgroup of the selected group.

3 Type a name for the new group.

15.13.8 Adding a palette to a group

1 Place the cursor on a palette.

2 Select a palette, then drag the palette onto a palette group name.

15.13.9 Removing a palette from a group

1 Select the palette in the group.

2 Right-click and choose **Remove** in the context menu.

15.13.10 Setting a palette group current

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Set Current** in the context menu.

15.13.11 Deleting a palette group

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Delete** in the context menu.

15.13.12 Renaming 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.

15.13.13 Deleting 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.

15.13.14 Managing tools

1 Right-click the tool.

1 A context menu appears.

2 Choose an option:

- **Cut**: Removes the item from the palette. Open another panel, then right-click and choose **Paste** in the context menu.
- **Copy**: Copies the tool. Open another panel, then right-click and choose **Paste** in the context menu.
- **Delete**: Deletes the tool. A dialog box appears, asking you to confirm the deletion.
- **Rename**: Type a new name for the tool.
- **Update tool image**: Updates the image if the image source file has been modified.
- **Specify Image...**: Allows you to replace the default image.

  Choose a new image in the **Select Image File** dialog box, then double-click or click the **Open** button.
- **Remove Image...**: Restores the default image.
- **Properties**: When you click an item on a tool palette, a command is launched. The properties of the tool palette item control the execution of the command.

  The available properties on the **Tool Properties** dialog box depend on the tool type: Command, Hatch, Block or Component.

  Edit the tool properties, then click the **OK** button.

15.13.15 Creating a new tool palette

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

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.

15.13.16 Adding items to a tool palette

The following items can be added to a tool palette:

- Hatch patterns
- Blocks
- Components
- Commands

15.13.17 Adding hatch patterns

1 Select the palette you want to add a hatch pattern to in the **Tool Palettes** panel.

2 Open the **Hatch and Gradient** 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** appears.

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 selected tool palette.

6 (option) Repeat steps 4 and 5 to add more hatch patterns.

7 (option) Drag-and-drop patterns to change their order on the palette.

15.13.18 Adding blocks

Blocks can be added from:

- the current drawing.
- any drawing.
- all drawings in a selected folder.

15.13.19 Adding blocks from the current drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Blocks**.

3 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

4 Right-click and choose **Add to Current Tool Palette** in the context menu.

15.13.20 Adding blocks from any drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in the **Local Folders** list.

5 Click the **Expand** icon (+) of the containing drawing and click the **Blocks** icon.

All blocks in the selected drawing are displayed in the **Details** pane.

6 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

7 Right-click and choose **Add to Current Tool Palette** in the context menu.

15.13.21 Adding a drawing as a block

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in **Local Folders** list.

5 Right-click and choose one of the following in the context menu.

- **Add to Current Tool Palette as block**.

  Blocks are inserted using the INSERT command.

- **Add to Current Tool Palette as component**.

  Components are inserted using the BMINSERT command.

15.13.22 Adding all blocks from multiple drawings

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

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the folder, then right-click and choose **Add blocks to Tool Palettes** in the context menu.

15.13.23 Adding commands

1 Open the **Customize** dialog box.

2 In the **Available Tools** pane, on the **Customize** dialog, expand a menu group (e.g. BricsCAD).

All toolboxes in the selected menu group are listed.

3 Expand a toolbox.

4 All commands in the selected toolbox are listed.

5 Right-click a tool and choose **Add to Current Tool Palette** in the context menu.

15.14 Command aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the Command line. There is no need to type the complete name. After typing the command alias the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for the RECTANG command, whereas the last alias is even longer than the

command name. In general, command aliases can increase your speed in entering the commands using the 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 is used.

If a command can be executed transparently (= while another command is running), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

Nota:

- Unlike commands, aliases are only local and cannot be used as global commands preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

15.14.1 Editing 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 or BOUNDARY, 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 command to reload the alias file when it was externally edited.

15.14.2 Using the Customize dialog to edit the alias file

1 Click the **Command Aliases** tab on the **Customize** dialog.

2 (option) To create a new alias,

- Do one of the following:
  - Click the **Add** button.
  - Right-click an alias and choose **Add Alias** in the context menu.
  - Right-click a command and choose **Add Alias** in the context menu to create an alias for the selected command.

    The **Add Alias** dialog box appears.

- Type the command alias in the **Alias** field.
- 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.
- Click the **OK** button.

3 (option) To edit an existing alias,
- Select the alias in the list.
- Click the **Edit** button, or right-click the alias and choose **Edit Alias** in the context menu.

  The **Edit Alias** dialog box appears.
- 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.

4 (option) To delete an alias,
- Select the alias in the list.
- Click the **Delete** button, or right-click and choose **Delete** in the context menu.

  A **Confirm** dialog box appears.
- Click the **OK** button on the **Confirm** dialog box to delete the alias.

Nota: If you use these procedures to edit the .pgp file, it is not necessary to execute the REINIT command.

15.15 User profiles

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 for each of your drawing types or workflows. User profiles can also be used when multiple users share the same computer, and each user wants to create their own drawing environment.

User profiles can be exported 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 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 tools needed to **Load**, **Create**, **Set Current**, **Copy**, **Rename**, **Delete**, **Export** and **Import** user profiles.

15.15.1 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, such as the following.

- Main Cui file.
- Current Workspace. BricsCAD has six standard workspaces: Drafting, Drafting (toolbars), Modeling, Modeling (toolbars), Mechanical and BIM.

- Configuration settings: all system variables which are saved in the Registry and user preferences.
- Default plot settings.
- Settings in dialog windows, for example, search options in the **Settings** dialog.
- Project settings.
- Recent paths.
- Status bar settings.
- Properties panel settings.
- Toolbar and menu configuration.
- Worksets.

15.15.2 The User Profile Manager

To open the **User Profile Manager** dialog box:

- When BricsCAD is running: launch the PROFILEMANAGER command.
- Execute **profilemanager_app.exe** in the BricsCAD program folder (e.g. *C:\Program Files\Bricsys \BricsCAD Vx*).

In the **User Profile Manager** dialog:

- The **Current Profile** is marked.
- **Create**: Creates a new profile, using default settings.
- **Set Current**: Loads the selected profile.
- **Copy**: Creates a copy of the selected profile in the **Copy profile** dialog box.
- **Delete**: Deletes the selected profile.
- **Export...**: Exports the selected profile to a .arg file.
- **Import...**: Imports a **.arg** file as a new profile.
- **Start**: Starts BricsCAD with the selected user profile. This option is available only when the dialog box is opened using **profilemanager_app.exe** in the BricsCAD installation folder.
- **OK**: Closes the dialog box.

15.15.3 Starting BricsCAD with a specific user profile

When you restart BricsCAD, the user profile that was current when you closed the previous session is used.

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**
- Execute **profilemanager_app.exe**, then either double-click a user profile or select a user profile and click the **Start** button.

Nota:

- The **/P** option is case insensitive.

- A space is needed between the file location and /P option.
- 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 page 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".

15.16 Workspaces

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.

The WSSETTINGS command opens the **Customize** dialog box at the Workspaces tab.

Workspaces define settings and preferences regarding a specific working environment.

The following workspaces are available:

- Drafting
- Drafting (toolbars)
- Modeling
- Modeling (toolbars)
- Mechanical
- BIM

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile.

If the GETSTARTED system variable is ON, you can select the workspace on the **BricsCAD Launcher** dialog box on startup.

The availability of a startup option depends your license level.

Startup Option	Workspace	License Level
2D Drafting	Drafting	Lite or higher
3D Modeling	Modeling	Pro or higher
Mechanical design	Mechanical	Mechanical or Ultimate
BIM	BIM	BIM or Ultimate

15.16.1 Managing workspaces

Workspaces are defined in the **Workspace** tab page on the **Customize** dialog box.

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand a workspace.

3 A workspace definition is composed of:
- **Menus**: defines which menus are available.
- **Toolbars**: defines which toolbars are displayed by default.
- **Panels**: controls the visibility of dockable panels.

  Right-click a palette, then choose an option in the context menu:
  - **Show**: the panel is visible.
  - **Hide**: the panel is not visible.
  - **Don't change**: keep the current state.
- **Ribbon**: defines the available ribbon tabs.
- **Quad Tabs**: defines the order and availability of the Quad tabs.
- **On Switch**: defines the system variables and user preferences to be set, and the commands to be executed, when switching to the workspace.

15.16.2 Setting the current workspace

Do one of the following:

- Right-click the **Current Workspace** field in the **Status** bar, then select a workspace in the context menu.
- Choose a workspace in the **Workspaces** toolbar.
- Type **WSCURRENT** at the command prompt.

  You are prompted: New value for WSCURRENT <"2d DRAFTING">:

  Type the name of the new current workspace, then press Enter.

Nota:

- If the WSAUTOSAVE 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 switch workspaces if the name of the user profile refers to a workspace.

15.16.3 Creating a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Do one of the following:
- Select Workspaces, then right-click and choose **Append workspace** in the context menu.

  The new workspace will be added below the last existing workspace.
- Select an existing workspace, then right-click and choose **Insert workspace** in the context menu.

  The new workspace will be added above the selected workspace.

  The **Add workspace** dialog box appears.

3 Type a name in the **Name** field then press Enter or click the **OK** button.

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 set of Quad tabs
- An empty **On Switch** sequence

Default properties. See the **Settings** dialog box under **Program Options / Workspace**.

15.16.4 Editing a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand the workspace.

3 (option) Edit the menu set:
- Change the order: Drag a menu.
- 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 (option) Edit the toolbar set:
- Change the order: Drag a toolbar.
- 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 properties of the panels:
- Set the desired 'on switch' behavior: **Show**, **Hide** or **Don't change**.
- Select a panel, then edit its properties in the property grid.

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: drag the tab.

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.

15.16.5 Deleting a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Right-click the workspace, then choose **Delete** in the context menu.

A **Confirm** dialog box appears.

3 Press the **Yes** button to delete the selected workspace.

Nota: Do not delete default workspaces, as they are used in the **BricsCAD Launcher** dialog box. When one of the default workspaces is deleted, use the **Importing a Workspace** procedure below to restore the missing workspace from Default.cui in the .../UserDataCache\Support\en_US subfolder of the BricsCAD installation folder.

15.16.6 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 In the **Customize** dialog box, choose **Import workspaces...** in the **File** menu.

The **Choose a Customization File** dialog appears.

2 Browse to the folder where the CUI file, you want to import the workspace from, is saved.

3 By default, BricsCAD saves CUI files in the **Support** folder of the Roamable Root folder. See BricsCAD User Files.

4 Select the CUI file.

5 A dialog box appears, listing all workspaces in the selected CUI file.

6 Select the workspace(s) to import in the list, then click the **OK** button.

Press and hold the Ctrl key to select multiple workspaces.

15.17 Express tools

15.17.1 About Express tools

Express tools is a collection of productivity tools that can be added to BricsCAD. You might already know these tools from AutoCAD.

The tools are available for you to provide additional options or possibilities but they are unsupported. BricsCAD is not responsible for the successful operation of these tools.

If you want to know more about how these tools work, go to CADwiesel.

15.17.2 Adding Express tools to bricsCAD

1 Go to the Bricsys application page.

2 If Express Tools is not present on the application page, enter **Express tools** in the search field.

3 Click the Express Tools tile.

The Express Tools detail page displays.

4 Do the following:

- Select the BricsCAD version.
- Tick the **I agree to the extended privacy statement** box.
- Click the Download button.

You will receive a zip file.

5 Unzip this file to a location of your choice.

6 Without BricsCAD running, double click on the installation file and install the Express Tools.

- Select a language.
- Specify the installation folder or accept the default folder.
- Click the **Install** button.

The Express tools are installed.

The **Express.cui** file is added in the Support folder of the roamable root folder. By default this is *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\<version>\en_US\Support*

7 The next time you launch BricsCAD, the Express tools are available in:

- The Express menu.
- Four Express tools toolbars.
- The Express Tools tab on the ribbon.

15.17.3 Unloading the Express tools

1 Launch the CUIUNLOAD command.

The Customization Groups dialog box displays.

2 Select EXPRESS.

3 Click the Unload button.

15.17.4 Reloading the Express tools

Type **expressmenu** in the Command line, then press Enter.

The **Express.cui** file is reloaded as a partial cui and the Express tools menu, toolbars and ribbon tab are available again.

15.18 Lisp

How to create LISP routines for drawing setup.

15.18.1 BricsCAD Lisp

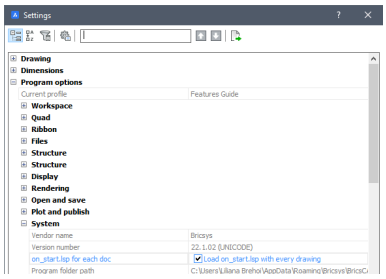
Migrating from AutoCAD

When you start migrating to BricsCAD, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your code loads 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's 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 them as **on_start.lsp** or **on_doc_load.lsp** respectively.

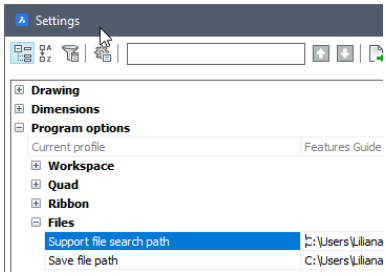
BricsCAD Settings Panel

To enable auto-launching your program set the ACADLSPASDOC system variable to 1 or simply toggle the option in *Settings / Program options / System*:



It is also recommended that you spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD, as there are settings and options specific to BricsCAD environment.

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:



It is recommended to use the support paths added as in the manner above and/or located one level below the current project folder. It is a good practice to use a "findfile" function to validate the file exists before proceeding with your commands, as in the method below:

```
(if (findfile "custom-code.lsp") (load "custom-code"))
```

This is one of the methods you may use to demand-load your 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 of your **on_start.lsp** file and that function redefines the calling function when it's launched.

15.18.2 Drawing Setup Lisp Routines for BricsCAD

Code sample

The below simple drawing setup routine could be adapted to your needs with minimal changes.

```
(defun C:MYSETUP ( / )                  ;define the command line function "mysetup"
  (setvar "cmdecho" 0)                  ;set display of prompts and input to OFF
  (command "_.audit" "_y")              ;audit current drawing
  (command "_-purge" "_all" "*" "_n")   ;purge all unused styles, etc
  (command "_-layer" "_m" "1" "_c" "1" "" "")  ; create layer "1" color "1"
  (command "_-layer" "_m" "2" "_c" "2" "" "")
  (command "_-layer" "_m" "3" "_c" "3" "" "")
  (command "_-layer" "_m" "4" "_c" "4" "" "")
  (command "_viewres" "_y" "20000")     ;set view resolution to maximum
  (setvar "facetres" 10)                ;set facet resolution to max
  (setvar "filedia" 1)                  ;turn file dialogs on
  (setvar "attdia" 1)                   ;turn attribute dialogs on
  (setvar "attreq" 1)                   ;enable attribute editing on placement
  (setvar "cecolor" "bylayer")          ;set current color bylayer
  (setvar "clayer" "0")                 ;set current layer to "0"
  (setvar "maxsort" 5000)               ;set maximum layer sort to 5,000
  (command "_zoom" "_e" "_zoom" "0.95x") ;zoom extents, then cut a little
  (setvar "cmdecho" 1)                  ;set display of prompts and input to ON
  (prino)
)
```

Each individual line in this drawing setup routine could have been typed in at the command line in BricsCAD. 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 you 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**.

The commands with the preceding dashes (e.g. **-layer** and **-purge**) are used to prevent the command’s dialog boxes from launching.

> ***Nota:*** Always use English command and options names to avoid fails on localized (non-English) BricsCAD versions.

A good practice when opening files from others is to execute an **audit** followed by a **purge** to remove any unused items.

Purge command could be run multiple times to remove nested items. See the below example:

```
(repeat 3 (command "-purge" "all" "*" "n"))
```

> ***Nota:*** **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 it’s easier to read when each line stands alone.

The **setvar** calls for setting a system variable to a specified value, e.g. style settings for dimensions and text, unit settings, etc.

The **maxsort** setting was introduced for two reasons. First, **Maxsort** controls the maximum number of dialog box entries that will be sorted. The second reason is that the “default value” 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!

A more versatile Layer Maker for your drawing setup routine

The following example could be added to your version of mysetup.lsp. By creating a simple text file “Layers.txt” with an editor like Notepad, you 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. You can place your default layer setup text file in the support path and place alternative versions of it in individual project folders.

```
(if (not (findfile "layers.txt"))    ;"not" = looking for no result from findfile
(alert "Can't find layers.txt!")     ;No file? Yell at the user and exit!
(progn                               ;Otherwise, do the stuff in the progn group
(setq layerfile (open (findfile "layers.txt") "r")) ;open layers.txt to "read"
(while (setq layername (read-line layerfile))       ;while reading each line, store the text to layername
(command "-layer" "m" layername "")                 ;make layer with the layername
)                                    ;end the while loop
(close layerfile)                    ;close the layers.txt file
)                                    ;end the progn
)                                    ;end the if
```

16. Modificar Entidades

16.1 Modifying commands overview

Icon	Command	Description
	3DARRAY	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.
	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	ARRAY	Creates 2D and 3D arrays of entities in rectangular or polar patterns, or along a path.
	ARRAYCLASSIC	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	ARRAYPATH	Associatively distributes entity copies evenly along a path into multiple rows, and levels.
	ARRAYPOLAR	Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
	ARRAYRECT	Associatively distributes copies of entities into any number of rows, columns, and levels.
	BREAK	Splits an entity into two entities.
	CHAMFER	Creates chamfers at intersections of linear entities, defined by two lengths, or a length and an angle.
	CHANGE	Changes the properties of entities, through the Command line. This command has largely been superseded by the **Properties** panel.
	COPY	Draws duplicates of the selected entities.
	COPYGUIDED	After selecting the entities, their relation to their reference curves will automatically guide you to inserting the copy in similar locations.
	DIVIDE	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

Icon	Command	Description
	EXPLODE	Reduces complex entities to simpler ones, such as blocks into constituent parts, or polylines into lines and arcs.
	EXTEND	Lengthens lines, arcs, polylines or rays to meet another entity.
	FILLET	Fillets intersections; joins intersecting lines with an arc of any suitable radius.
	FLATTEN	Projects 2D and 3D entities onto the XY-plane of the current view.
	HATCHEDIT	Edits hatches and gradient fills in a dialog box.
	JOIN	Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints.
	LENGTHEN	Changes the length of a line, polyline, freehand entity or arc.
	MANIPULATE	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.
	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	MIRROR	Moves or copies the reflected image of selected entities about a line.
	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	MOVE	Moves entities to another location in the same drawing.
	MOVEGUIDED	After selecting the entities, their relation to their reference curves will automatically guide you to moving the entities in a similar location.
	OFFSET	Creates a parallel or offset copy of linear entities.
	OVERKILL	Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; optionally moves duplicates to a dedicated layer. Combines duplicate blocks; optionally purges duplicate block definitions.

Icon	Command	Description
	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	PROPERTIES	Opens the BricsCAD **Properties** panel.
	ROTATE	Rotates entities around a specified point.
	ROTATE3D	Rotates entities around an axis.
	SCALE	Changes the scale of existing entities, either enlarging them or reducing them proportionately in X, Y, and Z directions.
	SIMPLIFY	Reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Simplified entities are easier to manipulate and can significantly reduce file size.
	SPLINEDIT	Edits a two-dimensional or three-dimensional spline.
	STRETCH	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	TRIM	Erases the portions of selected entities that cross a specified boundary.

16.2 Entity modification settings

Entity modification settings are found in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity modification**.

16.3 Match properties

The MATCHPROP command (short for "match properties") assigns the properties of one entity to other entities.



16.3.1 Copying properties between entities

1 Launch the MATCHPROP command.

2 **You are prompted**: Select entity to copy properties from:

3 Select the source entity.

4 **You are prompted**: 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/selection options (?)]:

5 (option) Choose the **Settings** option, then press Enter.

Adjust the settings, then click the **OK** button.

6 Select the target entities.

7 **You are prompted**: Select entities to copy properties to or [Settings/selection options (?)]:

8 Keep selecting entities or right-click to stop.

16.3.2 Copying properties between entities with BIM data

Whether the MATCHPROP command copies BIM data depends on the value of the BIMMATCHPROP system variable.

- **BimMatchProp = 0**: Match standard properties

  Only standard properties will be copied, according to the currently selected options (see above).

- **BimMatchProp = 1**: Match standard and BIM properties

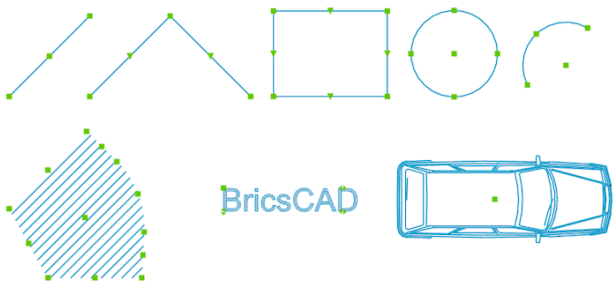
  BIM data, including IFC properties and user defined properties are copied, except:

  - BIM Type*
  - Composition/Profile
  - GUID
  - Quantities

* When using MATCHPROP between a BIM classified entity and an entity without any BIM data, the BIM type and the corresponding BIM properties will be copied along to the unclassified entity, except Composition/Profile, GUID and Quantities.

16.4 Grips editing

Grips are displayed as small colored squares at key-points on selected entities.



Whether grips are displayed or not and the appearance of grips are controlled through a series of grips settings in the **Settings** dialog box under **Drawing** > **Drafting** > **Display/Viewing** > **Grips**.

In complex drawings, using too many grips may clutter the display. The GRIPOBJECTLIMIT system variable allows you to suppress the display of grips if the number of selected entities exceeds a certain limit. If set to zero, all grips are displayed.

Grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

When the GEOMRELATIONS system variable is set, the geometric relation (perpendicular or tangent) between the entity being edited and other lines, polylines, arcs or circles is indicated (see Geometric Relations).

To select a grip, move the mouse over it, then click. Selected grips are displayed in a different color, which is defined by the HOTGRIP system variable. A selected grip is called a hot grip. By default, hot grips are displayed in red.

The **midpoint** grip on lines and infinite lines, the **startpoint** grip on rays, the **center** grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the **insertion point** grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

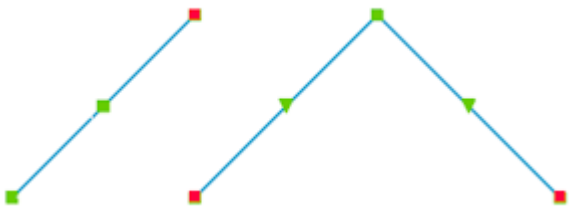
Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

16.4.1 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 is displayed in red.

3 Keep the Shift key pressed, then click more grips. All selected grips are displayed in red.



4 Release the Shift key, then click the hot grip you want to use as the base point. The hot grips move simultaneously.

16.4.2 Grip Edit Command Options

When you start dragging a selected grip, BricsCAD displays a prompt in the command line:

Enter new point [Base point/Copy/Undo/eXit]:

You can choose an option using the keyboard, or in the **Prompt** menu.

Keyboard	Prompt Menu	Description
B	Base Point	You are prompted 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.

Keyboard	Prompt Menu	Description
R	Reference	(Applies to Grip Rotate only) Allows you to specify a reference angle. Type the angle in the Command line or specify two points.
X	Exit	Aborts the grip edit command.

16.4.3 The grips editing context menu

The grips editing commands can be chosen by repeatedly hitting the space bar or the Enter key, or in a right-click context menu.

The command options are: stretch, move, mirror, rotate, scale.

The stretch command option is selected by default.

16.4.4 Stretching entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point.

4 All selected grips move simultaneously.

5 **You are prompted**: Enter new point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to define the displacement point:

- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

16.4.5 Moving entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 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.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

16.4.6 Copying entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 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.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following:
- Choose **Copy** in the context menu.
- Type **C**, then press **Enter**.

8 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

9 Repeat the previous step to create more copies or press the Space bar or Enter to stop.

16.4.7 Mirroring entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the mirroring.

4 The **Grip Stretch** command is launched.

5 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.

You are prompted: Enter second point: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Specify a second point to define the mirror axis.

9 The selection is mirrored.

16.4.8 Rotating entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the rotation.

4 The **Grip Stretch** command is launched.

5 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.

You are prompted: Enter rotation angle: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to rotate the selection:

- Specify a point to define the rotation angle.
- Type the rotation angle in the Command line, then press Enter.

16.4.9 Scaling entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the scaling.

4 The **Grip Stretch** command is launched.

5 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.

You are prompted: Enter scale factor: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

8 Specify a point.

9 Type the scale factor in the Command line, then press Enter to scale the selection.

16.5 Selection methods

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, you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

The SELECTIONPREVIEW and PREVIEWEFFECT system variables facilitate the selection procedure.

The SELECTIONPREVIEWFILTER system variable allows you to exclude entities from being selected: entities on locked layers, entities in Xrefs, tables, MTexts, hatches, entities in group.

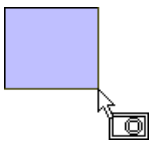
16.5.1 Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click entities.
- Selected entity highlight and grips are shown.
- Area selection:
  - Click to define the first corner of a selection area.
  - Move the mouse to the left to define a **Crossing area**:

  - All entities which overlap or are completely inside the rectangle, are added to the selection set.
  - By default, the selection area is displayed in green with a dashed outline.
  - Click to define the first corner of a selection area.
  - Move the mouse to the right to define a **Window area**:

  - All entities which are completely inside, are added to the selection set.
  - By default the selection area is displayed in blue with a continuous outline.
- When working with 3D solids and surfaces, pressing the Ctrl key during window selection allows you to select faces or edges instead of the entire solid or surface. The icon next to the cursor indicates the selection mode:
  - Solids (default)

- Faces
- Edges

Depending on the value of the TIPS system variable, the Hotkey Assistant widget indicates the current selection mode:



- Press an hold the **Shift** key, then use one of the above methods to **remove** entities from the selection set.

Nota:

- The Selection Area system variables control the display of the area selection effects.
- When the PICKADD system variable is **Off**, you cannot add entities to a selection set.
- Press and hold the left mouse button to start window selection when the entity under the cursor highlights. This method is especially helpful for window selection in rendered 3D models.

16.5.2 Composing a selection set afterwards (post-pick)

If you launch a modification command when no selection set is active, **you are prompted** to select entities first. 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.

16.5.3 Selecting overlapping entities

When entities lie on top of each or are obscured by other entities, it is difficult or sometimes impossible to select such entities..

Cycling through the overlapping entities, highlighting each entity one by one, helps you to select such entities.

Make sure SELECTIONPREVIEW is ON and/or the Quad cursor menu is active.

1 Place the cursor over the overlapping entities.

1 One of the entities highlights.

2 Repeatedly hit the Tab key.

3 The overlapping entities highlight one by one.

4 Click when the desired entity highlights.

5 Repeat the previous steps to select more entities.

Nota:

- 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.

16.5.4 Selecting subentities

By default the entire entity is selected. This method allows you to select subentities of compound entities, such as polylines, 3D polylines, regions, 3D solids, etc..

1 Place the cursor over the subentity.

2 When the entire entity highlights, hold down the Ctrl key.

3 Click when the subentity is highlighted. The properties of the selected subentity show in the **Properties** panel.

4 (option) Repeat the previous steps to select more subentities.

The following subentities can be selected: polyline segments, vertices and segments of 3D polylines, vertices, edges and faces of regions, surfaces, meshes and 3D solids.

16.6 Quick select

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.

16.6.1 To create a selection set

1 Click the **Quick Select** switch button () on the **Properties** panel.

The **Properties** panel switches to **Quick Select** mode.

If a selection set exists then the **Apply to current selection set** option is checked.

The **Selection list** button indicates the number of entities in the drawing, or the number of selected entities.

2 (option) Click the **Selection list** button, to see a list of all entity types in the current drawing or current selection set.

A list of all entities in the drawing and the number of entities of each type is displayed.

3 Select an entity type in the list, e.g. **Line**.

All Line properties show in the **Properties** panel.

4 Select a property you want to use as a filter, e.g. **Color**.

5 Specify a color.

6 By default all entities of this color will be selected.

7 Optionally, specify a different selection operator.

Click the = sign, then select an operator in the list.



8 Do one of the following:

- Click the **Add to new selection set** button ( ).

  The entities that match the selected criteria are selected.

- 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.

9 (option) Repeat the above procedure to further refine the selection set.

16.7 Draw order

When new entities overlap existing entities, they display and print on top of the previously drawn entities. The DRAWORDER command 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.

1 Launch the DRAWORDER command.

2 **You are prompted**: Select entities to change the draw order:

3 Select the entities, then right-click or press enter.

4 **You are prompted**: Change draw order: Above/Under/Clear/Front/<Back>:

5 Choose an option.

6 Depending on your choice:

- If you have chosen **Above** or **Under**, the Command line 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.

16.7.1 Using Draw Order tools

Draw Order tools are available in the **Tools** menu and the **Draw Order** toolbar.

Icon	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)
	Send Under	Place the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back (HATCHTOBACK command).
	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 line or in the prompt menu. The default option is **Both** (TEXTTOFRONT command).

16.8 The manipulator

The manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities. Holding down the Ctrl key when starting the action keeps the original entities, thus creating copies. 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 to a dynamic UCS. When multiple entities are selected simultaneously, the manipulator is displayed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can be moved to any desired location or orientation using its handles and the context menu options.

Depending on the operation type and if dynamic dimensions are active, a dynamic entry field is available to set the distance, angle or scale factor.

16.8.1 Adaptive snap step

The snap step size automatically adapts to the current screen zoom factor if SNAPTYPE= 2. The ADAPTIVEGRIDSTEPSIZE system variable controls the smallest available step size. The adaptive step size is used to dynamically adjust the step size in the Manipulator Ruler, which enables you to quickly modify entities using precise values, without having to key in numbers.

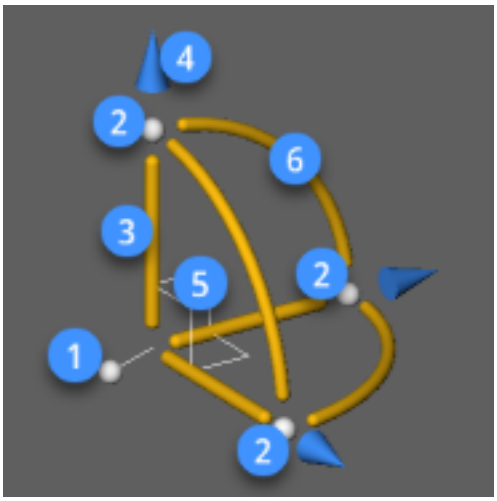
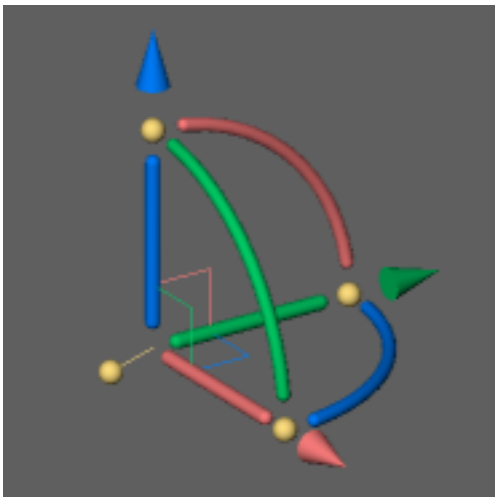
16.8.2 The manipulator settings

The manipulator settings can be found in the **Settings** dialog box under **Program Options** > **Manipulator**.

When the Manipulator is displayed, select **Manipulator Settings** in the right-click context menu.

16.8.3 The manipulator layout

Check the **Classic Color** option in the manipulator context menu for the classic layout. Uncheck the option for the default layout.

	
Default	Classic

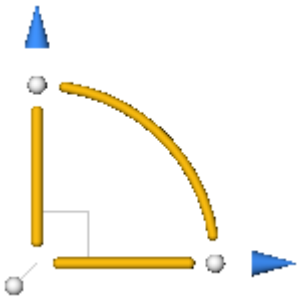
4 handles

- **1**: manipulator 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.



16.8.4 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	Manipulator - Orientation
Line	Near endpoint	At endpoint	X-axis through the other endpoint
Line	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
Polyline 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

Entity Type	Selection Point	Manipulator - Anchor Point	Manipulator - Orientation
Region	On an 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 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.

16.8.5 Displaying the manipulator

The value of the MANIPULATOR system variable controls when the MANIPULATOR appears automatically:

- 0: Never
- 1: When entities are selected.
- 2: When the left mouse button is hold down longer when selecting an entity.

In order to display the Manipulator when multiple entities are selected:

- Select the entities one by one or use window selection, then long-click to select the last entity.
- Set the value of the MANIPULATOR system variable to 1.
- Compose the selection set, then launch the MANIPULATE command.

16.8.6 Moving the manipulator

Do one of the following:

- Long-click anywhere on the manipulator and move it to a new location.
- MANIPULATORHANDLE = 0: Click the manipulator handle (1) and move it to a new location*.
- The anchor handle stands out to avoid overlap with entity grips.
- MANIPULATORHANDLE = 1: Long-click the manipulator handle (1) and move it to a new location*.
- Place the cursor over the manipulator, then right-click and choose **Move** in the context menu and move it to a new location*.

* Specify the new location using entity snaps.

16.8.7 Rotating the manipulator

Select 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.

Press the Shift key to rotate the Manipulator 90 degrees about its normal axis (Z-axis).

16.8.8 Reorienting the manipulator

1 Place the cursor over the manipulator, then right-click and choose **Reorient Manipulator** in the context menu.

You are prompted: Specify origin of manipulator:

2 Pick a point.

3 The anchor point of the manipulator is placed at the specified point.

You are prompted: Point on X-axis or <Accept>:

4 Pick a point.

5 The manipulator rotates about its X-axis.

You are prompted: Point on the XY-plane with positive Y value or <Accept>:

6 Pick a point to define new position of the manipulator.

16.8.9 Aligning 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 to become 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 to become parallel with the corresponding axes of the UCS.

16.8.10 Aligning the manipulator with the face of a solid

Dynamic UCS (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.

3 The XY-plane of the manipulator aligns with the solid face under the cursor.

4 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.

16.9 Moving entities

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 (short for "paste clipboard") pastes entities from the Clipboard into the current drawing.

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 of the Grips Editing commands allows you to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

16.9.1 Moving entities in a drawing



The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

1 Launch the MOVE command.

1 **You are prompted**: Select entities to move [selection options (?)]:

2 Select the entities, then right-click or press Enter.

3 The selection set is now attached to the cursor.

You are prompted: Enter base point [Displacement] <Displacement>:

4 Specify a point.

5 **You are prompted**: Enter second point or <Use base point as displacement>:

6 Specify a point.

7 The selection set is moved.

16.9.2 Moving entities between drawings

1 In the source drawing, select the entities you want to move.

2 Do one of the following:

- Right-click, then choose **Cut** in the context menu.

  Hold down the right-mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + 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:

- Choose **Paste** in the **Edit** menu.
- Press **Ctrl + V**.

The bottom left corner of the bounding rectangle of the selection is attached to the cursor.

5 Do one of the following:
- Specify a point.
- Press to paste the selection set using the coordinates of the source drawing.

Nota: Launch the UNDO command in the source drawing bring back the moved entities.

16.9.3 Moving a selection as a block

1 Select the entities you want to move.

2 In the source drawing, select the entities you want to move.

3 Do one of the following:
- Right-click, then choose **Cut** in the context menu.

  Hold down the right mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + X**.

  The entities are deleted in the source drawing and copied to the Clipboard.

4 The entities are deleted in the drawing and copied to the Clipboard.

5 (option) Switch to the target drawing.

6 Choose **Paste as a Block** in the **Edit** menu or press **Ctrl + Shift + V**.

  The bottom left corner of the bounding rectangle of the selection is attached to the crosshairs in the target drawing.

7 Specify the insertion point.

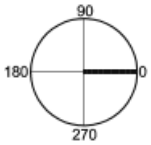
8 The selection is pasted as a block, using a default name.

9 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

16.10 Rotating entities

The ROTATE command rotates entities at a specified point along 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 (East) and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction, you can enter a negative value.



Nota:

- 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.



16.10.1 General procedure

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Do one of the following:

- Specify a point.
- Type a value and press Enter.

The selection set is rotated.

16.10.2 Rotating a selection set relative to a base angle

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Choose the **Base angle** option.

You are prompted: Base angle <0>:

9 Do one of the following:

- Type a value and press Enter.
- Click two points to specify the base angle.

10 **You are prompted**: New angle <30.84>:

11 Do one of the following:

- Type a value and press Enter.

- Click a point.

The selection is rotated.

16.10.3 Rotating in 3D



1 Launch the ROTATE3D command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities, or press Enter to stop selecting.

5 **You are prompted**: Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis/2points] <2points>:

6 Press Enter to accept the 2points method to select the rotation axis.

7 **You are prompted**: Specify the first rotation axis point:

8 Specify a point.

9 **You are prompted**: Specify the second rotation axis point:

10 Specify a point.

11 **You are prompted**: Rotation angle or [Base angle/Copy]:45

12 Type a value and press Enter.

13 The selection is rotated.

16.11 Mirroring entities

The MIRROR command creates a mirrored copy of a selection set. The selection is mirrored along a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The MIRROR3D command creates a mirrored copy of selected entities in three-dimensional space. In this case the selection is mirrored along a mirror plane, 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.

Nota: Whether text is mirrored or not by the MIRROR command is controlled by the MIRRTEXT system variable.

16.11.1 Mirroring entities along a line



1 Launch the MIRROR command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Start of mirror line:

6 Specify a point.

7 The selection set rotates dynamically.

You are prompted: End of mirror line:

8 Specify a point.

9 The selection is mirrored.

You are prompted: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

10 Decide whether or not to keep the original entities.

16.11.2 Mirroring entities along a plane



1 Launch the MIRROR3D command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>:

6 Press Enter to accept the 3 points method to define the mirror plane.

7 **You are prompted**: First point on plane:

8 Specify a point.

9 **You are prompted**: Second point on plane:

10 Specify a point.

11 **You are prompted**: Third point on plane:

12 Specify a point.

13 **You are prompted**: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

14 Decide whether or not to keep the original entities.

16.12 Aligning entities

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 also scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.



16.12.1 Aligning in 2D

1 Launch the ALIGN command.

2 **You are prompted**: Select entities:

3 Select the entities, then right-click or press Enter.

4 **You are prompted**: Specify first source point:

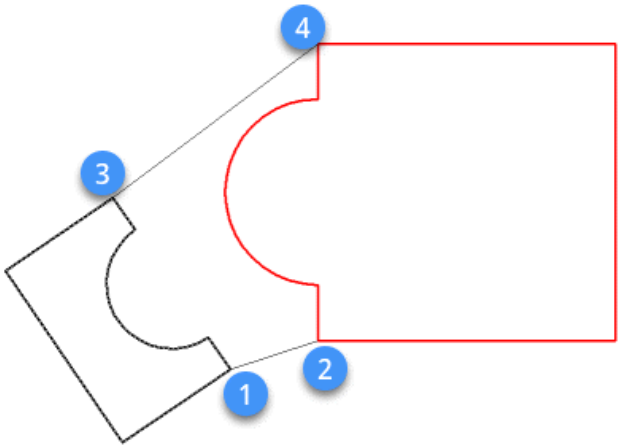
5 Snap to the first source point in the selection set (1).

6 **You are prompted**: Specify first destination point:

7 Snap to the first destination point on the reference entity (2).

8 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



9 Snap to the second source point in the selection set (3).

10 **You are prompted**: Specify second destination point:

11 Snap to the second destination point on the reference entity (4).

12 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

13 Right-click to use the ALIGN command in 2D.

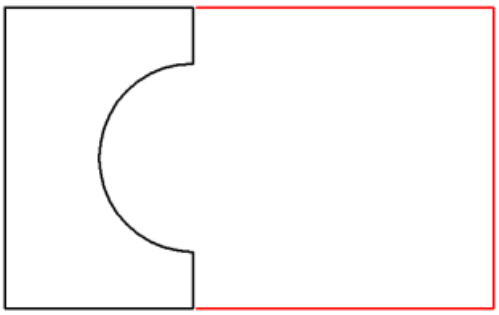
14 **You are prompted**: Scale objects based on alignment points [Yes/No] <No>:

15 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.
- Scaling adjusts the distance between the source points to the distance between the target points.

16



Result of the Align procedure with scaled selection set.

16.12.2 Aligning in 3D

1 Launch the ALIGN command.

1 **You are prompted**: Select entities:

2 Select the entities, then right-click or press Enter.

3 **You are prompted**: Specify first source point:

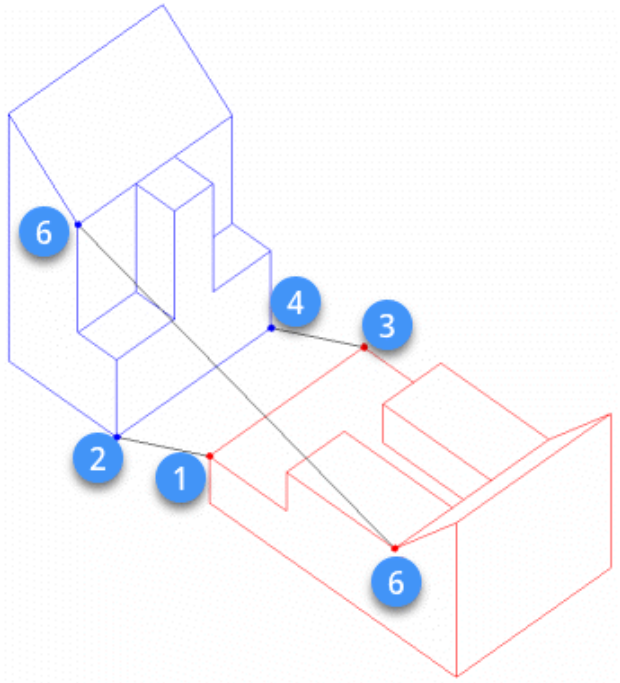
4 Snap to the first source point in the selection set (1).

5 **You are prompted**: Specify first destination point:

6 Snap to the first destination point on the reference entity (2).

7 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



8 Snap to the second source point in the selection set (3).

9 **You are prompted**: Specify second destination point:

10 Snap to the second destination point on the reference entity (4).

11 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

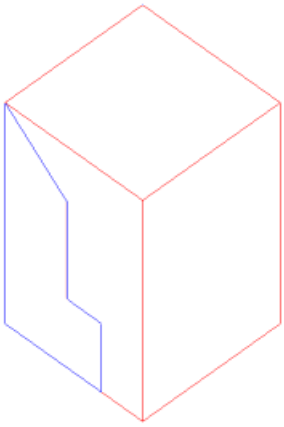
12 Snap to the third source point in the selection set (5).

13 **You are prompted**: Specify third destination point:

14 Snap to the third destination point on the reference entity (6).

15 A witness line is drawn between the source point and the target point.

The entity is aligned.



16.13 Changing entities

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 panel is much more versatile.



16.13.1 Changing text entities

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the text entities, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a new insertion point or press Enter to keep the original.

5 **You are prompted**: New text style Standard:

6 Type the name of a new text style or press Enter to keep the current style.

7 The text entity scales dynamically.

You are prompted: New height <current height>:

8 Do one of the following:

- Click to define the new text height.
- Type a value to specify the new text height.
- Press Enter to keep the current text height.

The text entity rotates dynamically.

You are prompted: New rotation angle <0>:

9 Do one of the following:

- Specify a point
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

You are prompted: New text <current text>:

10 Type a new text or press Enter to keep the current text.

11 When multiple text entities are selected, the procedure restarts for each entity.

16.13.2 Relocating blocks

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the blocks, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a point.

5 The block rotates dynamically.

You are prompted: New rotation angle <current angle>:

6 Do one of the following:
- Specify a point.
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

7 When multiple blocks are selected, the procedure restarts for each entity.

16.14 Copying entities

The COPY command duplicates entities within the current drawing using. 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.

The COPYCLIP command copies entities to the clipboard, which you can then paste in the current drawing or in another drawing using the PASTECLIP or the PASTEBLOCK commands.

Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.



16.14.1 General procedure

1 Launch the COPY command.

1 **You are prompted**: Select entities to copy [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

4 **You are prompted**: Select entities to copy [selection options (?)]:

Current copy mode: Multiple

You are prompted: Enter base point [Displacement/mOde] <Displacement>:

5 Specify a point.

6 The copied selection moves dynamically.

You are prompted: Enter second point or [Array] <Use base point as displacement>:

7 Specify a point.

8 A first copy of the selection is created.

A second copy moves dynamically.

You are prompted: Enter second point [Undo/Array/Repeat/Exit]:

9 Do one of the following:
- Keep specifying points to create multiple copies.
- Press Enter to stop.

16.14.2 Using the Repeat option

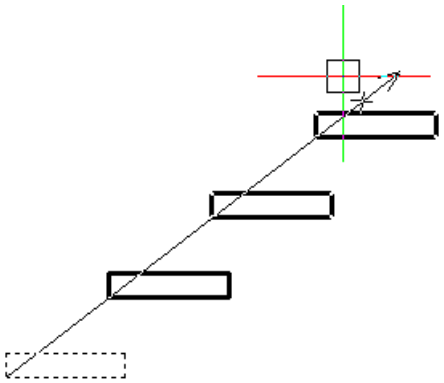
1 Execute steps 1-5 of the general procedure (see above).

1 **You are prompted**: Enter second point [Undo/Array/Repeat/Exit]:

2 Choose the **Repeat** option.

Multiple copies at regular intervals are created when you move the cursor.

You are prompted: Select end point (<ENTER> to key in number of repeats):



3 Do one of the following:

- Click when the desired number of copies is displayed.
- Press Enter.
- **You are prompted**: Enter number of repeats <1>:

  Type the desired number of copies, then press Enter.

16.14.3 Using the Array option

1 Execute steps 1-4 of the general procedure (see above).

1 **You are prompted**: Enter second point or [Array] <Use base point as displacement>:

2 Choose the **Array** option.

You are prompted: Enter number of items to array:

3 Specify the number of items.

4 The array is dynamically displayed.

You are prompted: Enter second point or [Fit]:

5 Do one of the following:

- Specify a point to define the distance between each item in the array.
- Choose the **Fit** option.

  You are prompted: Enter second point or [Array]:

  Specify a point to define the distance between the first and the last item in the array.

6 Do one of the following:

- Keep creating copies.
- Press Enter to stop.

16.14.4 Copying a selection as a block

1 Select the entities you want to copy as a block.

2 Launch the COPYBASE command.

3 **You are prompted**: Select base point:

4 Specify a point.

5 xx selected

6 (option) Switch to another drawing.

7 Launch the PASTEBLOCK command.

8 **You are prompted**: Select insertion point:

9 Specify a point.

10 The selection is pasted as a block.

11 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

16.14.5 Copying entities between drawings

Entities can be copied between drawings using the following methods:

- COPYCLIP and PASTECLIP*: to copy the selection using a default base point.
- COPYCLIP and PASTEORIG: to copy the selection using the coordinates of the source drawing.
- COPYBASE and PASTECLIP*: to copy the selection using a user-defined base point.
- COPYBASE and PASTEBLOCK: to copy the selection as a block in the target drawing.

* Press Enter at the 'Specify insertion point' prompt to paste the selection using the coordinates of the source drawing (= equivalent of the PASTEORIG command).

16.15 Making parallel copies

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 set by the SELECTIONMODES system variable, the OFFSET command accepts:

- an edge of a 3D solid if the **Select Edges** option is set.
- the outline of a 3D solid face if the **Select Faces** option is set.
- enclosed boundaries if the **Select Detected Boundaries** option is set.

Nota: The OFFSETGAPTYPE settings variable controls how potential gaps between segments are treated when closed polylines are offset.

16.15.1 Making a parallel copy at a specified distance

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:

- Press Enter to accept the current distance.
- Type a new distance in the Command line.
- Define a new distance by specifying two points.

You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 A preview of the parallel copy is displayed depending on the current cursor position with respect to the selected entity.

You are prompted: Select side for parallel copy or [Both sides/Multiple]:

5 Click a point.

6 The parallel copy is created.

If OFFSETERASE = ON, the original entity is deleted.

7 **You are prompted**: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

8 Do one of the following:

- Select another entity.
- Press Enter to stop.

16.15.2 Making a parallel copy through a point

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:

- Press Enter to accept the default **Through point** option (= when last used).
- Choose the **Through point** option.

You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 The parallel copy is dynamically displayed.

5 Click a point to create the parallel copy.

6 If OFFSETERASE = ON, the original entity is deleted.

You are prompted: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

7 Do one of the following:

- Select another entity.
- Press Enter to stop.

16.16 Arraying entities

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 around 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 array (ARRAYRECT command), a polar array (ARRAYPOLAR command) or a path array (ARRAYPATH command) array.

The ARRAYEDIT command allows you to edit associative arrays and their source entities.

16.16.1 Creating a static rectangular array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Rectangular**.

3 Click the **Select Entities** button.

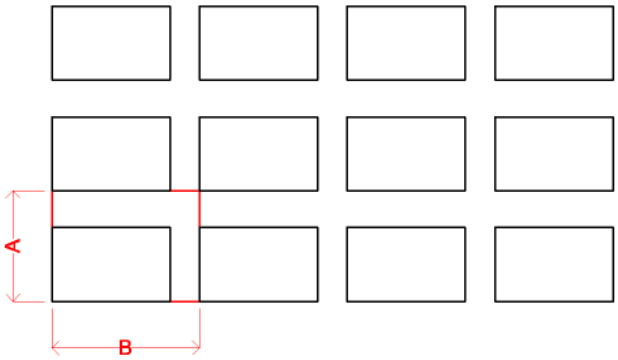
The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

5 The **Array** dialog box reopens.

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 a point in the drawing.



Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

7 Click the **OK** button to create the array.

Nota: Positive values in step 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

16.16.2 Creating a static polar array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

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, then right-click or press Enter.

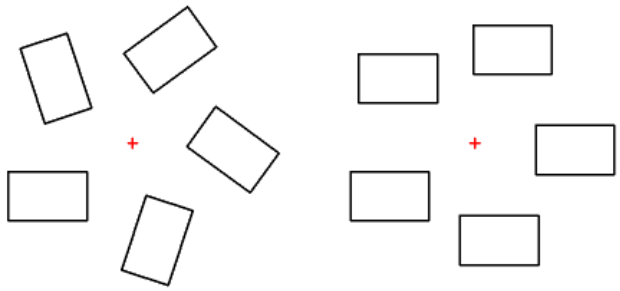
5 The **Array** dialog box reopens.

6 Choose a **Calculate** option: **Angle between**, **Angle to fill** or **Number of items**.

7 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

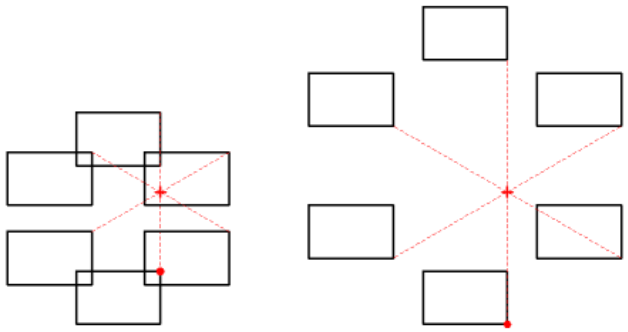
The **Array** dialog box closes temporarily to let you pick a point in the drawing.

8 (option) Check the **Rotate items as copied** option.



Entities rotated (left) or not (right) about the center point of the array.

9 (option) Define an **Entity base point**.



Polar array with base point (dot) and center point (cross).

10 Click the **OK** button to create the array.

16.16.3 Creating a static 3D rectangular array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Rectangular** option.

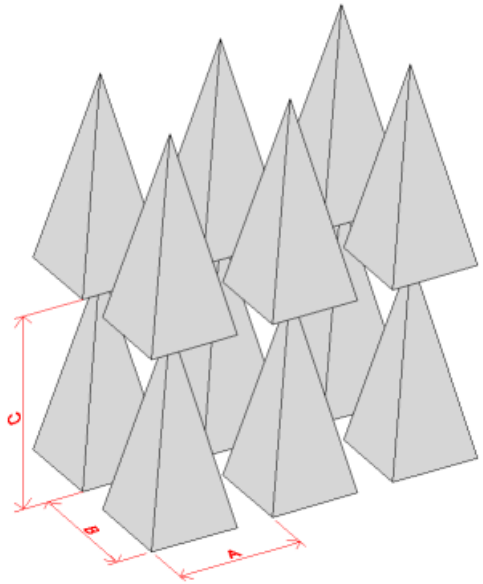
You are prompted: Number of rows in the array <1>:

7 Specify the number or rows.

8 **You are prompted**: Number of columns <1>:

9 Specify the number of columns.

10 **You are prompted**: Number of levels <1>:



Rectangular 3D Array:

row spacing A, column spacing B, level spacing C

11 Specify the number of levels.

12 **You are prompted**: Vertical distance between rows:

13 Type a value or click two points.

14 **You are prompted**: Horizontal distance between columns:

15 Type a value or click two points.

16 **You are prompted**: Depth between levels:

Type a value or click two points.

The array is created.

Nota: 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.

16.16.4 Creating a static 3D polar array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Polar** option.

You are prompted: Number of items to array or [enter to specify Angle between items] <enter to specify Angle between items>:

7 Specify the number of items.

8 **You are prompted**: Angle to fill (+ for ccw, - for cw <360>):

9 Type a value or press Enter to accept the default.

10 **You are prompted**: Rotate entities around the array? [Yes/No] <Yes>:

11 Choose **Yes** or **No**, or press Enter to accept the default.

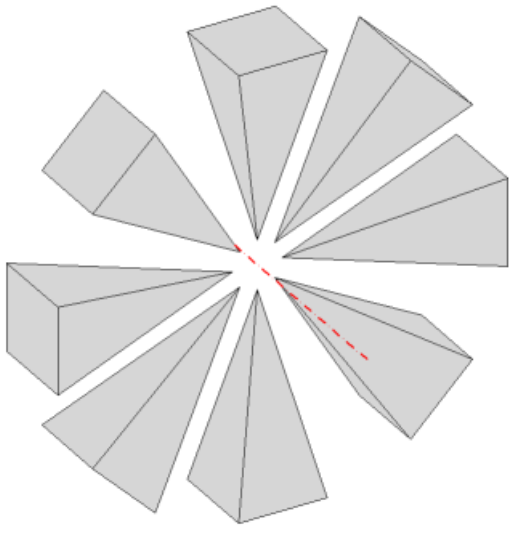
You are prompted: Center point of array:

12 Specify a point.

13 **You are prompted**: Specify second point along central axis of array:

14 Specify a point.

15 The array is created.



Polar 3D array around a horizontal axis (red).

16.17 Extending entities

The EXTEND command extends entities to a boundary, which is defined by one or more other entities.

If the EDGEMODE system variable is **On**, entities extend 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 PROJMODE system variable 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.

16.17.1 Extending entities

1 Launch the EXTEND command.

1 **You are prompted**: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

2 Select a boundary entity.

3 Entities in set: 1

You are prompted: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

4 Select more boundary entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/ Projection/eRase]:

6 Select an entity:

7 The entity extends to the boundary entity.

You are prompted: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/ Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Nota:

- Entities which are selected when you launch the EXTEND command will be used as boundary entities.
- Hold down the Shift key when you select an entity to trim the line by the boundary entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the EXTEND command is aborted.

16.18 Trimming entities

The TRIM command clips or trims entities by cutting entities.

If the EDGEMODE system variable is **On**, entities extend 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 PROJMODE system variable 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 pane
- 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.



16.18.1 Trimming entities

1 Launch the TRIM command.

1 **You are prompted**: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

2 Select a cutting entity.

3 Entities in set: 1

You are prompted: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

4 Select more cutting entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/ Projection/eRase]:

6 Select an entity.

7 The entity is trimmed by the cutting entity.

You are prompted: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/ Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Nota:

- Entities which are selected when you launch the TRIM command will be used as cutting entities.
- Hold down the Shift key when you select an entity to extend the line to the cutting entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.

16.19 Lengthening entities

The LENGTHEN command changes the length of lines, open polylines and arcs dynamically, incrementally or relatively. You can also modify the included angle of arcs.

16.19.1 Changing the length dynamically

1 Launch the LENGTHEN command.

1 **You are prompted**: Edit length: Select entity to list length or [DYnamic/Increment/Percent/Total]:

2 Choose the **Dynamic** option.

You are prompted: Select entity to change or [edit Mode]:

3 Click the entity near the endpoint you want to lengthen.

4 The entity is lengthened (shortened) dynamically.

You are prompted: New end point for line:

5 Specify a point to define the new length.

6 **You are prompted**: Select entity to change or [edit Mode/Undo]:

7 Do one of the following:

- Edit another entity.
- Press Enter to stop.

Nota: The **Geometry** properties in the **Properties** panel allow you to edit the total length of lines and arcs and the start and end angle of arcs.

16.20 Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities, e.g. whether grips are displayed or not, is controlled by a number of settings.

16.20.1 Selection settings

Selection settings sit in the **Selection** settings group of the **Program Options** settings category in the **Settings** dialog box.

Name	System Variable	Description
Highlight	HIGHLIGHT	Determines whether 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.

Name	System Variable	Description
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 you 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 you to first compose a selection set, then launch a modification command. **OFF**: You must first start the command, then compose the select entities.

Name	System Variable	Description
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 command.
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 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).

Name	System Variable	Description
Selection preview filter	PREVIEWFILTER	Controls which entity types are excluded from selection previewing. The options are: Entities on locked layers Entities 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. 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 it is possible to select 3D solids, if OFF it is possible 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.
Constraint Bar Display	CONSTRAINTBARDISPLAY	Controls the visibility of a geometric constraint at its creation or selection.

Name	System Variable	Description
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 is displayed.
Preview selection delay	PREVIEWDELAY	Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor.

16.20.2 Grips settings

If the Grips system variable is different from zero, grips are displayed on selected entities.

Grips settings sit under **Display/Viewing** in the **Drafting** settings group of the **Drawing** settings category in the **Settings** dialog.

Grips can be used to manipulate entities (see Grips Editing).

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), all grips are displayed.

Name	System Variable	Description
Grip tips	GRIPTIPS	(Not supported yet) Determines whether grip tips are displayed when the cursor hovers over 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.

16.21 Stretching entities

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.



16.21.1 Stretching entities

1 Launch the STRETCH command.

1 **You are prompted**: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/ selection options (?)]:

2 Use a crossing window (or crossing polygon) to specify the points that need to be moved.

3 Specify the bottom right corner of the crossing window.

A green rectangle is dynamically displayed.

You are prompted: Opposite Corner:

4 Specify the top left corner of the crossing window.

5 Entities in set: xx.

You are prompted: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/ selection options (?)]:

6 Do one of the following:

- Use a crossing window (or crossing polygon) to specify the points that need to be added or removed from the selection.
- Right-click or press Enter to confirm the current selection.

You are prompted: Base point of displacement:

7 Specify a point.

8 The selected points move with the cursor.

You are prompted: Second point of displacement:

9 Specify a point to confirm the stretch distance.

16.22 Scaling entities

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 line. 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.

16.22.1 Scaling a selection set

1 Launch the SCALE command.

1 **You are prompted**: Select entities to scale [selection options (?)]:

2 Select entities.

3 Entities in set: xx

You are prompted: Select entities to scale [selection options (?)]:

4 Select more entities or press Enter to continue.

5 **You are prompted**: Base point:

6 Specify a point.

7 The base point is not moved in the scaling.

You are prompted: Scale factor or [Reference/Copy] <current scale*>:

8 Type a scale factor or press Enter to accept the previously used value.

* The previously used scale factor in a drawing is remembered.

16.22.2 Scaling a selection set using a reference scale

1 Repeat steps 1 through 4 of the procedure above.

1 **You are prompted**: Scale factor or [Reference/Copy] <current scale*>:

2 Choose the **Reference** option.

You are prompted: Reference scale <1>:

3 Click the base point again (see step 3 in the previous procedure) (1).

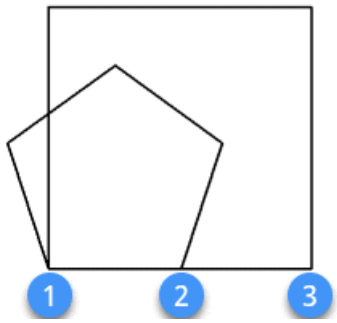
4 **You are prompted**: Second point:

5 Specify a point. (2)

6 The distance between the base point and the second point will be used as the reference scale.

The selection scales dynamically.

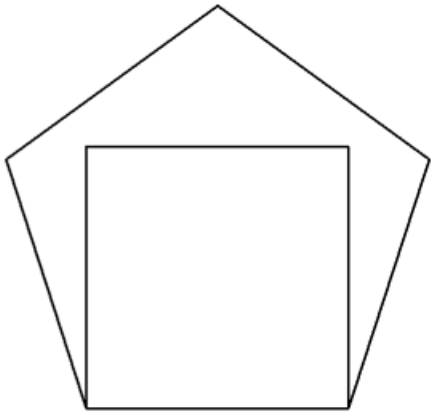
You are prompted: New scale factor or [Points] <2>:



7 Do one of the following:

- Type a value.
- The distance between the reference points is set to the specified value.

  The size of the selection set is scaled accordingly.
- Specify a point to position the second reference point (see step 4) (3).



16.23 Breaking entities

The BREAK command removes 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.

16.23.1 To break an entity

1 Launch the BREAK command.

1 **You are prompted**: Select entity to break [selection options (?)]:

2 Select the entity.

3 **You are prompted**: Select second breakpoint or [First break point/same as first point (@)]:

4 Select the **First** break point option.

You are prompted: First break point:

5 Specify a point.

6 **You are prompted**: Second break point:

7 Specify a point.

8 The part of the entity between the two break points is removed.

16.24 Joining entities

The JOIN command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

When you join colinear lines, the farthest endpoints remain at their existing locations. BricsCAD draws a new line between these points.

The resulting entity type depends on the types of the input entities and their coplanarity.



16.24.1 General procedure

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select source entities, then right-click to stop selecting.

3 BricsCAD reports the result:

4 37 entities joined into 10 polylines, 4 entities discarded from the operation

16.24.2 Joining two arcs

Coplanar arcs, sharing midpoint and radius (= lie on the same circle), are joined counterclockwise, therefore the result depends on the selecting order.

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select the first arc.

3 **You are prompted**: Select entities to join [selection options (?)]:

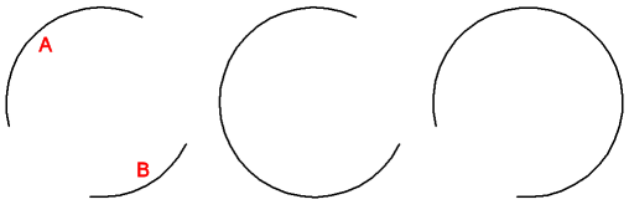
4 Select the second arc.

5 Entities in set: 1

6 **You are prompted**: Select entities to join [selection options (?)]:

7 Right-click.

8 BricsCAD reports: 2 arcs joined into 1 arc.



start situation (left) and result when selecting A first (middle) or B first (right).

16.25 Chamfering entities

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 current chamfer method, distances and angle are saved through a series of system variables:

- CHAMMODE: sets the chamfer method
- CHAMFERA: defines the first distance
- CHAMFERB: defines the second distance
- CHAMFERC: defines the chamfer length
- CHAMFERD: defines the chamfer angle
- TRIMMODE: defines whether or not to keep the original length of the chamfered entities or to trim or lengthen them to the endpoints of the chamfer line.

You can set the values in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity Modification** > **Chamfer/Fillet**. The **Settings...** option of the CHAMFER command allows you to edit the variables in the CHAMFER command procedure.

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 all vertices of the polyline. When chamfering a polyline, the chamfer line is joined with the polyline segments if TRIMMODE=ON.



16.25.1 Chamfering using the distance-distance method

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Directional/Select second entity (select with pressed SHIFT to make corner)

[Directional]:

5 Select the second entity.

6 The chamfer is created.

16.25.2 Chamfering using the length-angle method

1 Launch the CHAMFER command.

1 Chamfer (length=20, angle=45) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

5 Select the second entity.

6 The chamfer is created.

16.25.3 Chamfering all vertices of a polyline

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Choose the **Polyline** option.

You are prompted: Select 2D polyline to chamfer:

4 Select the polyline.

5 The polyline is chamfered.

Nota: When the chamfer method is **distance-angle**, the direction of the polyline defines which is the first entity of a vertex. Use the **Reverse direction** option of the PEDIT command if necessary to get the desired result.

16.25.4 Removing a chamfer

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 Select the first entity.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Hold down the Shift key, then select the second entity.

5 The chamfer is removed.

Nota: You can use this method to create a corner between two lines or an end segment of a polyline. When the two entities intersect, the part you click is kept.

16.26 Filetting entities

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.

The TRIMMODE system variable defines whether or not to keep the original length of the filleted entities or to trim or lengthen them to the endpoints of the fillet arc.



16.26.1 Filleting two entities or polyline segments

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Select the first entity or polyline segment.

6 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

7 Select the second entity or polyline segment.

8 The fillet is created.

16.26.2 Filleting all vertices of a polyline

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Choose the **Polyline** option.

You are prompted: Select 2D polyline to fillet [selection options (?)]:

6 Select the polyline.

7 The polyline is filleted.

16.26.3 Filleting two parallel lines

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Select the first line.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Select the second line.

5 The fillet is created.

16.27 Editing polylines

Use the PEDIT command to:

- Convert an entity into a polyline
- Fit a curve to a polyline
- Taper a polyline uniformly along its length
- Reverse the direction of a polyline

Edit a polyline in the **Properties** panel to:

- Open or close a polyline
- Apply a uniform width to a polyline
- Change the width of a polyline segment
- Set the linetype mode (PLINEGEN system variable)

Use the edit polyline segments tools in the Quad to:

- Insert a vertex
- Add vertices at the start or end
- Modify the bulge factor of a segment
- Stretch a vertex or a segment
- Delete a segment

Use grips editing to:

- Move a vertex
- Move a segment

16.27.1 Launching the PEDIT command



Do one of the following:

- Double-click a polyline.
- Click the Edit Polyline tool in the **Modify** menu.
- Click the Edit Polyline tool on the **Modify** toolbar.

16.27.2 Converting an entity into a polyline

1 Launch the PEDIT command.

2 Select the entity.

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>:

3 Press Enter or right-click.

4 **You are prompted**: Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>:

5 Choose an option or right-click to stop.

16.27.3 Fitting a curve to a polyline

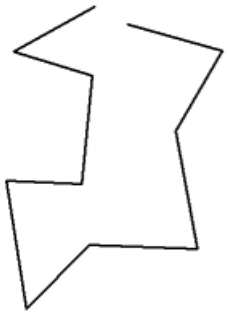
1 Launch the PEDIT command.

2 Select the polyline.

3 Do one of the following:

- Choose the **Fit** option.
- Choose the **Spline** option.

4 Press Enter or right-click to conclude the PEDIT command.

		
Original polyline	After applying Fit	After applying Spline

Nota:

- Use the **Decurve** option to restore the original polyline.
- The SPLINESEGS (Spline Segments) system variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

16.27.4 Tapering a polyline uniformly along its length

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Taper** option.

You are prompted: Enter new starting polyline width <current width>:

4 Type a value.

5 **You are prompted**: Enter new ending polyline width <current width>:

6 Type a value.

7 The polyline is tapered.

8 Press Enter or right-click to conclude the PEDIT command.

16.27.5 Reversing the direction of a polyline

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Reverse direction** option.

The direction is reversed.

4 Press Enter or right-click to conclude the PEDIT command.

16.27.6 Opening or closing a polyline

1 Select the polyline.

2 Select the **Closed** property under **Misc** in the **Properties** panel.

3 Choose **No** or **Yes**.

Or

1 Double-click the polyline.

2 Type C to close, or O to open.

16.27.7 Applying a uniform width to a polyline

1 Select the polyline.

2 Select the **Global width** property under **Geometry** in the **Properties** panel.

3 Type the desired width.

16.27.8 Changing the width of a polyline segment

1 Select the polyline.

2 Select the **Vertex** property under **Geometry** in the **Properties** panel.

An X mark is displayed at the start vertex of the polyline.

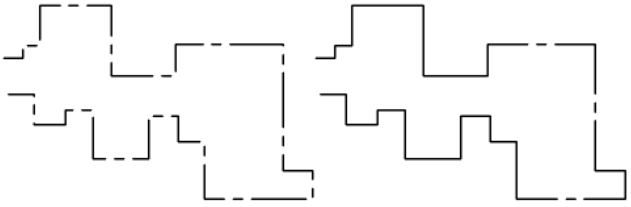
3 Click the arrow keys at the right hand side of the vertex property field to set the X marker at the start vertex of the segment you want to change the width of.

4 Select the **Start width** property and type the desired width.

5 Select the **End width** property and type the desired width.

16.27.9 Setting the linetype mode

The Linetype generation mode (PLINEGEN system variable) controls how a dashed linetype is applied to a multi-segment polyline. When disabled, dashed linetypes are applied per segment: start/end with a dash at each vertex.



PLINEGEN On (left) and Off (right)

1 Select the Polyline.

2 Select the **Linetype generation** property under **Misc** in the **Properties** panel.

3 Select **disabled** or **enabled**.

16.27.10 Using the edit polyline segments tools in the Quad

- 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.
- Adjust bulge: 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.

1 Place the cursor over the segment you want to edit.

1 The entire polyline is highlighted.

2 Hold down the Ctrl key.

3 Only the segment under the cursor is highlighted.

The editing tools are available in the Quad.

4 Select a tool, then follow the instructions in the Command line.

16.28 Exploding entities

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.

MText entities explode into single line texts.

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 entities into polylines. You can then use the JOIN command to create a single polyline for each letter. For some letters, such as a, e or p, a second polyline is needed for the 'hole').

16.28.1 Exploding entities

Do one of the following:

- Launch the EXPLODE command.
- Select entities, then press Enter.
- Select entities, then choose **Explode** in the **Modify** command group in the Quad.

16.29 Measuring entities

The MEASURE command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The markers are placed starting at the closest 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.

16.29.1 Measuring an entity using points

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Do one of the following:

- Type a value.
- Specify two points.

The points are placed along the entity at the specified interval.

Nota: The visibility of the points depends on the current value of the PDMODE system variable.

16.29.2 Placing blocks a specified interval along an entity

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

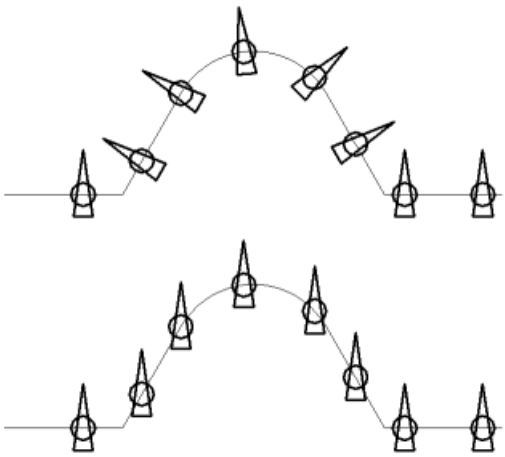
8 Choose whether or not to align the blocks.

9 **You are prompted**: Segment length:

10 Do one of the following:

- Type a value.
- Specify two points.

The blocks are placed along the entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

16.30 Dividing entities

The DIVIDE command places markers - points or blocks - along a selected entity. The markers divide the entity into a specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

16.30.1 Dividing an entity using points

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Type a value, then press Enter.

5 The entity is divided by points.

Nota: The visibility of the points depends on the current value of the PDMODE system variable.

16.30.2 Dividing an entity using blocks

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

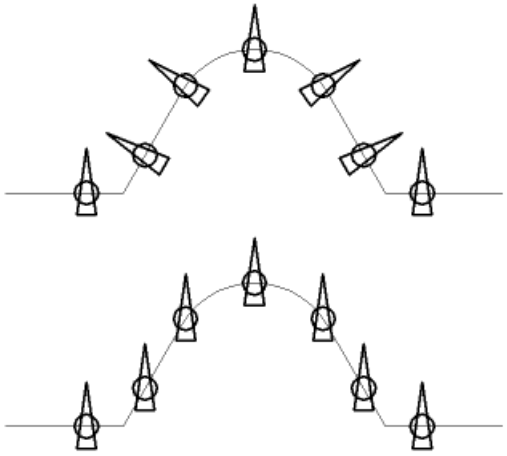
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

8 Choose whether or not to align the blocks.

9 **You are prompted**: Number of segments:

10 Type a value, then press Enter.

11 The entity is divided by blocks.



Blocks aligned (top) or not aligned (bottom)

17. Imprimir e Plotar

17.1 General procedure for printing

The PLOT and PRINT commands plot drawings to printers and to files, through the **Print** dialog box.

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 QPRINT command prints the drawing with the default plot configuration, without displaying the **Print** dialog box.

The PAGESETUP command creates and edits page setups for plotting drawings in the **Drawing Explorer** dialog box.

A standard print procedure consists of the following steps:

- Choose either a **Layout** tab or the **Model** tab.
- Open the **Print** dialog.
- Select a **Page Setup name**. (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**. (2)
- Choose a **Plot style table**. (3)
- Choose a **Paper size**. (4)

  The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the paper size unit. (5)
- Choose the **Drawing orientation**.(6)
- Define the **Plot offset**. (7)
- Specify the **Plot area** (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.

Nota: All print settings and options are saved in the **Model** and each of the **Layouts**.

Nota:

On Mac or Linux, printing will always print to PDF. So a PDF document will be generated that then still needs to be sent to the printer for physical printing.

On MAC and Linux there is a relevant PAGESETUP command.

17.1.1 Opening the Print dialog box

Do one of the following:

- Launch the PRINT or PLOT command.
- Press **Ctrl + P**.
- Choose **Print...** in the **File** menu.
- Click the **Print** tool in the **Output / Plot** ribbon panel (Drafting workspace).
- Select the **Model** tab or a **Layout** tab, then right-click and choose **Print** in the context menu.

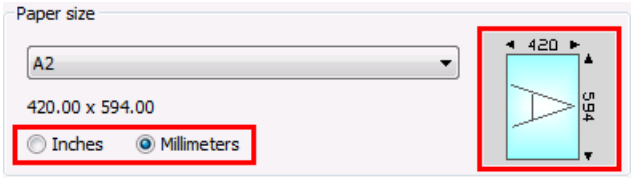
17.1.2 Choosing the paper size

Press the down arrow, then select the paper size in the **Paper Size** list.

The available sizes depend on the selected printer.

17.1.3 Choosing the paper unit

Select either **Inches** or **Millimeters**.



The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

Nota: The **Scale** and **Plot offset** setting units are changed according to the chosen **Paper unit**.

17.1.4 Choosing 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.

17.1.5 Defining 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.

When in a layout, click the **Apply** button to update the display.

To specify the plot area:

Option	Result
View	Prints the portion of the drawing that is displayed 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 X- and Y-coordinates of the corners of the print area in the **X:** and **Y:** fields.

17.1.6 Defining the plot scale

Nota: The SCALELISTEDIT command allows you to edit the list of available scales in the **Print** dialog and the **Standard Scale** property of a paper space viewport.

... when printing a paper space layout:

If you print in paper space, the final scale of the plot output is the product of the viewport scale 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 unit.
- Set the scale of each viewport to the scale you want in the plot output.
- The **plot scale** reflects the relationship between paper unit and drawing unit. (see table below)

Paper unit	Drawing unit	Plot scale	Printed mm or inches =	Drawing unit
mm	mm	1:1	1	1
mm	cm	10:1	10	1

Paper unit	Drawing unit	Plot scale	Printed mm or inches =	Drawing unit
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

... when printing model space:

If you print in model space, the scale of the plot output equals the plot scale, on the condition that your drawing unit matches the paper unit. Otherwise you need to multiply the plot scale with respect to the relationship between the paper unit and the drawing unit. 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

17.1.7 Setting the plot options

Click to toggle the **plot options** On / Off.

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. ***Nota:*** This applies to viewports which have the 2D Wireframe or Wireframe visual style only.
Plot transparencies	Plots entities using the value of the **Transparency** property.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities are plotted 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.

17.1.8 Setting the shade plot mode

By default the Shade Plot mode is set to **As displayed**.

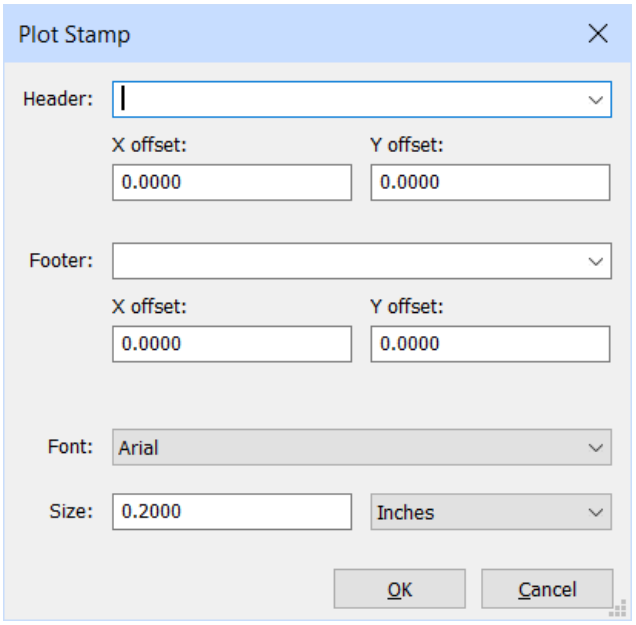
Click the list button, then select an option to plot using a different visual style.

Nota: This option applies to model space printing only. The Shade Plot mode of a paper space viewport is defined in the Shade Plot property of the viewport.

17.1.9 Defining the plot stamp

1 Click the **Edit plot stamp** button.

The **Plot stamp** dialog box appears.



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 (see table below).
- Leave the field empty.

3 Define the X and Y offset for the header.

4 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 (see table below).
- Leave the field empty.

5 Define the X and Y offset for the footer.

6 Select a font in the **Font** list button.

7 Enter a value in the **Font Size** field.

8 Set the unit type to **Millimeters** or **Inches**.

9 Click the **OK** button to save the plot stamp.

Variable Name	Description
$Name	Adds the name of the current user, as defined in the **User Name** field of the **License Information**.
$Company	Adds company name of the current user, as defined 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.

Nota:

- 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,$Date** places your name left, company name centered and the current date at the right.
- Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.
- e.g. **$Company, ,** ($Company,<space>,<space>) outlines the company name left. , ,$Company (<space>,<space>,$Company) outlines the company name right.
- The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the **Settings** dialog box under **Program Options** > **Plot and Publish** > **Plot Stamp**.

17.1.10 Specifying 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.

17.1.11 Previewing the plot output

Click the **Preview** button on the **Print** or **Page Setup** dialog box.

Choose **Print Preview...** in the **File** menu.

Type **preview** in the Command line, then press Enter.

17.2 Plotter configuration

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 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.

17.2.1 Configuring the built-in PDF printer

1 Open the **PRINT** dialog box.

2 Click the **Printer/Plotter configuration** list button and choose **Print As PDF.pc3** in the list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter Configuration Editor** dialog is displayed with the **Print As PDF.pc3** configuration file loaded.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

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 is displayed.

8 (Option) Adjust the **Output quality** and **Layers support** settings.

9 Click the **Manage Custom sizes ...** button.

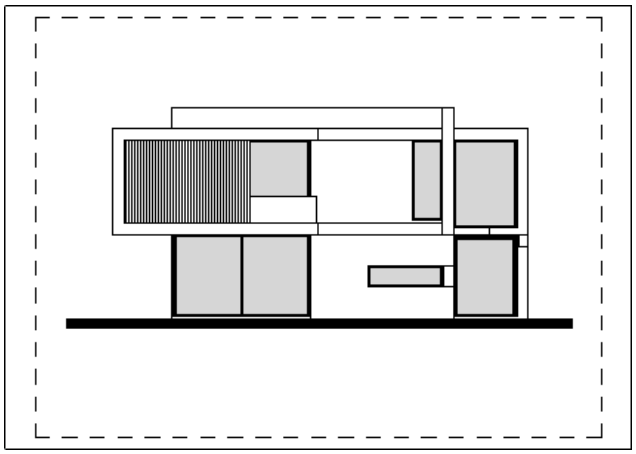
10 To create a custom paper size, do the following:

- Click the **Add** button.

  A new paper size is added.
- Double-click the newly created paper size.
- Type a name for the new paper size, then click the **OK** button.
- Define **Width** and **Height** for the new paper size.
- Define the **Paper Margins**.

Paper margins define the printable area of paper sheet.

If the DISPPAPERMARGINS system variable is ON, dashed lines indicate the paper margins on the paper sheet in a layout.



The paper sheet background is displayed if the DISPPAPERBKG 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.

17.2.2 Creating a plotter configuration

1 Launch the PLOTTERMANAGER command.

1 A **File** dialog box is displayed with the folder, defined by the PLOTCFGPATH system variable.

2 Double-click **Create a Plotter Configuration** shortcut.

The **Plotter Configuration Editor** dialog box is displayed.

3 In the **Printer name** list, select the printer you want to create a plotter configuration (PC3) for.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 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.

6 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.

7 Click the **OK** button on the **Plotter Configuration Editor** dialog.

The **Save Plotter Configuration As** file dialog appears.

8 (option) Edit the PC3 file name.

9 The default name is **<Selected Printer>.pc3**.

10 Click the **OK** button on the **Changes to a Printer Configuration File** dialog.

The PC3 file is created.

Nota: You can create multiple plotter configurations for the same printer if you save them under different names.

17.2.3 Editing a plotter configuration

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Select a plotter configuration (PC3) in the **Printer / Plotter configuration** list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter configuration editor** dialog opens.

4 Proceed in the **Creating a plotter configuration** procedure above starting from step 4.

Nota: 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 **Creating a plotter configuration** procedure.

17.2.4 Assigning 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 line, 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.

Nota:

- 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 you to choose a similar size on any of the available printers.

17.3 Plot style tables

The **Plot Style Table Editor** displays the plot styles contained in the specified plot style table.

17.3.1 Opening the Plot Style Table editor

Do one of the following:

- Click the **Edit Plot Style** button on the **Print** or **Pagesetup** dialog box.
- 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 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.

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. ***Nota:*** 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.

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.

Property	Description
Plot styles	Displays the names of plot styles contained in the selected 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 (STB): • 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	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.
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 when 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.

Property	Description
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.

Property	Description
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.

Nota: Entities which are created in **true color** are always plotted using their own object properties (color, lineweight, ...).

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.

17.3.2 Assigning a plot style table

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**.

Plot style tables are used only if the **Plot with plot styles** plot option on the **Print** dialog box is ticked. When you turn off plot style tables, entities print according to their own properties.

Assigning a plot style table

1 Open the **Print** or **Page Setup** dialog box.

2 Select a **Plot style table** from the list box in the **Plot Style Table (pen assignments)** section.

3 Click the **OK** button.

17.4 The plot styles manager

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.

17.4.1 Launching the plot styles manager

Do one of the following:

- Choose **Plotstyle manager...** in the **File** menu.
- Launch the STYLESMANAGER command.

A **File** dialog box is displayed, showing the content of the PlotStyles folder, which is defined by the PLOTSTYLEFOLDER system variable.

17.4.2 Creating a new plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the **Create a Plot Style Table** shortcut. The **Add Plot Style Table Editor** dialog box opens.

3 Choose **Create new from scratch**.

4 Do one of the following:

- Select either **CTB** or **STB** from the **Table type** selection button.
  - **CTB** (Color-dependent Plot Style Table) - 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 Style Table) - 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.

5 Type a name in the **Table name** field, then click the **OK** button.

The **Plot Style Table Editor** dialog box is displayed.

17.4.3 Creating a new plot style table in the Print dialog box

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Under **Plot style table**, click the **Create new plot style**.

The **Add Plot Style Table** dialog box appears.

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** dialog box appears.

If the **Use this plotstyle for the current drawing** option is chosen, the new plot style table is used in the current layout.

Nota: The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

17.4.4 Editing a plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the plot style table you want to edit.

3 The **Plot Style Table Editor** dialog appears.

17.4.5 Editing a plot style table in the Print dialog box

1 Open the **Print** dialog box.

2 Select a plot style table from the **Plot Style Table** list button.

3 Click the **Edit plot style** button.

17.5 Batch plot

The PRINT command prints a single layout at a time. 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 you 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.

18. Renderização

18.1 Using TwinMotion

Nota: This feature is only available for BricsCAD on Windows.

Twinmotion offers real-time, immersive 3D architectural visualization using the power of Unreal Engine from Epic Games. All your BricsCAD model data are exported using the Unreal Engine file type (*.udatasmith) in the EXPORT command.

To use Twinmotion with BricsCAD V21 you need to:

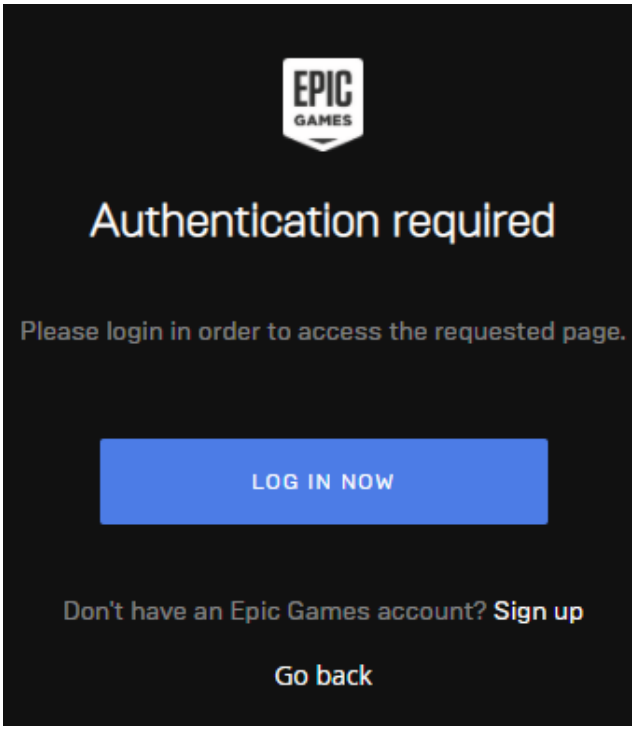
- Install the **Epics Games Launcher**.
- Download and install **Twinmotion**.
- Export your model to a ***.udatasmith** file.

18.1.1 Installing Twinmotion

From the Twinmotion Help pages:

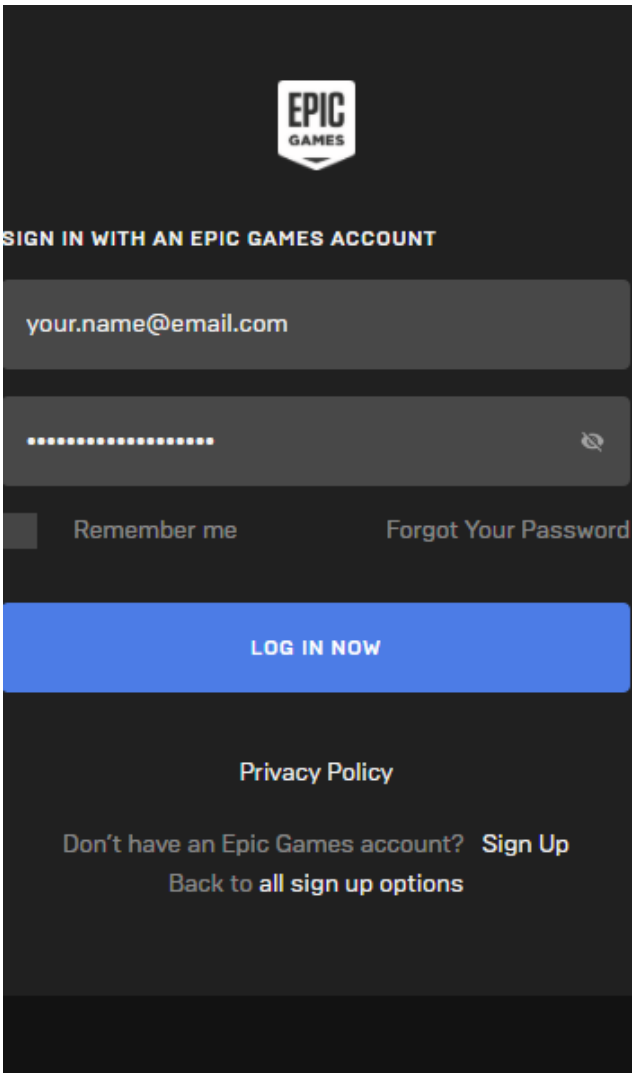
1 Go to https://www.unrealengine.com/en-US/twinmotion.

2 Click on **Get Started Now**.



If you do not have an Epic Games account yet, click **Sign up**, then fill in all required information and click **Create Account**.

3 Click **LOG IN NOW**.



4 Once you are logged in, click on **Download the Epic Games Launcher**.

5 Once the Epic Games Launcher is downloaded, make sure to click on the **Unreal Engine** tab on the left side.

You will now see the **Twinmotion** tab in the top right.

6 Click on **Install** to install Twinmotion.

7 Carefully choose your file location, it is strongly recommended to use the default location.

18.1.2 Exporting your model

1 Open your model in BricsCAD BIM.

2 Launch the DATASMITHEXPORT command.

3 The **Export drawing to** dialog box appears.

The **Save as type** is Unreal Engine (*.udatasmith).

4 Type a name or accept the default name, then click the **Save** button.

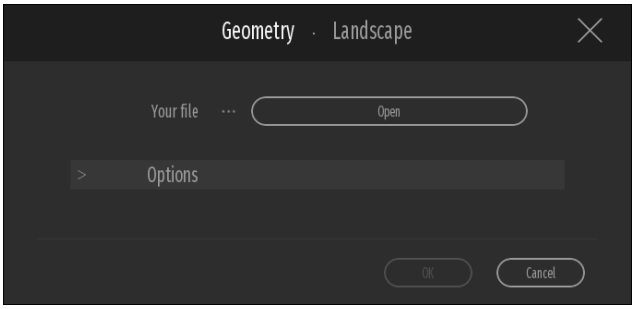
The exported model can now be imported in Twinmotion.

18.1.3 Importing your model in Twinmotion

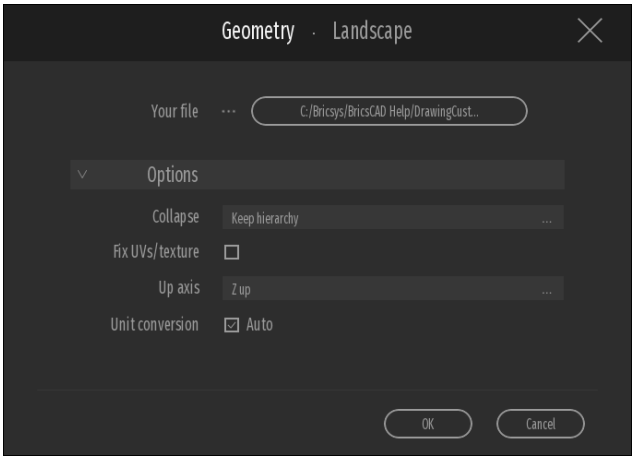
1 Launch Twinmotion.

2 Do one of the following:
- Click the **Import Files** button.
- Press **Ctrl + I**.

3 Make sure **Geometry** is selected, then click the **Open** button.



On the **Import** dialog, select a **Unreal Engine** (*.udatasmith) model file.



4 Click **Options**, and set the **Collapse** option to **Keep Hierarchy**.

This structures the model the same way as the layers used in BricsCAD.

5 Click the **OK** button to import the model.

18.2 Rendering overview

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 system variable. It is recommended to install the most recent drivers for your graphics card.

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset via dialog box.

18.3 Rendering preferences

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset through a dialog box.

18.3.1 Setting the render preferences

1 Open the **Settings** dialog box.

2 Under **Program Options**, expand the **Rendering** settings category.

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.

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 may 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 illumination 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.

Tile mode light synch:

Controls the synchronization of lighting in all model space viewports. (Internal use only)

18.4 Rendering tools

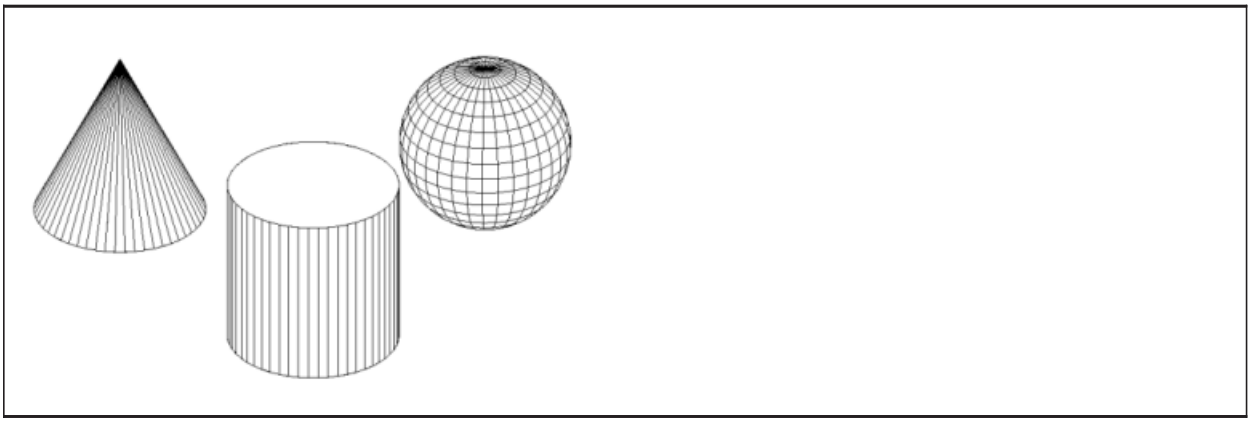
Rendering tools are available in:

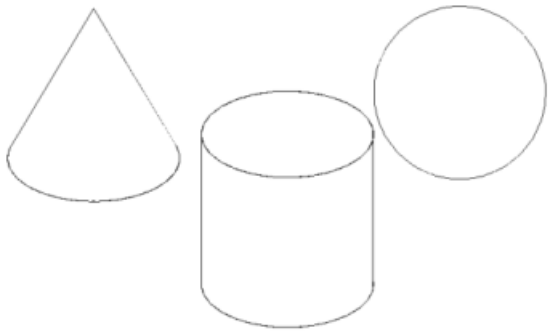
- The **Rendering** toolbar.
- In the **View** > **Rendering** menu.
- The **View** and **Visualize** ribbon tabs.
- In the **General** tab of the **No Selection** Quad.

Icon	Tool Name	Command	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 one other than **2Dwireframe**, the hidden line representation is kept until a different shade mode is chosen.
	Shade	SHADE	Shades 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	Shades the entities between the polygon faces. Curved surfaces are segmented

Icon	Tool Name	Command	Description
	Gouraud	SHADEMODE + G	Shades 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 you 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 Drawing Explorer - Lights
	Geographic Location	GEOGRAPHICLOCATION	Opens the **Geographic Location** dialog box.
	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.
	Animation Editor	ANIMATIONEDITOROPEN	Opens the **Animation Editor** panel.

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.



Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (top) and On (bottom).


18.5 Render materials

The MATERIALS command creates materials and edits their properties through the Drawing Explorer.

The MATBROWSEROPEN command opens the **Render materials** panel, which shows an overview of available high-resolution render materials.

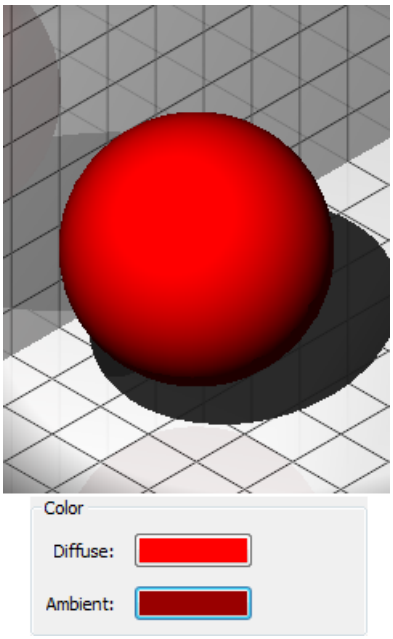
The MATBROWSERCLOSE command closes the **Render materials** panel.

The MATERIALMAP command allows you 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.

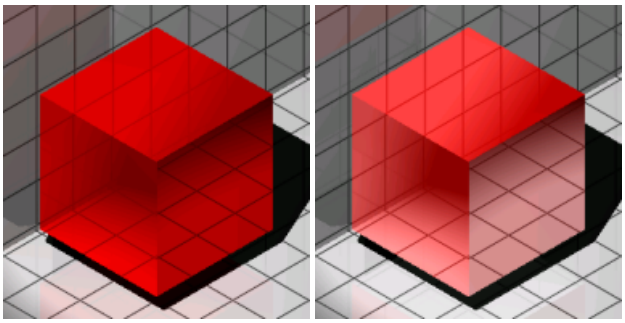
Material definitions are saved in the drawing. Each drawing contains the **Global** material, which you cannot to delete or rename. You can edit the properties of the **Global** material though.

18.5.1 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 than direct light.

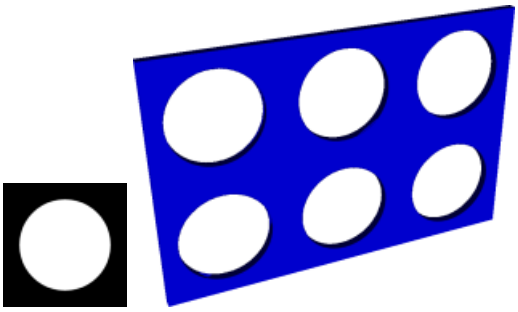


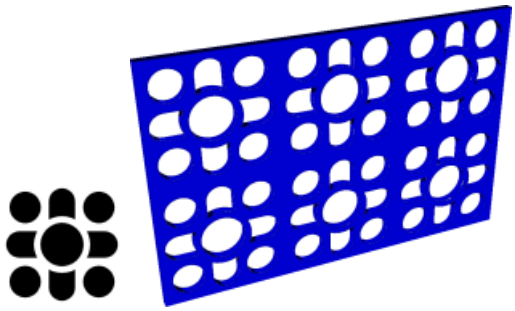
- **Highlights**: highly reflective highlights in the color of the material.



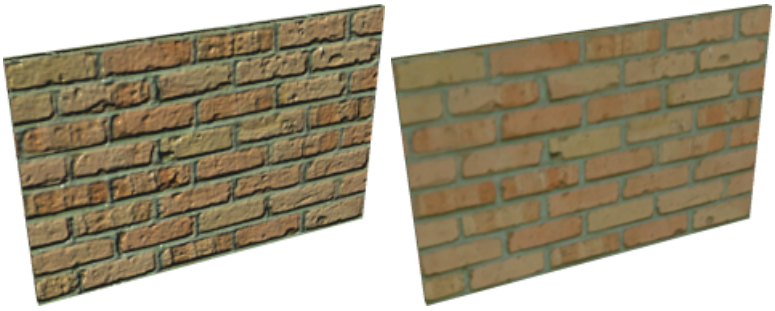
Metallic Highlights (left) and Non-metallic Highlights (right)

- **Shading**: how the color and brightness of a surface varies with lighting.
- **Texture-Mapping**: adds detail to a surface that 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 (grayscales). (Black=Transparency map)





- **Bump map**: simulates a certain roughness or bumpiness on surfaces. (left = with bump map)



- **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.
- **Self-illumination**: it appears as if a surface is emitting its own light. No light is cast on other objects.

18.5.2 Defining render materials

See the MATERIALS command.

18.5.3 Assigning render materials

Materials can be assigned to a layer and to entities or subentities. 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.

18.5.4 Adding a render material to the drawing

1 Select the material in the **Render Materials** panel.

2 Right-click and choose the **Add to drawing** option.

18.5.5 Assigning a render material to entities

When the current visual style is BIM, Modeling, Realistic or X-Ray, entities and subentities are rendered using their material.

Method 1: Using the Properties panel

1 Select the entities or sub-entities (e.g. faces or 3D solids).

2 Under **3D Visualization** > **Material**, click the down arrow and select the material in the **Material** list.

Nota: Materials in the drawing are available only.

Method 2: Using the Render Materials panel

1 Select the material, which launches the MATERIALASSIGN command.

1 The cursor turns into a paintbrush.

You are prompted: Select object to assign material to:

2 Click the entities or subentities to assign the material.

3 (option) Select a different material, then click more entities or subentities.

4 (option) Hold down the Alt key.

5 The cursor turns into a dropper.

You are prompted: Select material from object:

Click an entity or subentity to copy the material from, then click entities or subentities to assign the material to.

Nota: The Hotkey Assistant appears at the bottom of the screen, indicating the current selection mode: 3D solids or 3D solid faces. Hold down the Ctrl key to invert the selection mode.

To assign a material to a selection set:

1 Select the entities.

2 Search the material in the **Render Materials** panel.

3 Click the material.

4 The material is assigned to all entities or subentities in the selection.

18.5.6 Removing a render material

Assigning the Global material, removes a rendering material from an entity or subentity.

1 Select the entities and/or subentities.

2 Select the **3D Visualization / Material** field in the **Properties** panel and select the **Global** material.

Nota: Clicking the **Erase Material** tile in the **Render Materials** panel sets the Material property to BYLAYER.

18.5.7 Assigning a material to a layer

1 Select the **Material** field of the layer in the **Layers** panel or the **Drawing Explorer – Layers** dialog box.

2 Click the down arrow and select the material in the list.

3 Materials in the drawing are available only.

18.5.8 Adding the Redway material library

Materials in the folder linked under **RENDERMATERIALSPATH** appear in the **Render** materials panel. To add the **Redway Material Library**, follow this procedure:

1 Download the complete Redway Material Library here.

2 Unzip the downloaded file and copy the path of the folder. The path should end with ...

\all_redway_materials.

3 In the **Render Materials** panel, click the hamburger menu ( ).

4 Select **Manage libraries** in the menu.

The **Settings** dialog box is displayed with the RENDERMATERIALPATH system variable selected.

5 Click the Browse button ( ).

The **Path List** dialog box appears.

6 Click the **Add Path** button.

An empty path field is added.

7 Do one of the following:

- Paste the path you have copied in the empty path field.
- Click the Browse button ( ), then select the folder in the **Choose a folder** dialog box.

The downloaded materials appear in the **Render Materials** panel.

18.5.9 Adding your own materials

1 Download the textures you want to use.

Nota: you can download a photo of your texture, but in order to get the most realistic view it is best to download a zip file that gives you different representations of the texture:



2 Launch the MATERIALS command.

3 The **Drawing Explorer – Materials** dialog box appears.

4 Click the **New** button.

5 Give your new material a name.

6 Check on the **Diffuse Map** and the **Bump Map**.

7 Edit the path of the diffuse map ( ) to the colored image of your texture.

8 Edit the path of the bump map ( ) to the normal image of your texture.

This will give your render material a more 3D and realistic look.

9 Edit the scale of your image until you get a realistic image in the preview.

10 The scale of the texture represents the number of drawing units covered by the full width of your image.

11 Right-click on your new material and choose **Add material(s) library** in the context menu.

12 Close the **Drawing Explorer** dialog box.

13 Open the **Render Materials** panel and click on the folder **User Defined**.

You find your new material in this folder of the library.

This folder is automatically generated in the following path: C:\ProgramData\Bricsys\RenderMaterials\UserMaterials\User defined.

18.6 Lights

Lighting is the key to make the viewer believe they are looking at a realistic scene.

18.6.1 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 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.

18.6.2 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.

LIGHTGLYPHDISPLAY (Light glyph display): Controls the display of light glyphs.

FBXEXPORTLIGHTS (Fbx Export Lights): Specifies whether to export lights from model space.

18.6.3 General properties

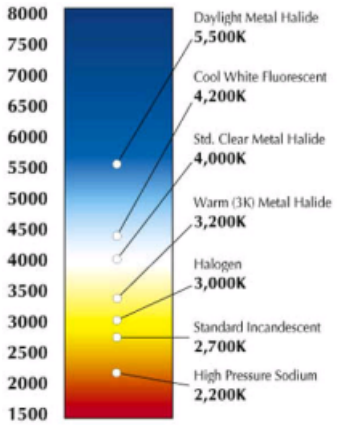
The general properties are common to all lights:

- **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.

18.6.4 Photometric light properties

LIGHTINGUNITS = 1 (American units) or 2 (International units).

- **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.

	D65White Fluorescent Cool White White Fluorescent Daylight Fluorescent Incandescent Xenon Halogen Quartz Metal Halide Mercury Phosphor Mercury High Pressure Sodium Low Pressure Sodium
color temperatures	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).

18.6.5 Geometry properties

- **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.

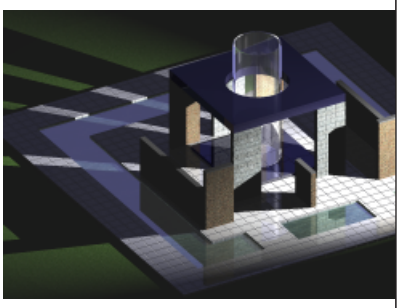
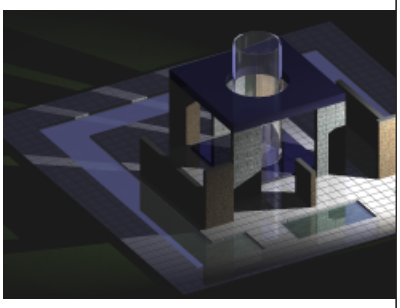
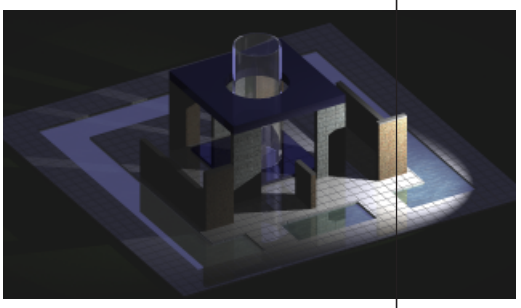
18.6.6 Attenuation properties

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.

		
No attentuation	Inverse linear attentuation	Inverse square attentuation

Nota:

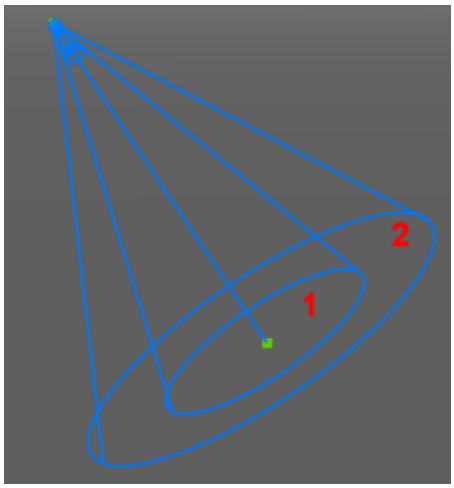
- The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0).
- Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

18.6.7 Rendered shadow details

- 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.

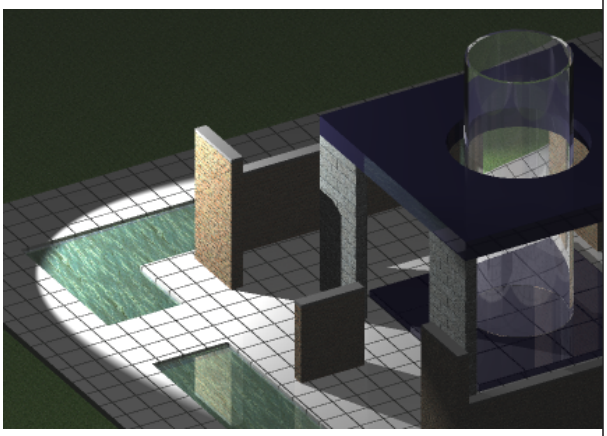
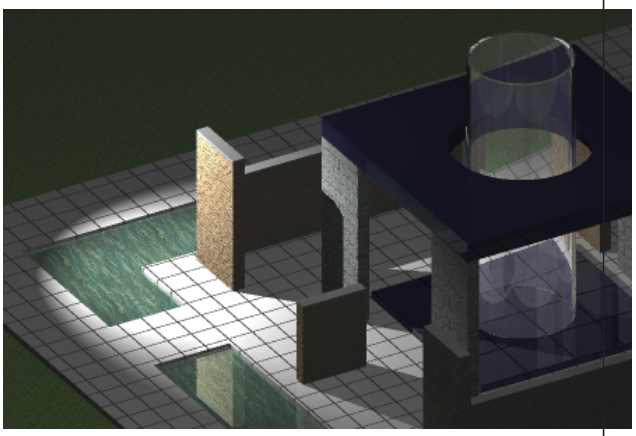
18.6.8 Spotlight hotspot and falloff properties

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).



Hotspot angle: angle of the central light cone (defines the hotspot)

Falloff angle: angle of the full light cone

	
Hotspot angle = 55, Falloff angle = 60	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).

18.6.9 Managing Lights

The LIGHTLIST command opens the **Drawing Explorer – Lights** dialog box.

Here you can:

- Create new lights.
- Delete lights.
- Select lights in the drawing.
- Switch lights On/Off.

18.6.10 Defining a light

1 Do one of the following:

- Launch the LIGHT command.
- Click the New () button on the **Drawing Explorer – Lights** dialog box.

  You are prompted: Enter light type [Point/Spot/Web/Distant] <Web>:

  Specify the light type.
- Click a toolbutton on the **Lights** toolbar or the **View** or **Visualize** ribbon tab.

-  : spot light
-  : point light
- : distant light
-  : web light

You are prompted: Specify source position <0,0,0>:

2 Specify a point in the drawing.

3 **You are prompted**: Specify target position <0,0,-10>:

4 Specify a point in the drawing.

5 **You are prompted**: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Attenuation/FilterColor/<eXit>:

6 Specify the properties of the light.

7 Choose **Exit** to conclude the command and create the light.

A light glyph is displayed in the model, if LIGHTGLYPHDISPLAY = On.

Nota: It is not possible to define web lights if **LIGHTINGUNITS** =0 (No lighting units).

18.6.11 Editing lights

Lights can be edited in the **Properties** panel.

1 Do one of the following to select the light in the drawing:
   - Click the glyph of the light(s).
   - Select the light(s) in the **Drawing Explorer – Lights**, then right-click and choose **Select lights** in the drawing in the context menu.

   The properties of the selected light(s) appear in the Properties panel:

2 Edit the properties.

3 Press the Esc key to stop.

18.6.12 Defining the sun properties

1 Launch the SUNPROPERTIES command.

1 The **Drawing Explorer - Lights** dialog appears, showing the sun properties in the **Editor** pane.

2 Edit the properties.

Nota: When LIGHTINGUNITS = 1 or 2, the **Color** property of the sun cannot be edited.

18.6.13 Defining the geographic location

1 Launch the GEOGRAPHICLOCATION command.

1 The **Geographic Location** dialog box appears.

2 (option) Check the **Store geographic location information in drawing** option.

3 (option) Check the **Use Map Grid coordinate system** option.

4 Do one of the following:
   - Type the latitude and longitude of the location in the **Latitude** and **Longitude** fields and select the appropriate hemisphere for each setting.
   - Click the **Choose location...** button, then select a city and timezone and click the **OK** button.

5 (option) If you did not select a time zone in the previous step, choose a time zone in the **Time Zones** list.

6 (option) Specify the **Coordinates of the corresponding location in the drawing**.

7 Define the direction of the north.

8 Click the **OK** button.

18.7 Creating a rendered image

The rendered image appears 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.

Nota:

- High anti-alias values incur a high calculation cost.
- The values of the lighting system variables influence the rendering result.

18.7.1 Rendering a viewport

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Viewport** option.

3 Click the **OK** button.

The view is rendered.

18.7.2 Rendering in a window

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Window** option.

3 Click the **OK** button.

The view is rendered in a popup window.

18.7.3 Saving a rendering to a file

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **File** option.

3 Click the **Browse** button ( ), at the right hand side of the **File** field. The **Render output file** dialog box appears.

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.

8 Click the **OK** button to create the file.

19. Comentários do usuário

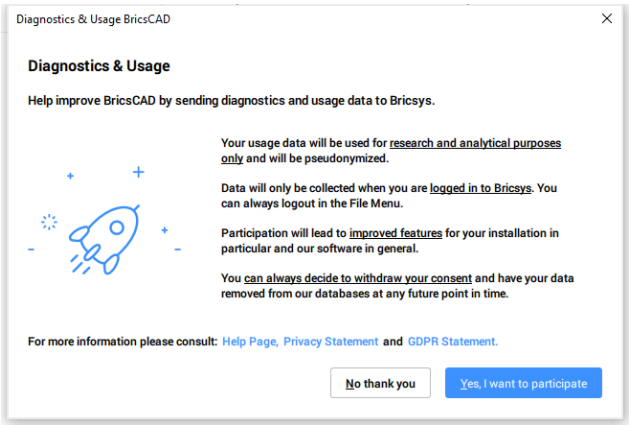
19.1 Diagnostics and usage data collection

The Diagnostics and Usage Data Collection process allows users to help improve BricsCAD by sending pseudonymized diagnostics and usage data to Bricsys. You can decide at any time to withdraw your consent and have your data removed from our databases.

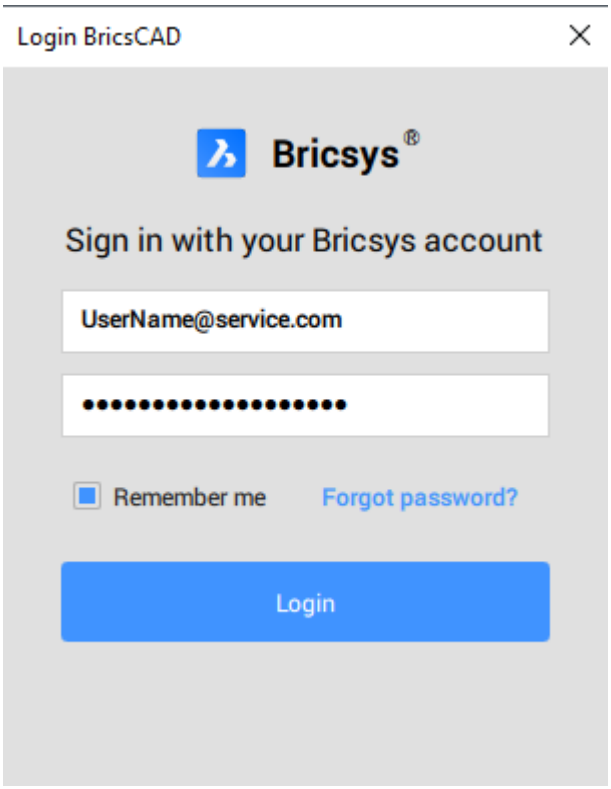
When transferring data:

- A unique, anonymous ID is used.
- The following data and information are removed:
- All personal project information .
- All texts and leaders.
- All information that is stored in the Settings dialog.

When you launch BricsCAD the second time, a dialog box displays asking you to participate.



When you agree, you are prompted to log in with your Bricsys account.



If you tick the**Remember me** checkbox, you will be signed in automatically each time you launch BricsCAD. The status of your agreement is saved through theDATACOLLECTIONsystem variable as indicated in the table below.

Value	Status	Result
-2	Not Asked	When you close BricsCAD the value is set to [-1] Not Set.
-1	Not Set	When you restart BricsCAD the**Diagnostics & Usage**dialog displays, asking you to participate.
0	Off	The Diagnostics and Usage Data Collection process is switched Off.
1	On	The Diagnostics and Usage Data Collection process is switched On. Data will be transferred when you are logged in only. To temporarily stop transferring data, choose**Logout...**in the**File**menu or launch the-LOGOUTcommand. To restart transferring data, choose**Login...**in the**File**menu or launch the-LOGINcommand.

19.1.1 Stopping the process

1 In the Settings dialog, under User Preferences, setDATACOLLECTION to[0] Off.

2 Close and restart BricsCAD.

19.1.2 Restarting the process

1 In the Settings dialog box, under User Preferences, setDATACOLLECTIONto [-1] Not Set.

2 Close and restart BricsCAD.

3 Choose**Yes, I want to participate**, then log in with your Bricsys account.

19.1.3 Removing your data from the database

To withdraw your consent and have your data removed from our databases send in a support request and select the **Data Collection** category under BricsCAD.

20. BricsCAD BIM

20.1 O Que há de novo

20.1.1 Overview

BricsCAD V21 offers a plethora of new and improved tools to help maximize your productivity.

This section gives an overview of all the new features and improvements in **BricsCAD V21 BIM**:

Modeling techniques na página 962

Building data na página 969

Project collaboration na página 978

Design documentation na página 979

Point clouds na página 988

For information about the new features and improvements in **BricsCAD V21 Core and Civil tools**, go to the What's new in BricsCAD V21 section. It contains the following articles:

Overview na página 31

User interface na página 32

Productivity na página 35

Drafting na página 49

Modeling na página 51

Civil tools na página 60

Performance and compatibility na página 65

For information about the new features and improvements in **BricsCAD V21 Mechanical tools**, go to the What's new in BricsCAD V21 Mechanical section. It contains the following articles:

Visão Geral na página 1532

Gerenciador de BOM na página 1533

Bibliotecas de peças na página 1536

Parâmetros e Restrições na página 1537

Peças padrão na página 1537

Chapa metálica na página 1537

20.1.1.1 Release notes

Click here for an overview of the release notes.

20.1.1.2 Diagnostics and Usage Data Collection

Many of the improvements were inspired by requests and suggestions from you, BricsCAD users. At Bricsys, we continue to expand our methods of gathering user feedback with new data collection tools. When you launch BricsCAD for the second time, a dialog box displays, inviting you to participate in the

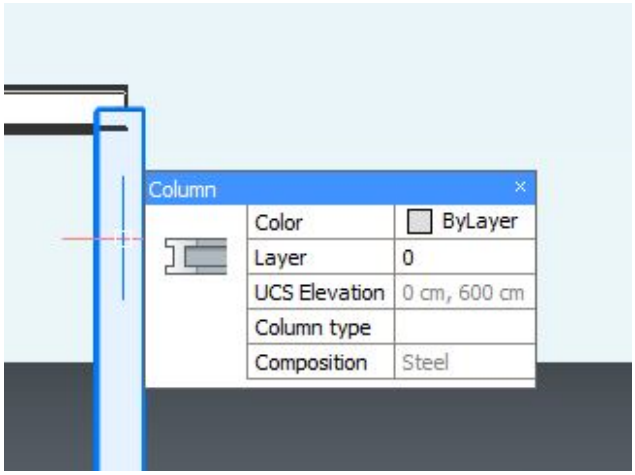
Diagnostics and Usage Data Collection program. The data collection process enables you to share anonymous diagnostic and usage data with Bricsys to help improve BricsCAD. You can withdraw your consent and have your data removed from our databases at any time.

20.1.2 Modeling techniques

20.1.2.1 BIM Profiles

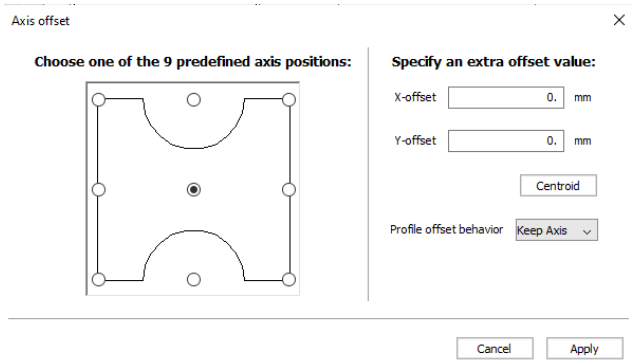
BIMAPPLYPROFILE / BIMLINEARSOLID

The composition material of a profile is now attached to the created solid in the BIMAPPLYPROFILE command and BIMLINEARSOLID commands.



BIMPROFILES

When changing the Offset property of a CustomShape profile via the Profiles dialog, you can choose whether to keep the position of the solid or its axis, for linear solids using said profile.



BIMPROFILESTANDARDS

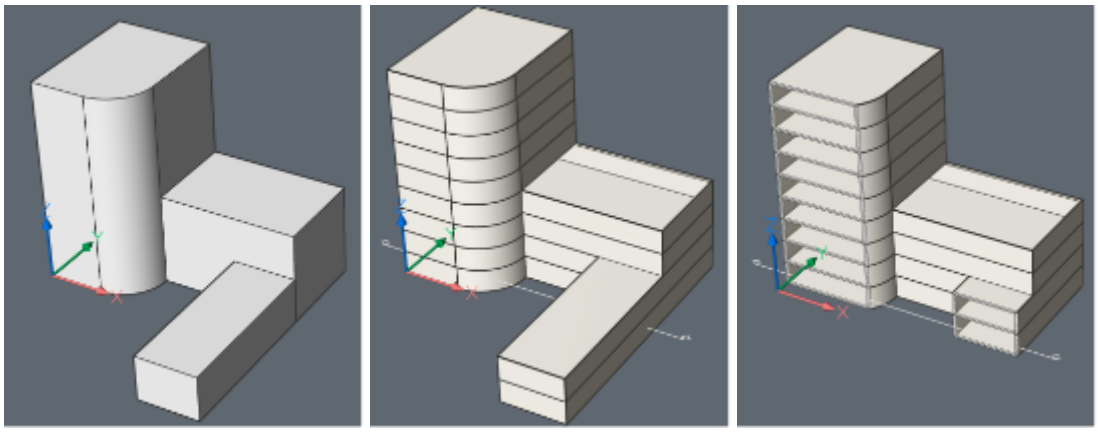
The BIMPROFILESTANDARDS setting is no longer case sensitive. In case there are no profiles matching the value of the BIMPROFILESTANDARDS setting, a warning message displays.

20.1.2.2 Mass Modeling tools

The first prototype of the new BIMQUICKBUILDING command has been implemented. The QuickBuilding tool allows you to generate a BIM building just from one massing solid. It automatically generates different levels, walls, spatial locations, etc. It can be very useful in early design stages, volume studies and conceptual design. The tool will assign BIM properties to each part of the new building.

Spatial Locations can now be used as input for BIMQUICKBUILDING. This workflow allows you to process different buildings simultaneously.

The command also works on solids without edges, for example, a sphere or a torus.



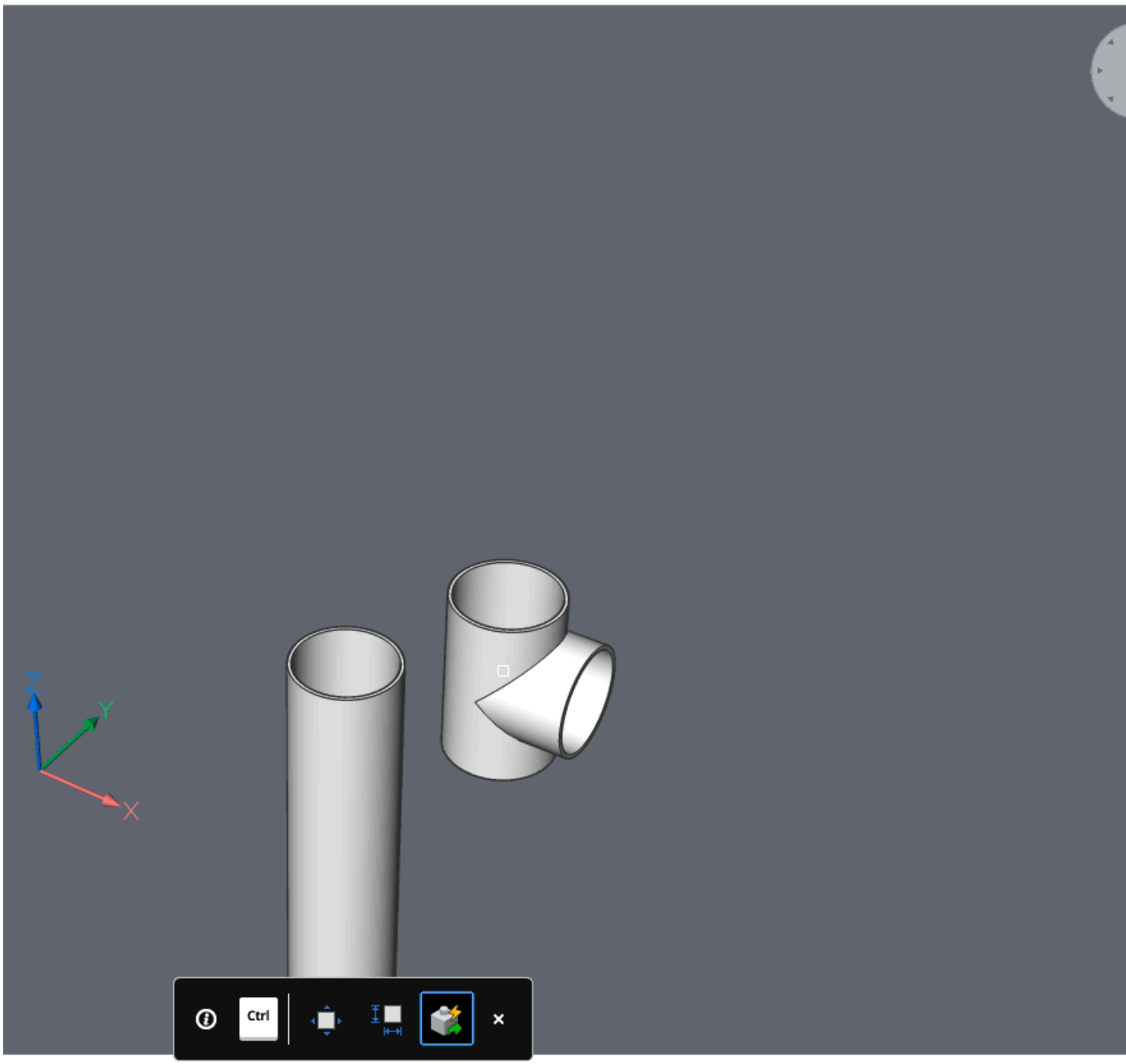
20.1.2.3 Insert BIM Component

The BMINSERT command now includes a SMART insert option, which automatically adjusts parts to match current and relevant parts in the model file when inserted.

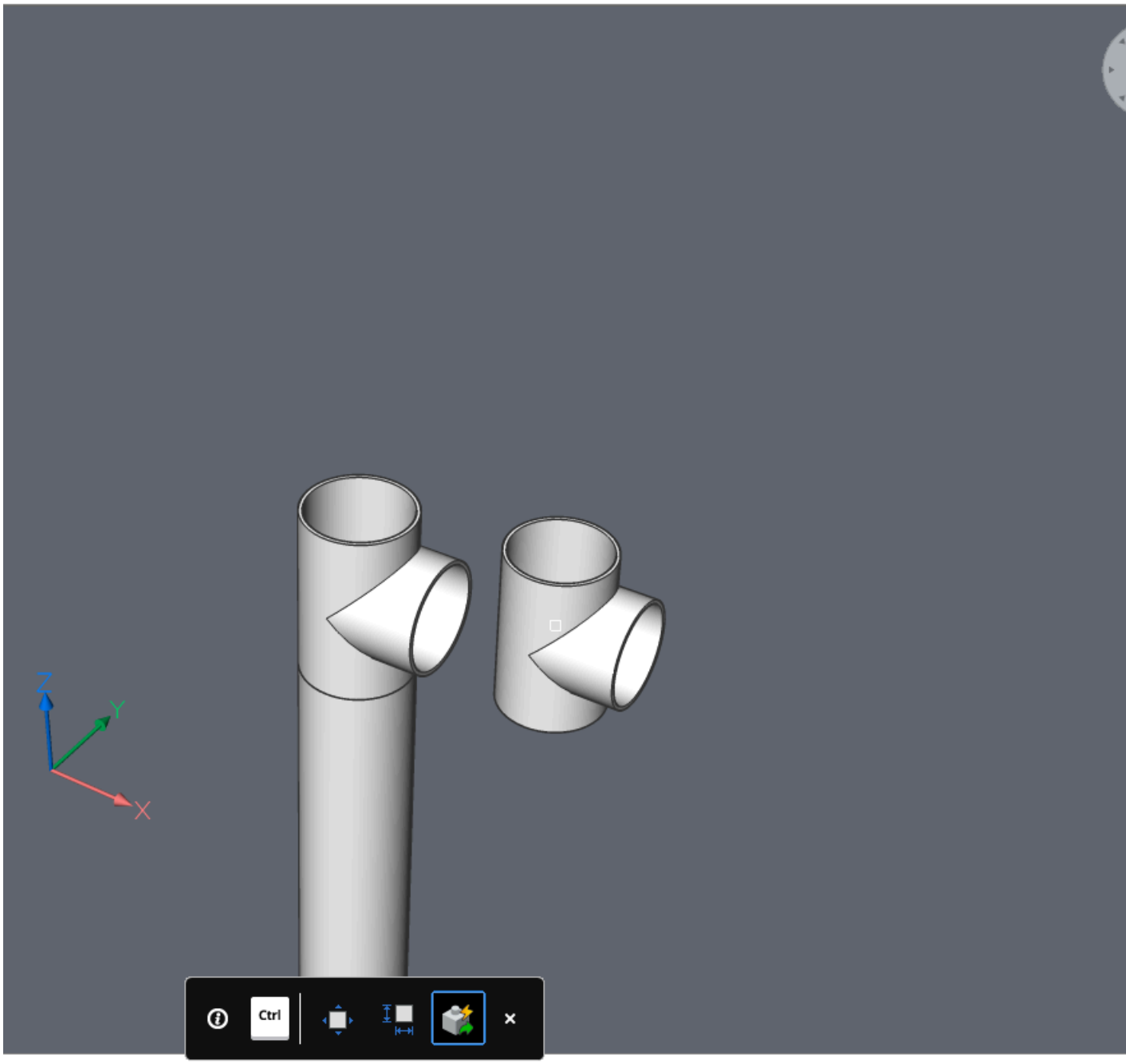
You can select SMART insert by clicking the option in the Command line, or toggle CTRL to the rightmost icon when prompted in the HotKey Assistant.

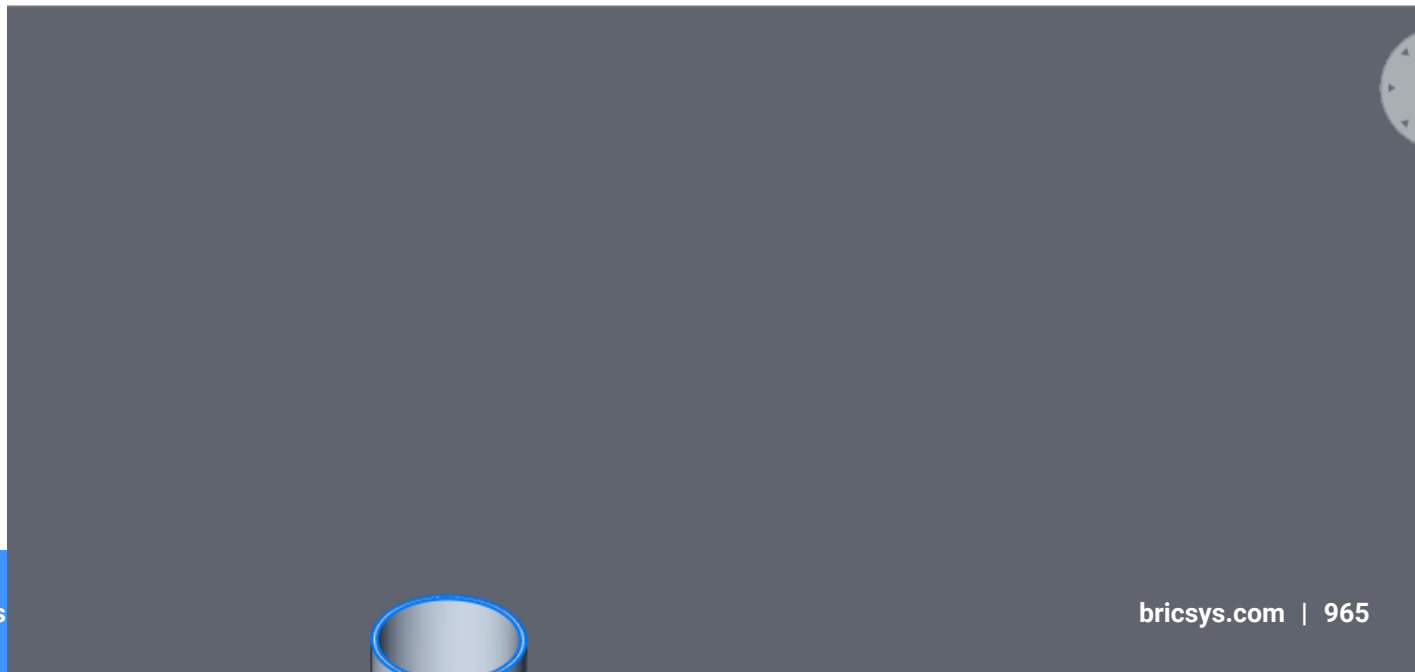


By detecting and creating 3D constraints between the relevant connection points, SMART insert resizes and attaches your desired component onto the current component/s with assembly constraints and common parameters to form a bigger part of a parametric assembly. This is particularly useful in modeling parametric MEP piping systems.


Z
Y
X
Ctrl

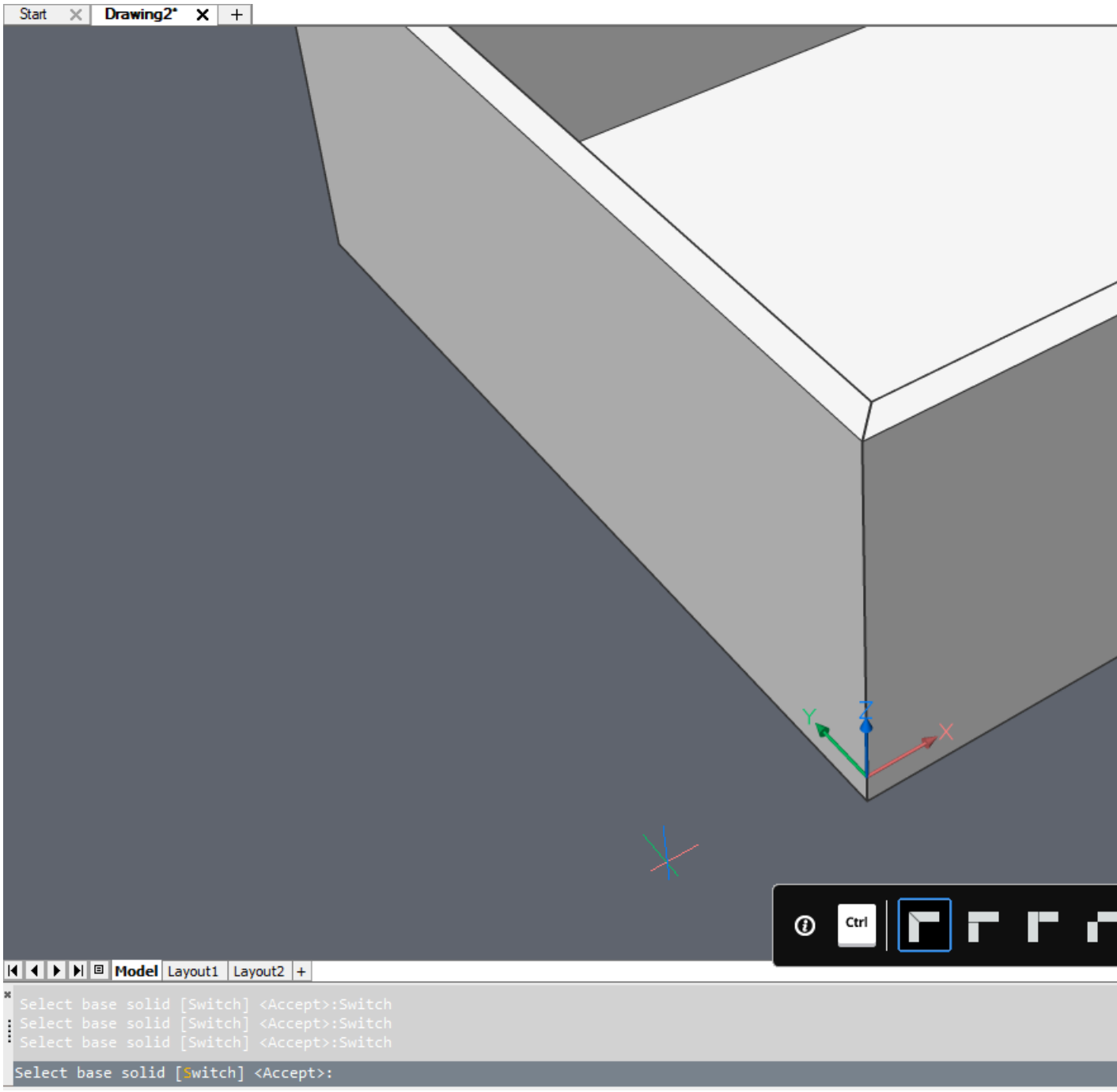


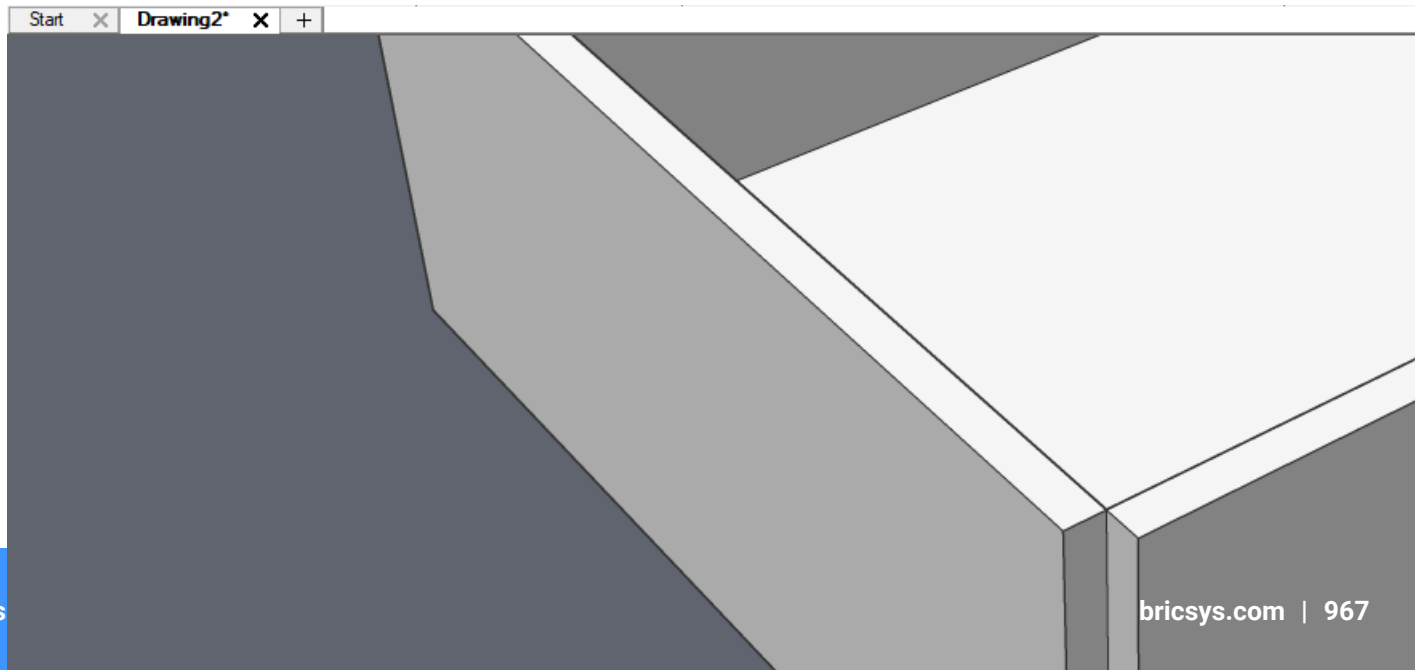

Z
Y
X
Ctrl

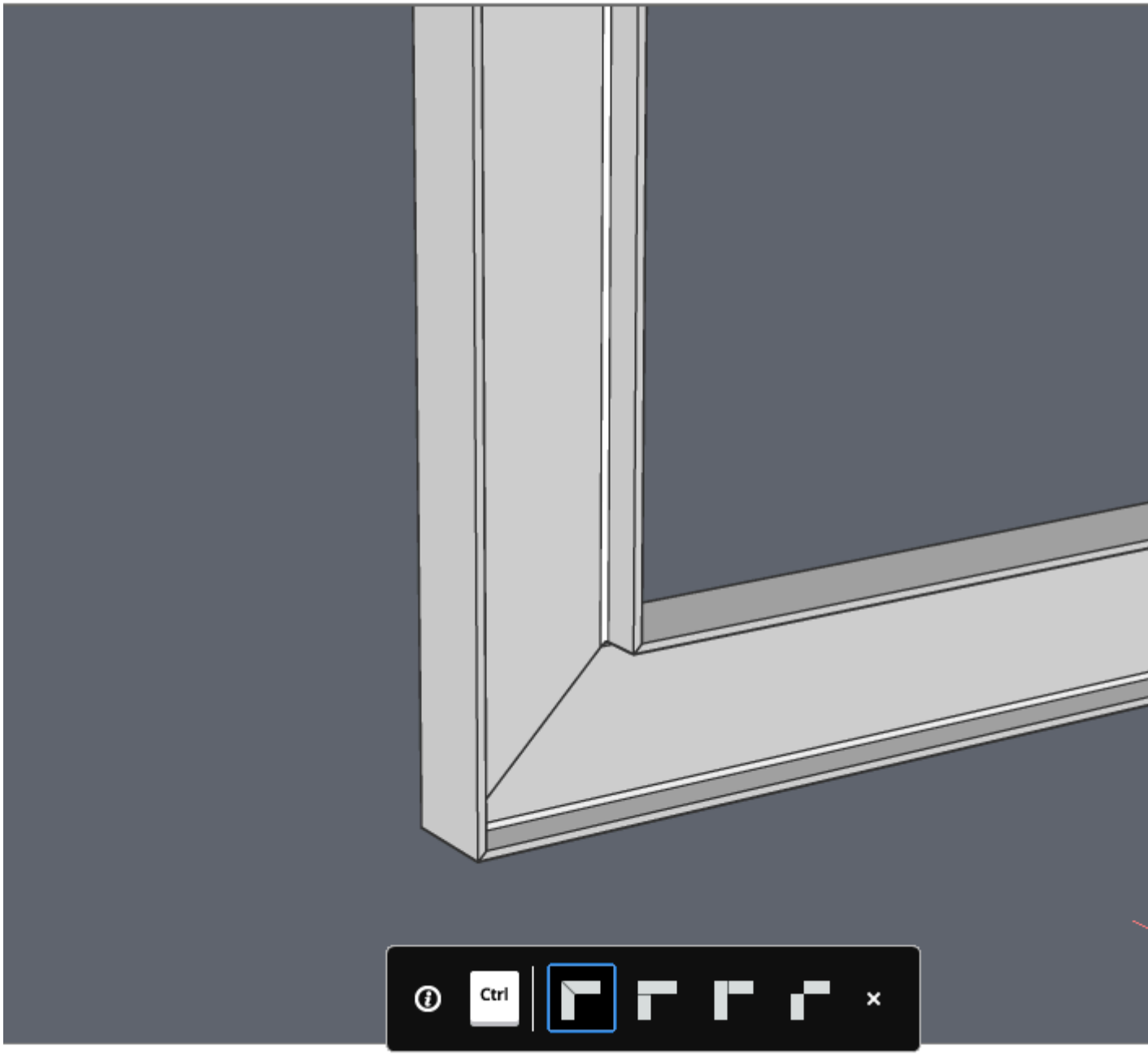


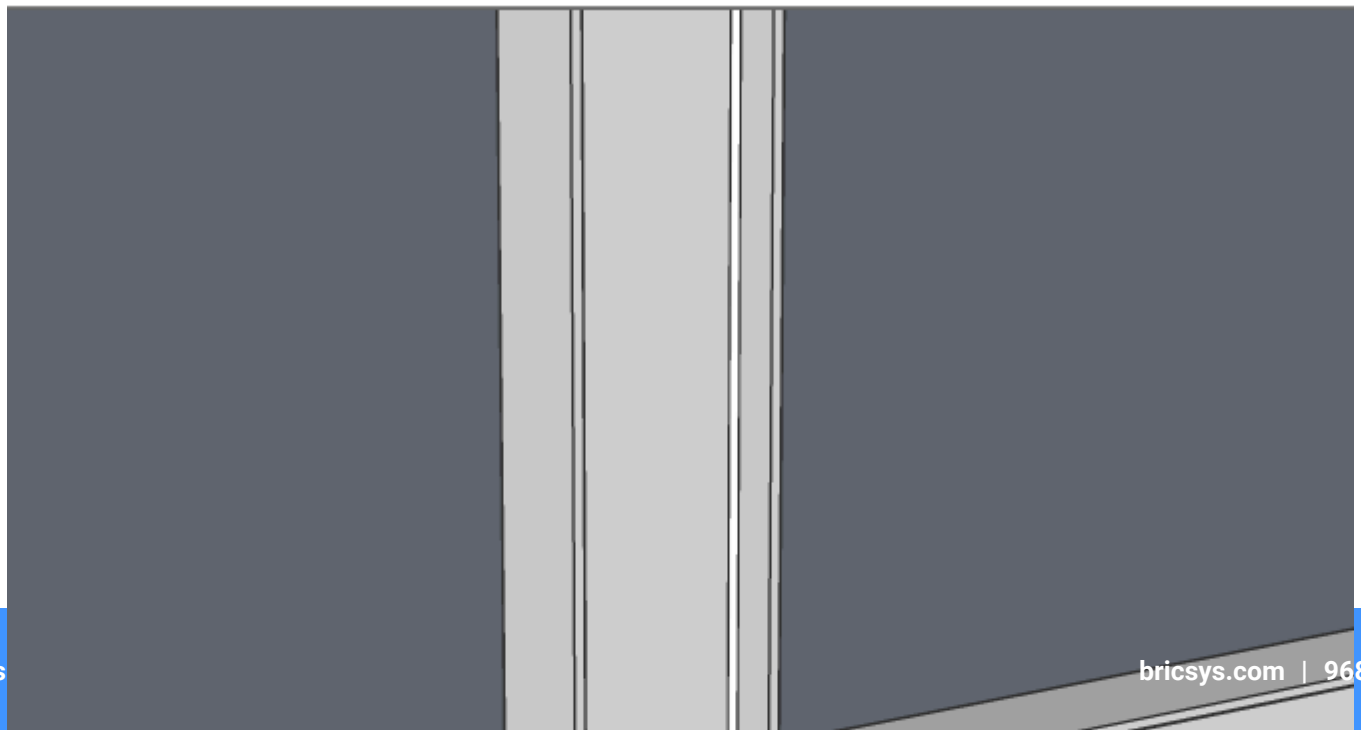
20.1.2.4 Disconnect

The LCONNECT and BIMSTRUCTURALCONNECT commands now include a fourth option: DISCONNECTED. You can cycle through the options with the CTRL button in the HotKey Assistant.


Start
Drawing2*
Y
Z
X
Ctrl
Model
Layout1
Layout2
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:


Start
Drawing2*


Ctrl

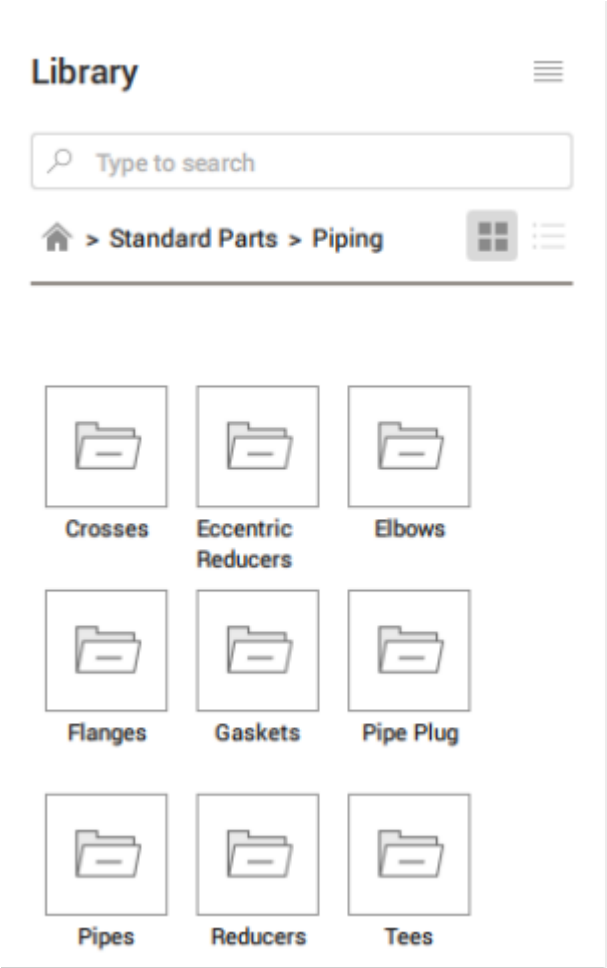


20.1.2.5 Quickdraw

The Quickdraw cursor now supports snapping to (double) lines inside XREFs. Walls now inherit the spatial location of the slab on which they are created.

20.1.2.6 Standards Parts

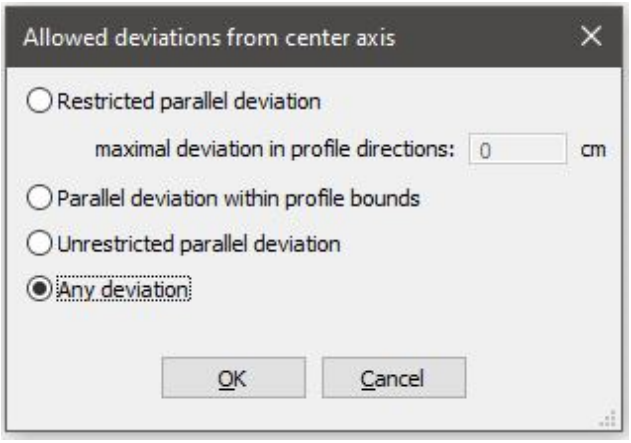
Piping parts were added to the standard parts category of the Library panel.



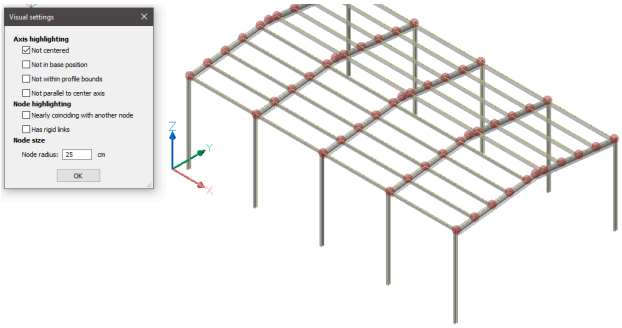
20.1.3 Building data

20.1.3.1 Analytical Model

This command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal, but allows you to further adapt the model to your desire.



The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. You can control the amount of such changes by the deviation settings.



After editing the model, you can export it to an IFC.

20.1.3.2 BIM API

Configurable quantities are now available in the BIM API, as well as the BIM API for Spaces.

20.1.3.3 BIM Compositions

In V21, the render material display and LOD setting have been separated. A new button has been added in the Home/View ribbon panel, next to the LOD button to toggle the rendering composition materials.

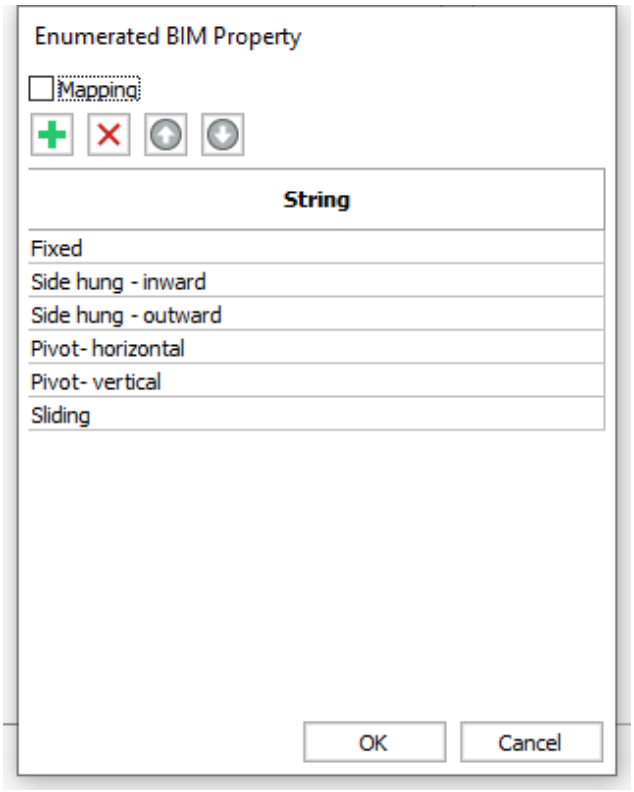


LOD settings control whether ply solids are shown or not. If ply solids are not visible and render composition materials is on, only major faces of the multi ply solids will get render materials.

20.1.3.4 Custom BIM Properties

The Per Instance option of a custom property set allows you to enable/disable user-defined properties, on a per-entity basis.

The Enum option allows you to define a fixed number of possible values for custom BIM properties.



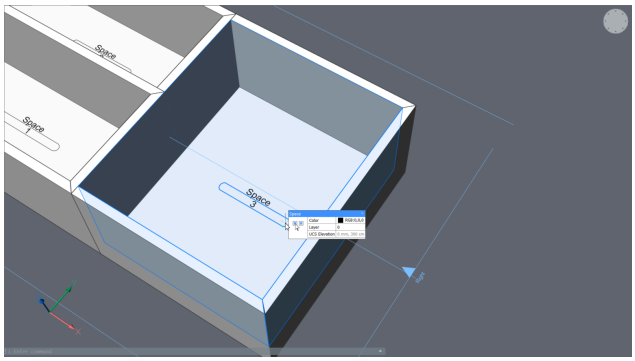
20.1.3.5 Spatial Elements

BimRoom

The BIMROOM command has been replaced by the BIMSPACE command. You can still open BIMROOMs created in V20, but it is not possible to create new BIMROOMs or update the geometry of existing BIMROOMs in BricsCAD BIM V21. Use the Space option of the BIMCLASSIFY command to convert a BIM room to a BIM space. Room properties and interior elevations, if any, are transferred to the corresponding Space.

BimSpace

You can create BIM Spaces entities from enclosed boundaries. When the wall elements enclose a space in your model, you can use the BIMSPACE tool to define a BIM Space. The space footprint preview now displays when executing the BIMSPACE command. These spaces are represented as regular solids so they can be easily manipulated. The Edit option of the BIMSPACE command allows you to add or remove the bounding elements of a space.



BimUpdateSpace

Recalculates a BIM space which is out-of-date, for example after adding or removing bounding entities. In the Properties panel, the Update Method property is added with the options Automatic and Manual.

The new Representation property allows toggling between Solid or Footprint representation of a space.

The Space tag has been added. The Space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. You can change the values of each attribute using the Properties panel.

The new Status property indicates whether a space is up-to-date or out-of-date.

BIM	
Type	Space
Spatial Element Type	Space
Representation	Footprint
Update method	Manual
Status	Up to date
Name	Office
Description	Standard office
Building	Building
Story	Floor 0
Composition	
GUID	1zcNYneH56hRdMV23U7Iw$
Number	2
Interior Elevations	Off

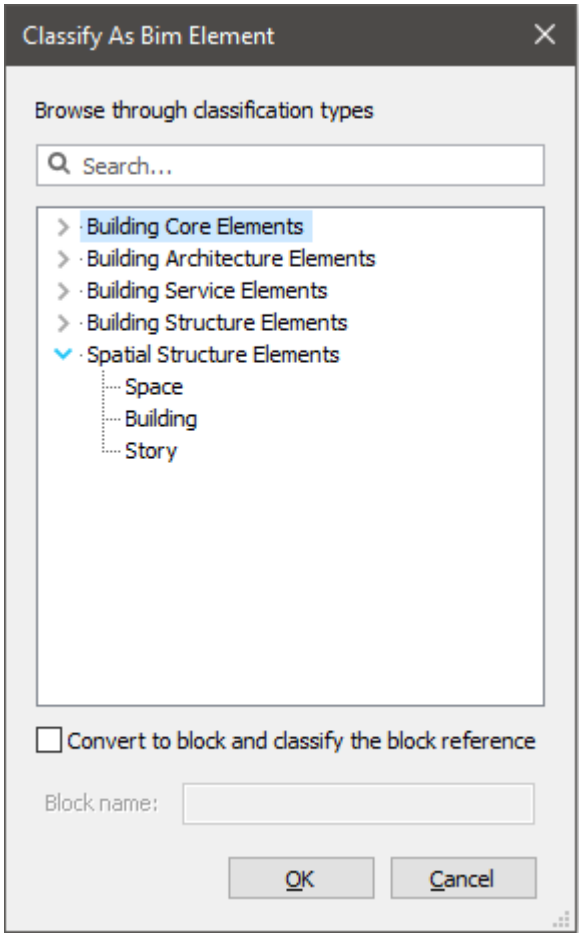
Space icons

BIM Space tool icons for Edit Space, Update Space and Define Interior Elevations are available in the Quad, the Ribbon and the BIM Spaces toolbar.



Spatial Structure Elements

When using the BIMCLASSIFY command, a new category of classification has been added to the dialog box. The name of the new category is Spatial Structure Elements. Under this category you will find Space, Building and Story.

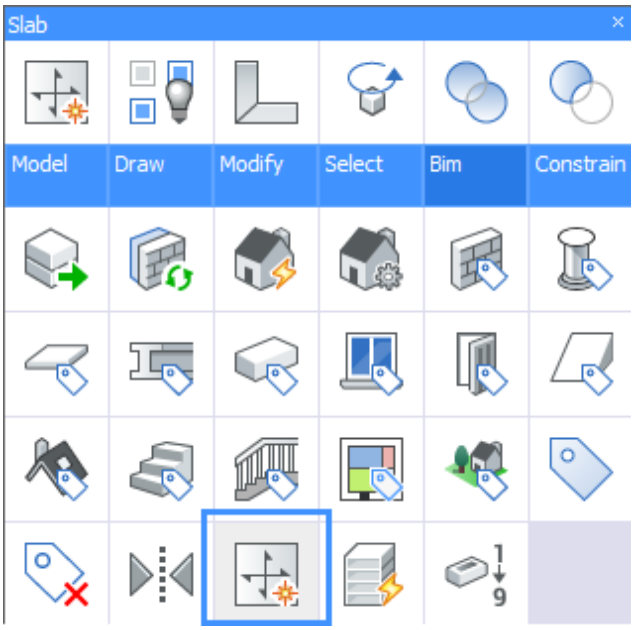


DefaultSpaceHeight

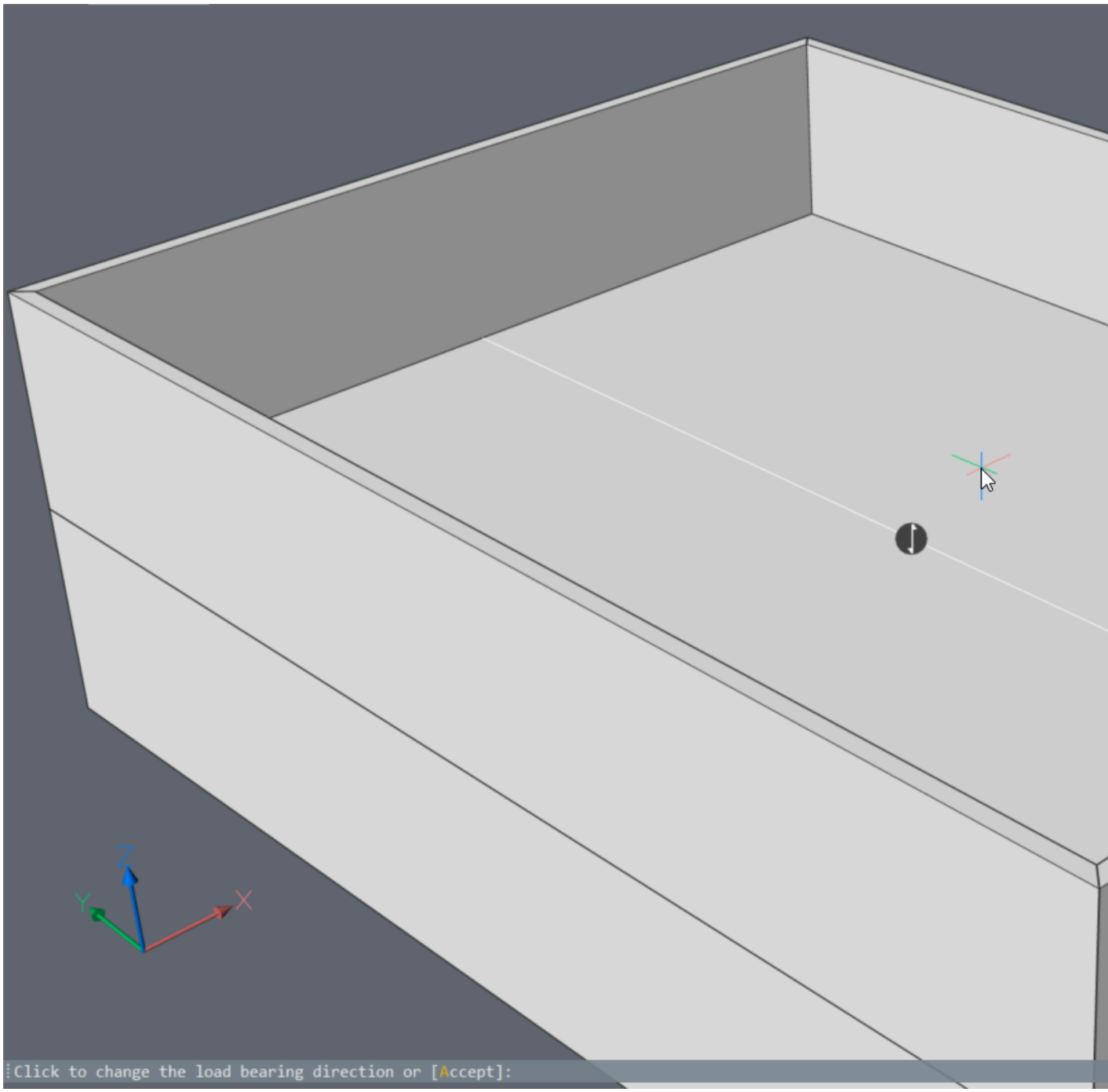
The ROOMHEIGHT system variable is replaced by the DefaultSpaceHeight user preference, the setting applies to Spaces.

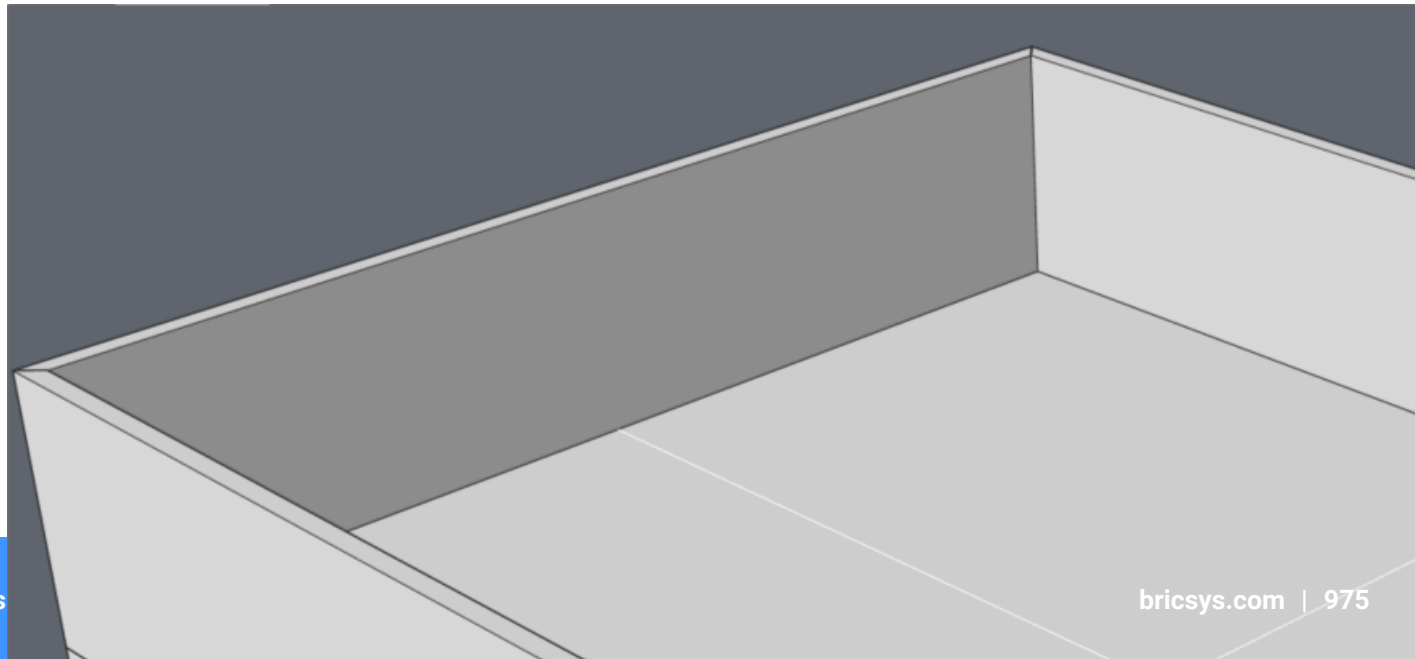
20.1.3.6 BIMSETLOADBEARINGDIRECTION

With the new tool button that launches the BIMSETLOADBEARINGDIRECTION command in V21, it is easy to set your loadbearing direction of your slab through the Quad.



As a result, you will get a black dot on your slab. Once you click on it, you can change the load-bearing direction of the slab in one of the two main directions of the slab or in both.


Z
Y
X
Click to change the load bearing direction or [Accept]:



20.1.3.7 Python scripts in BricsCAD BIM

Nota: This is a V21 Beta feature.

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a .py python script.

The python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

Executing BIMPYTHON opens a Windows dialog box where you can navigate and run your python script in .py format.

The new BIMACTIVATEPYTHON command allows you to activate and deactivate the virtual python environment in BricsCAD BIM, by toggling the values 0 and 1 for **OFF** and **ON** respectively. You need to ensure it is turned **ON** before using BIMPYTHON.

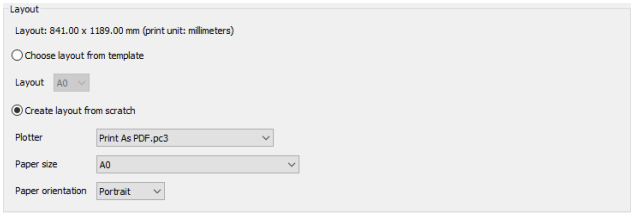
The Python Programming Language, together with its standard libraries, is embedded in BricsCAD BIM so there is no need for you to install them separately, unless you have custom packages and libraries which you wish to use in your scripts.

20.1.3.8 Project Browser

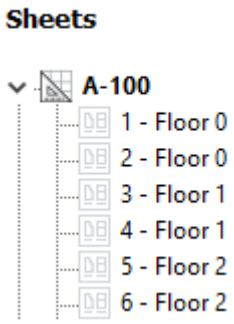
BIM project data are now stored in an external project **.bsyslib**, instead of the previously used **.projectroot**. When opening a pre-V21 BIM project, you can convert the **.projectroot** to a V21 project **.bsyslib**. To simplify the opening of a BIM project at a new location, paths to **.bsyslib** files are now stored with a relative path.

Sheets

It is now possible to create a layout from scratch in the Sheetset setup dialog box.



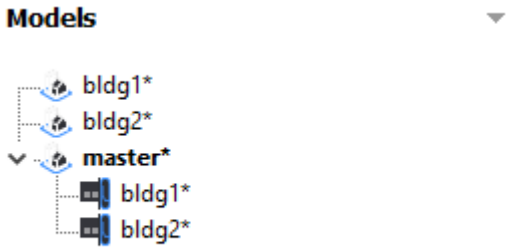
The Project Browser displays the Sheet Number (if it exists). The Sheet Number is assigned by the **Create Sheets** procedure. Sheet View Numbers are assigned starting from 1 for all section results created by the BIMSECTIONUPDATE command.



Models

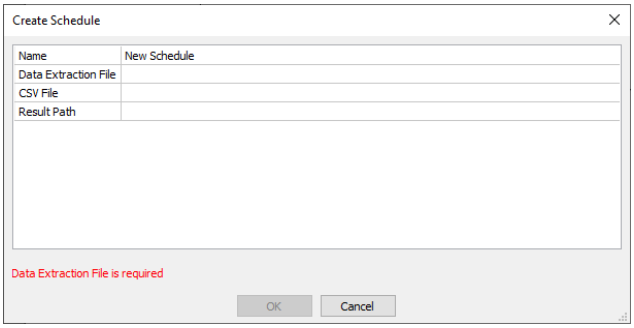
Master drawings and external references are listed in a tree-structured manner in the Models tab of the Project Browser:

A balloon is available to fix a project libraries' connection to a drawing sheet.

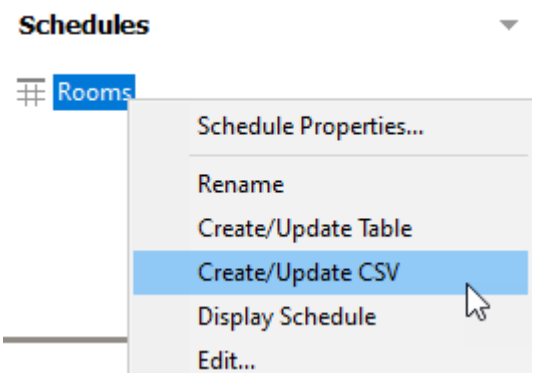


Schedules

The **Add Schedule** wizard, available in a context menu, allows you to quickly add a BIM schedule without the creation of a new .dxd file. The use of relative paths is now supported for Schedules.



The **Create/Update Table** and **Create/Update CSV** options are available in a context menu for creating or updating a schedule or a CSV output file.



20.1.4 Project collaboration

20.1.4.1 IFC IMPORT/EXPORT

A mechanism has been implemented to map IFC/BIM entities to BIM/IFC entities, to control which entities are imported/exported. BricsCAD BIM also added IFC Export of Annotation entities.

You can do this mapping by using mapping files. The IfcImportMappingMath and IfcExportMappingPath user preferences set the paths to the necessary mapping files. These files are simple text files containing some rules.

This mapping can be used to export BIM entities as IFC entities of another type, or to skip certain types altogether. An example of such an export file can be:

- BimDbColumn IfcMember
- BimDbBeam IfcMember
- BimDbFurnishingElement skip

In this example, columns and beams in the BricsCAD BIM model will be exported as Members in the IFC file, and furnishing elements will be skipped in the export process.

A similar process can be used when importing IFC files. An example of such an import file can be:

- IfcWall BimDbSlab
- IfcSlab BimDbWall

* skip

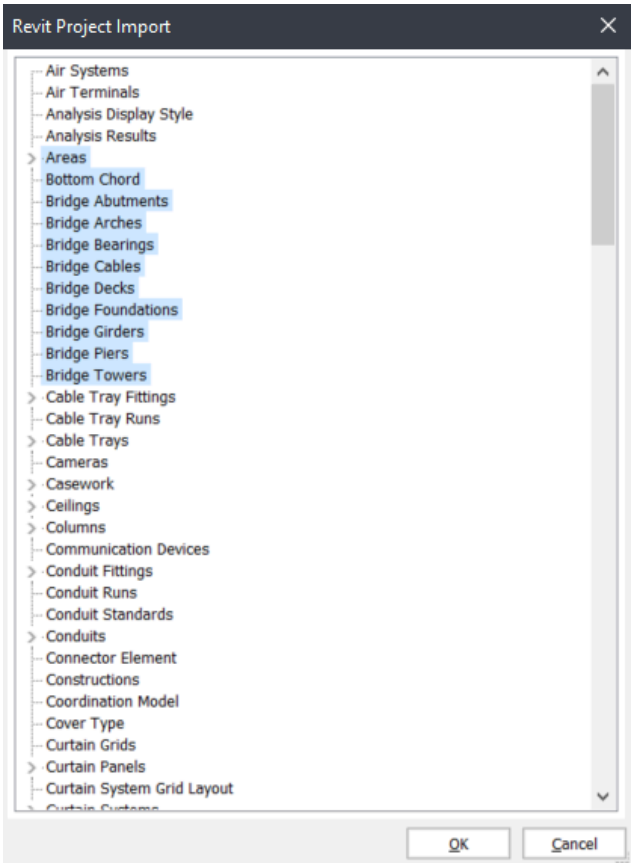
In this case, walls will be imported as slabs and slabs will be imported as walls. No other entities will be imported, as the asterisk means that only listed entity types will be taken into account, all unlisted entity types will be skipped.

The IfcCreateUniqueGuid user preference has been added to enable/disable the creation of unique GUIDs for nested elements inside classified blocks and Xrefs.

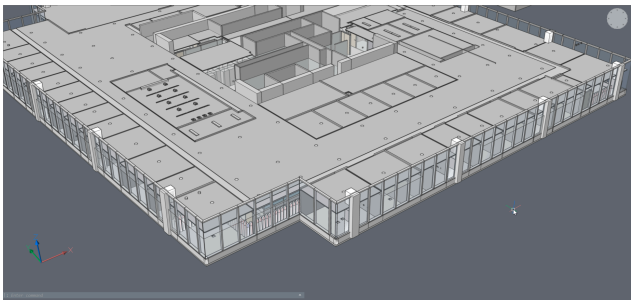
The flow connection point is exported to IFC as IfcBuildingElementProxy. Only 2D geometry or a block reference with 2D geometry can be classified as a flow connection point.

20.1.4.2 Importing RVT files

When importing Revit Objects using IMPORT, you can choose for Import all the solids or only import as an Overlay. You can filter the import based on Revit Views, Categories, Levels or WorkSets.



This is an example for the option Categories, where you can map specific categories for your RVT import. Colors and transparencies are now imported when importing a RVT file.



20.1.5 Design documentation

20.1.5.1 Sections

Activate SectionView

This new command allows you to look at the model from a section plane's perspective, without generating the section. This view allows you to manipulate the 3D model from a 2D point of view. To exit the active view, just orbit around the model.

Open BimSections

When switching to the model from a sheet view using the BIMSECTIONOPEN command, the model is rotated and zoomed in to the same zoom level as the sheet view.

Available properties and property values display in the filter parameters form.

The BIMSECTIONOPEN command now allows you to transition between the sheet viewport and drawing model seamlessly at the same zoom level, without the need to re-orientate yourself and the model to where you have been working earlier on the sheet.

When you switch between a sheet viewport to the drawing model using the updated BIMSECTIONOPEN command, the drawing model displays and orientates itself to the same zoom level as in the sheet viewport, regardless of the scale of the Standard Scale or Annotation Scale of the drawing or viewport.

Update BimSections

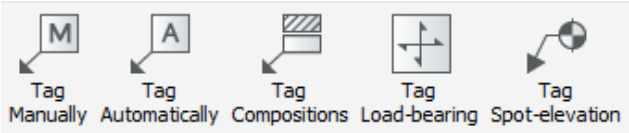
The associative data for 3D solids, needed for section generation and tagging of the section results by the BIMTAG command, are generated automatically in BricsCAD BIM V21 as you model, unless the new GENERATEASSOCATTRS (Generate Associative Attributes) system variable is set to Off.

Interior Elevations

Since Rooms in V20 and older are now deprecated and replaced by Spaces, the Interior option of the BIMSECTION command now supports Spaces.

20.1.5.2 Tags

The BIM Tags tool now includes options of tagging load-bearing directions and spot elevations on viewports on Paper Space. Both tag types work on plan and section drawings. Spot elevation tags also work on elevation drawings.



Automatically generated tags can have an offset, which is defined through the Offset attribute in TagTypeToStyle.xml found in the **SUPPORTFOLDER** > **Bim** > **Sections** folder. If the offset is not specified, the center of the tag block will be used.

Similarly, automatic and manually placed tags can be rotated if the autoRotation attribute is set to **true** in TagTypeToStyle.xml.

Tag Load-bearing

Load-bearing tags is only applicable if the load-bearing direction of the slab has already been set. Refer to the BIMSETLOADBEARINGDIRECTION command for more information.

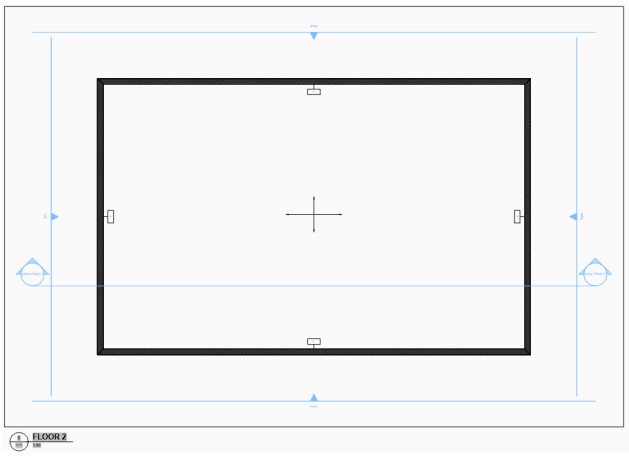
Tag Spot-elevation

The spot elevation level tag displays the actual elevation of a selected point of a BIM object in a section view. The height is measured relative to level 0.

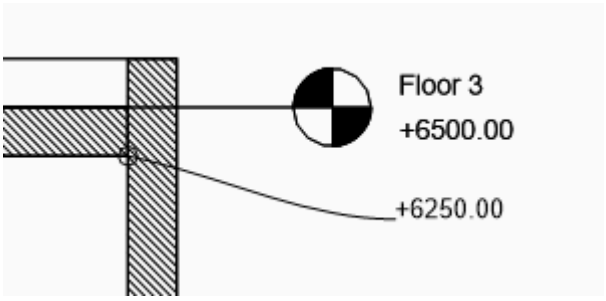
Tags in general

On Paper Space, click **Tag Load-bearing** from the ribbon and pick point on a sectioned BIM element as prompted. You will be prompted to pick a point on your desired sectioned BIM element or to change the current mleader style settings, like how you set BIMTAG in V20.

For example, you can have a two-way load-bearing span graphic tag on structural drawing plans.



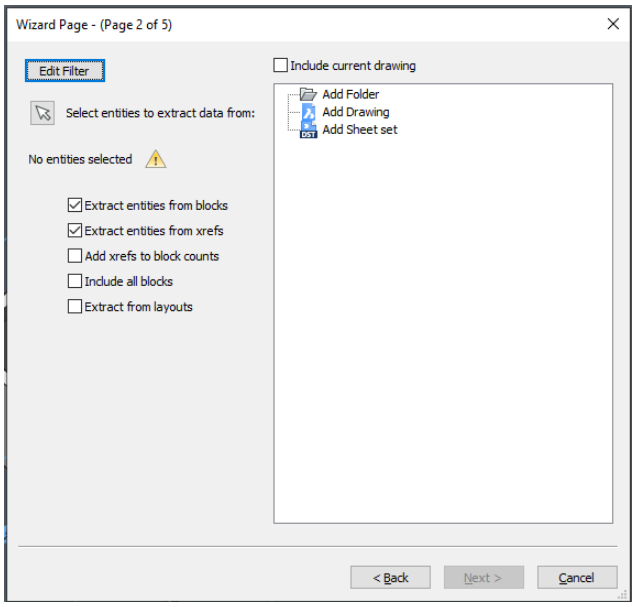
Or you can have spot elevations displaying elevation levels relative to the WCS.



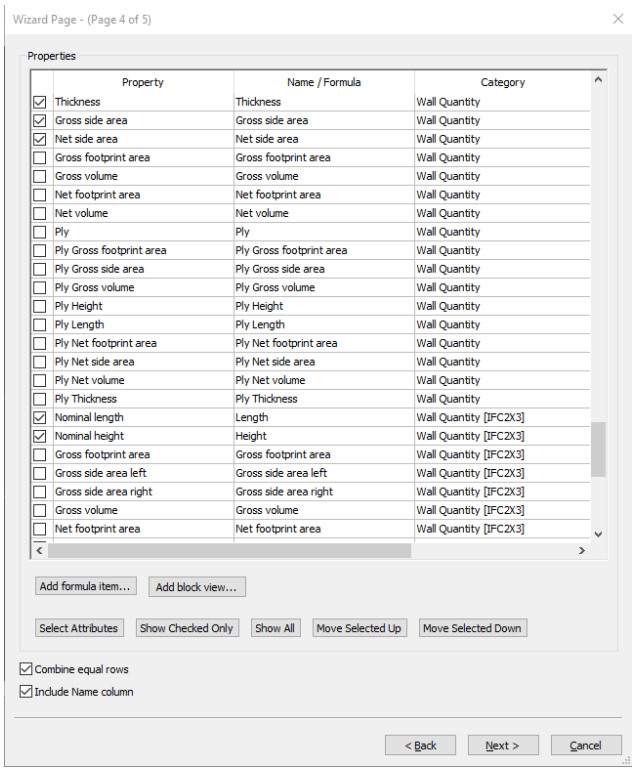
Like all other BIM tags, they are customizable multileader blocks. Therefore, for example, Spot Elevation tags can be further adjusted to suit your graphic and information preferences.

20.1.5.3 Data Extraction

The DATAEXTRACTION command now allows you to include all blocks and also extract from layouts in the new filter rule settings. You can now extract composition and physical material data.



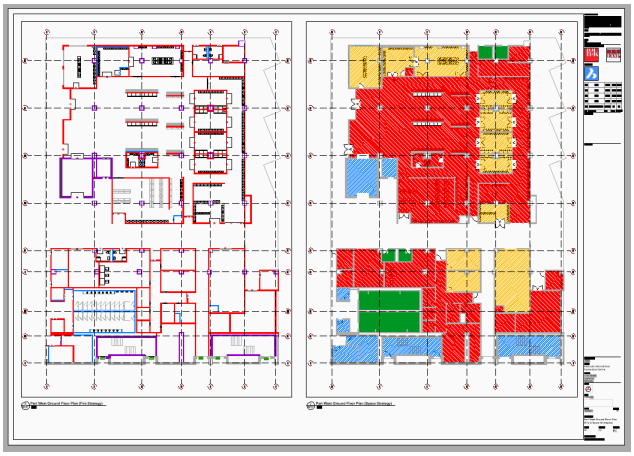
You can now extract and schedule quantity data based on IFC2x3 Element Types.



20.1.5.4 Drawing Customizations

Drawing Customization is a new tool that is available and only accessible from the panel. All the controls of Drawing Customizations are represented as graphic icons within the User Interface panel, such as the **+** symbol in a blue circle to create a new template. You can duplicate, rename or delete your customization templates.

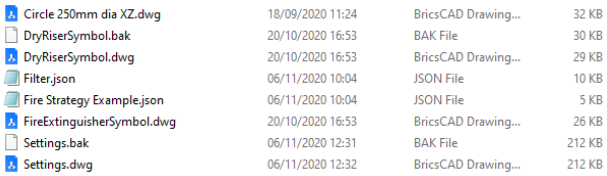
It allows you to create view templates to control and customize the visibility settings of your drawings on layouts. The Drawing Customizations tool in BricsCAD is operated by 3 principles – Entity Customizations, Styles and Filter Rules.



Drawing Customization templates are stored within the Customization folder located in the SUPPORTFOLDER, under **Bim folder** > **Sections subfolder**.

Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the Create New buttons.

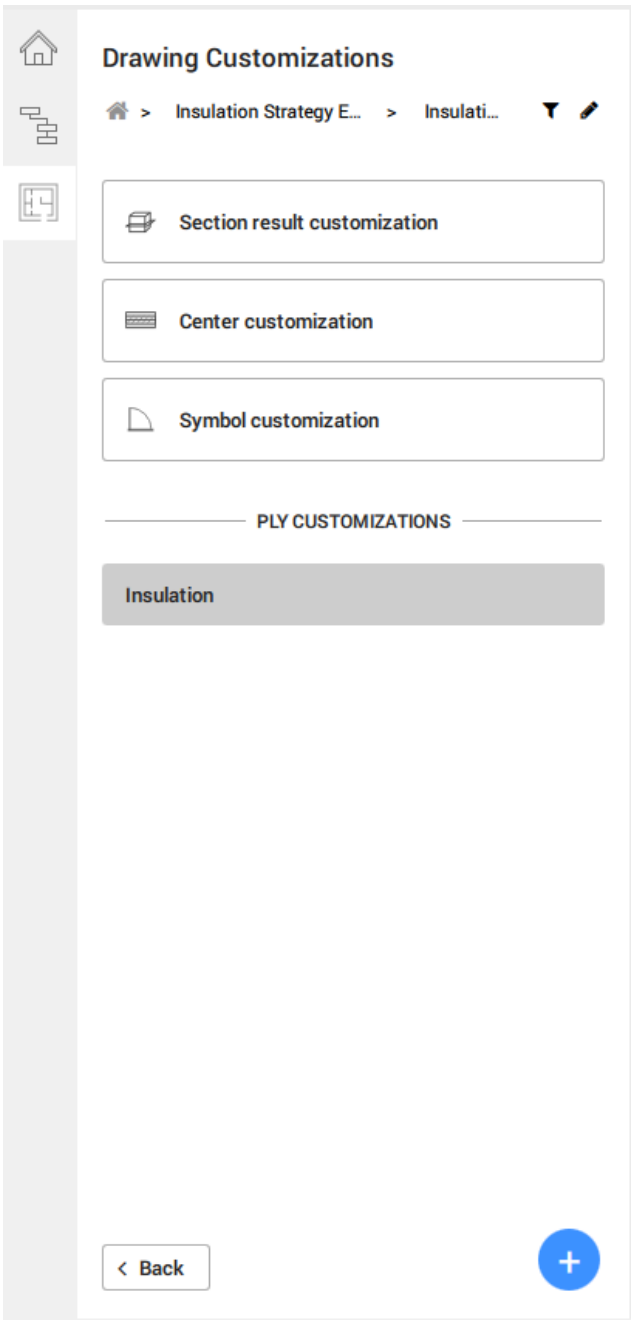
Settings.dwg contains the values created in the Style tab in the form of layers and if you have created any custom properties that you wish to use as a Filter parameter, you have to ensure they are also reflected in the Settings.dwg.



Entity Customizations

Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

One level deeper within one of those entity customization tabs brings you to its settings tab, where you can choose how the element should look like as a section result, or how a center line type representation should overlap, or how an external symbol representation should replace the original asset. You can further modify and customize a specific ply layer of your desired entity composition, but only if appropriate.



The section result customization displays a series of controls with drop-down values from the Styles tab for more specific control of the section result graphics. You can override the entities' hatches via the Appearance Override with preset or custom Physical Materials available in your project.

For the Symbol Customization settings, you can find an input dialog to specify your desired external symbol drawing in .dwg format. There are additional buttons to allow you to edit the selected symbol drawing or to create a new .dwg symbol.

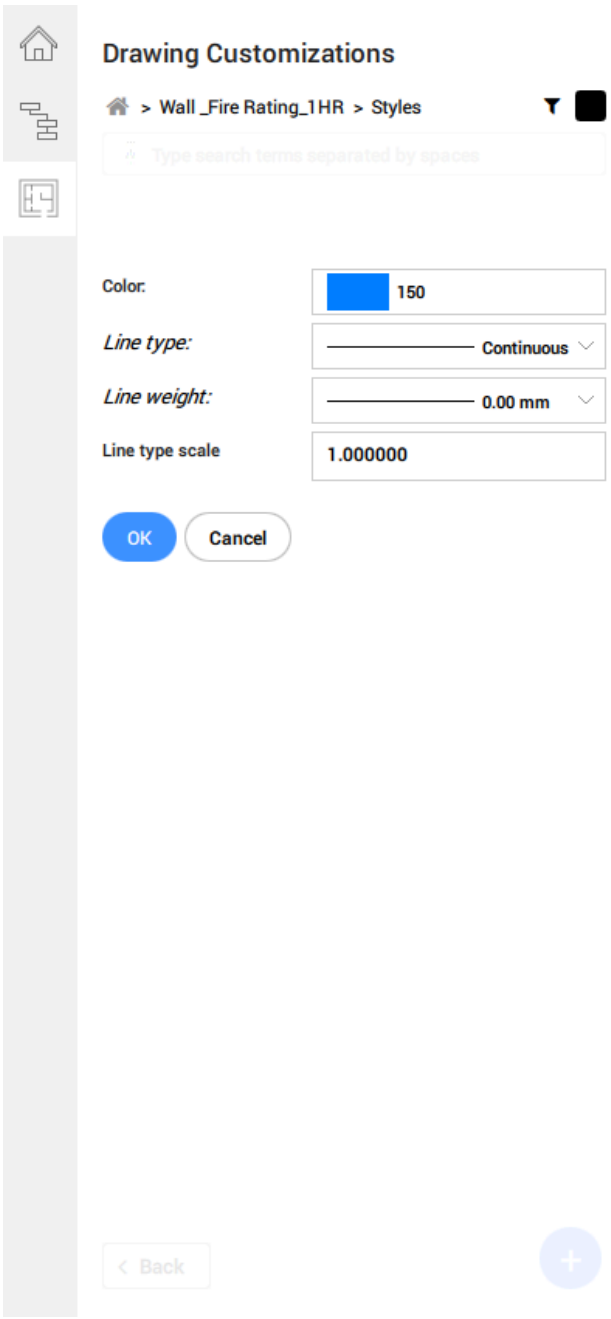
You can further apply your saved Styles to specific layers within your external symbol source, thereby allowing you to retain a general symbol drawing across several customization templates with different output results.

Styles

This is where your desired style options like Lineweight, line colors, etc, are stored and will appear as drop-down options in the Entity Customizations and Center Customizations settings as shown in the previous section above.

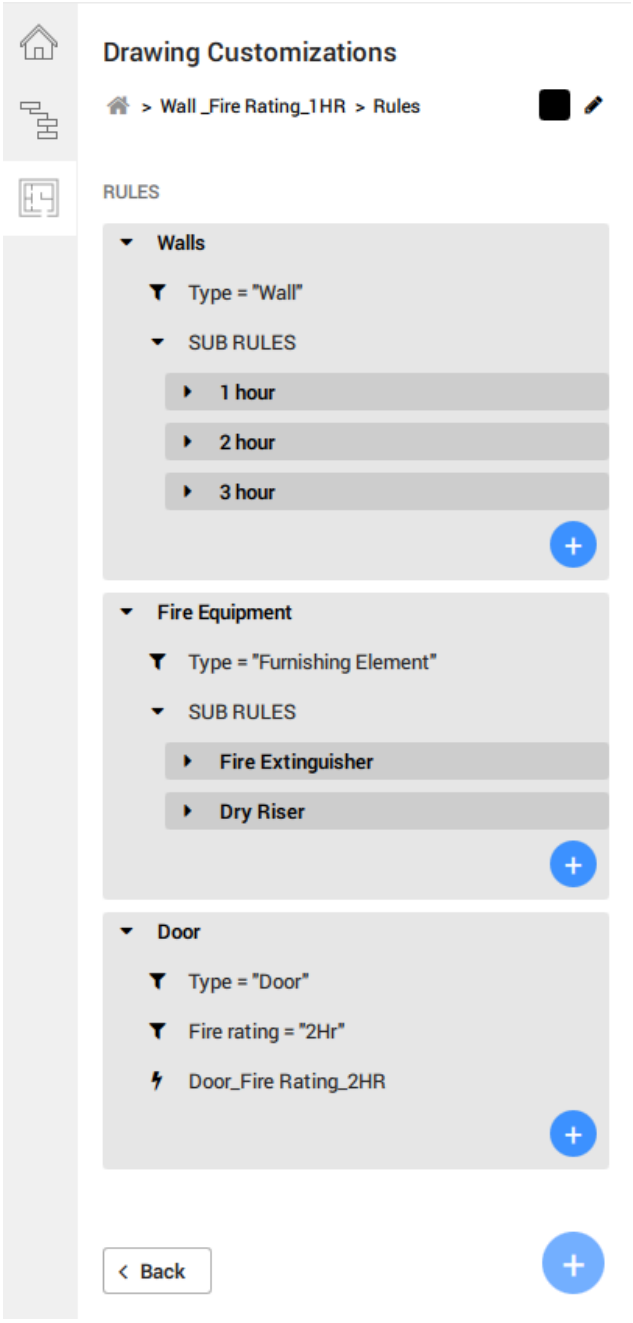
The operation of the Styles tabs is like how Entity Customization tabs can be controlled.

To edit any of the styles available, click the relevant tab to enter the styles settings, where you can specify your desired values for the style's color, linetype, lineweight and linetype scale.

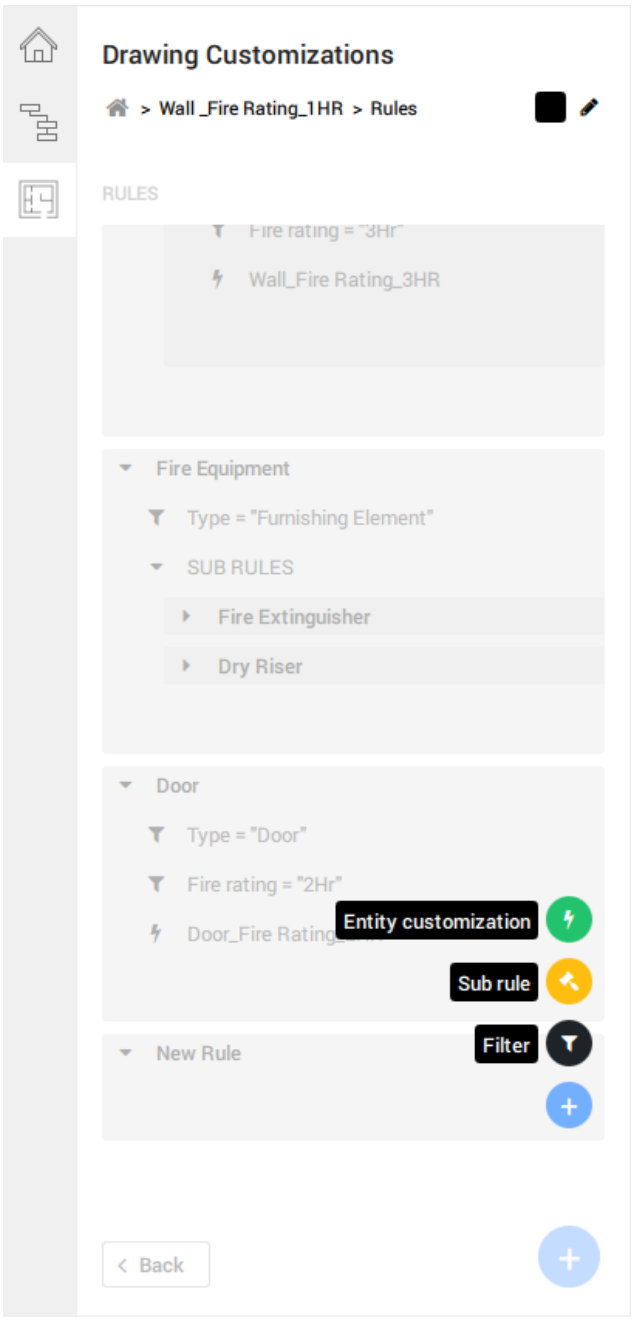


Filter Rules

To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the Filter Rules tab.



Filter rules can be further classed into sub-rules, which falls as a sub-set of the parent rule, ie. Fire Equipment as a Filter Rule, whereas Fire Extinguisher is classed as a sub-rule. The blue **+** symbol also appears respectively and accordingly at each rule level, where at the end of each rule, regardless of it being a subset or otherwise, has to end with an Entity Customization.



Adding and editing a filter parameter will lead you to a prompt, where the options of filter property is determined by the BIMPROPERTIES imported in the namespace settings.

To end and legitimize a filter rule, the entity customization value must be added and filled in. The options available in the drop-down selection reflects the existing customizations tab available in the Entity Customization page.

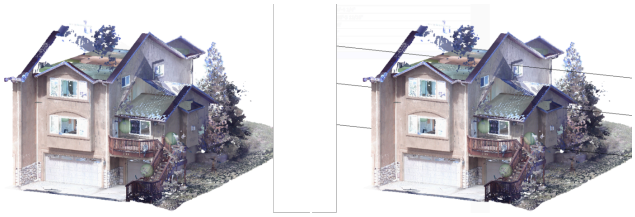
20.1.6 Point clouds

Point cloud functionality is even more powerful with a BIM license.

20.1.6.1 Floor Detection

Sectionplanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and sectionplanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

The new POINTCLOUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

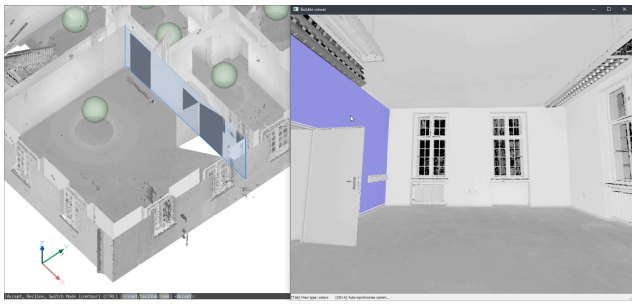


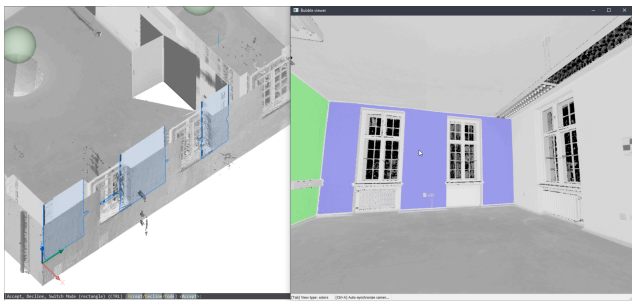
20.1.6.2 Planar Fit

The new POINTCLOUDFITPLANAR command enables you to create 3d geometry based on the point cloud. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

In bubble view

If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL key.

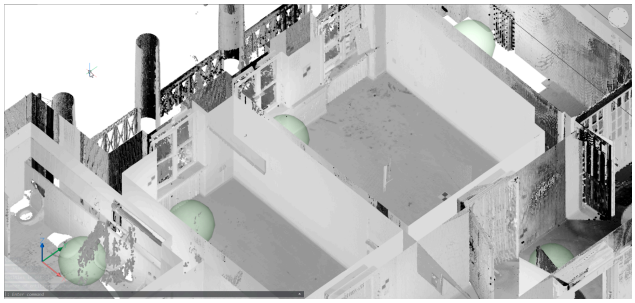


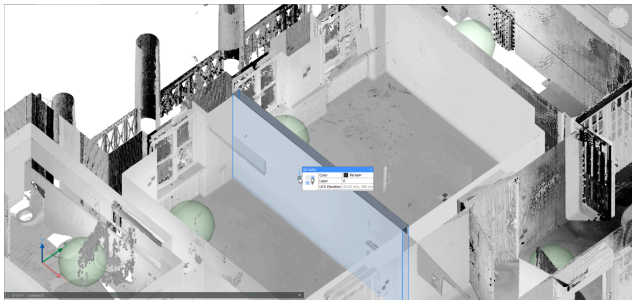


In model space

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time but it has 2 advantages by searching multiple scan positions:

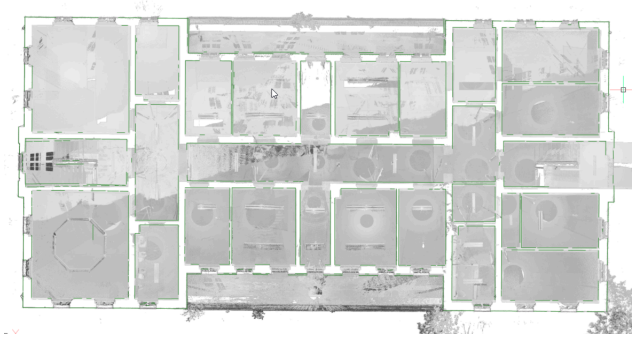
- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.





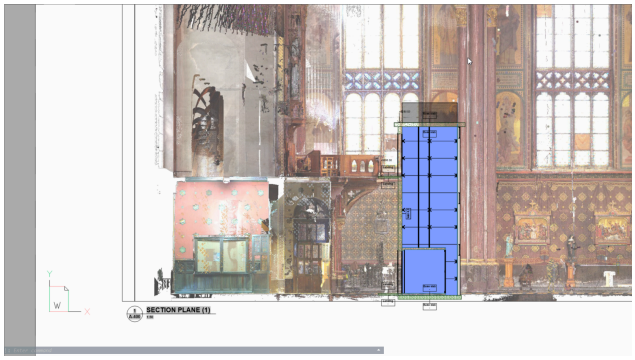
20.1.6.3 Point cloud projection

The new POINTCLOUDPROJECTSECTION command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building. You can use these sections to generate 2d lines to create a 2d floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



At the same time, a raster image will be generated. In some cases it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2d geometry but in high quality scans these images can also be used

as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.



20.2 Começar

20.2.1 Creating a BIM model

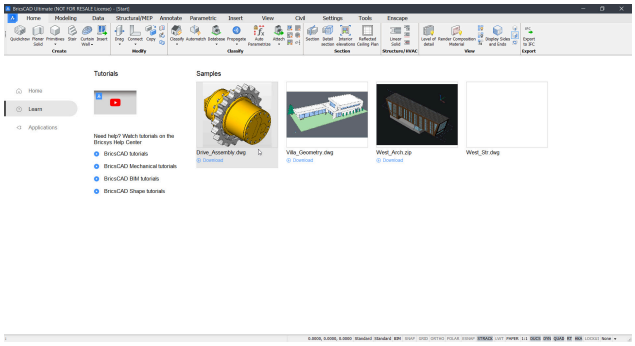
Do the following:

1 Start BricsCAD and choose the BIM workspace on the **BricsCAD Launcher**. In this workspace, all BIM tools and resources are available in the Ribbon, the Quad and various dockable panels.

2 On the Start tab, select a **BIM** template.

3 Click the **New Drawing** button to create the model.

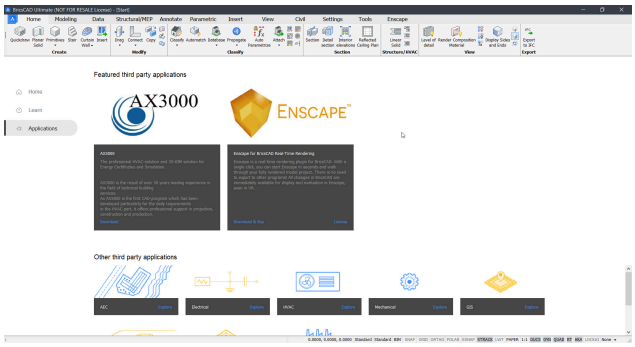
20.2.1.1 Learn



Here you will find links to tutorials for Shape, Core, BIM and Mechanical. You can also find some sample files that you can download and try out yourself.

20.2.1.2 Applications

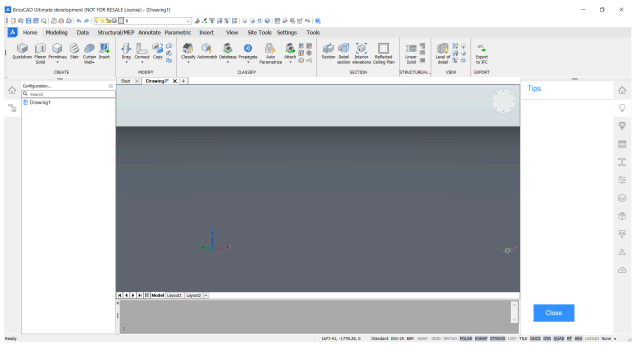
Here you can find some of the third party applications that are available for BricsCAD. You can find more applications on our Applications Store website.



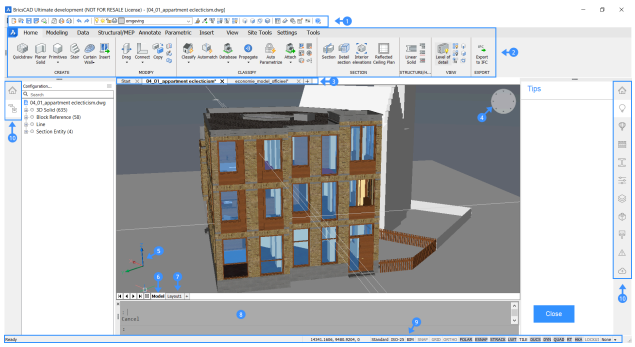
20.2.2 Exploring the interface

20.2.2.1 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.



20.2.2.2 Interface elements

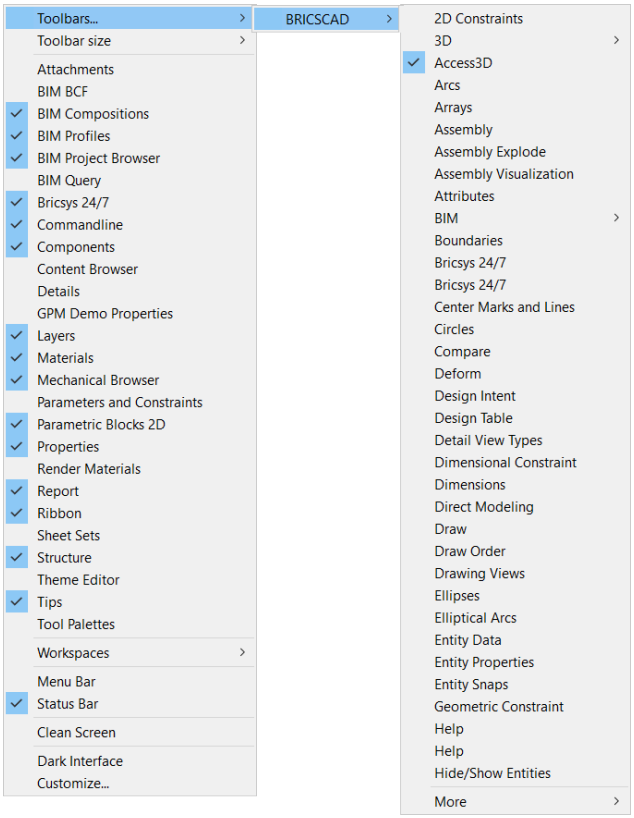


1. Toolbar

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 tool categories available 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, the Access3D Toolbar is shown on top of the screen.

Nota: To open more toolbars you can right-click, choose toolbars and check the topics you want.

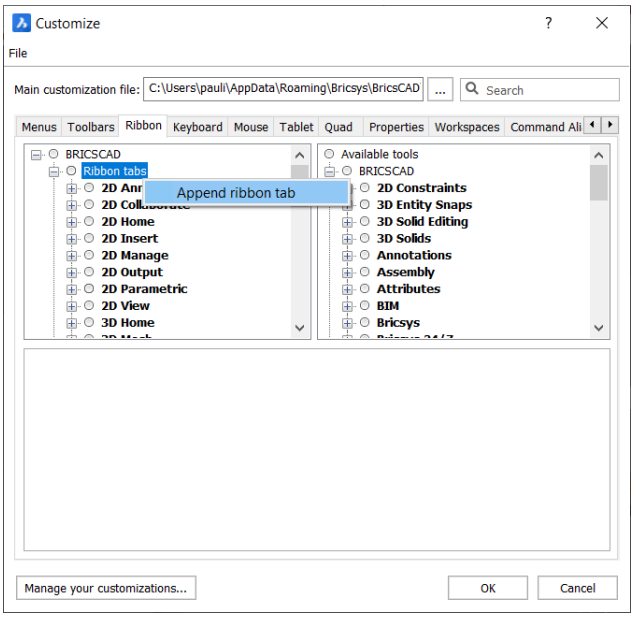
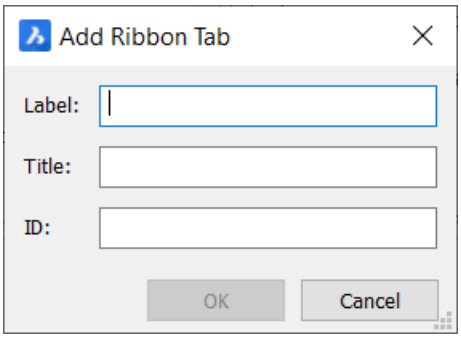


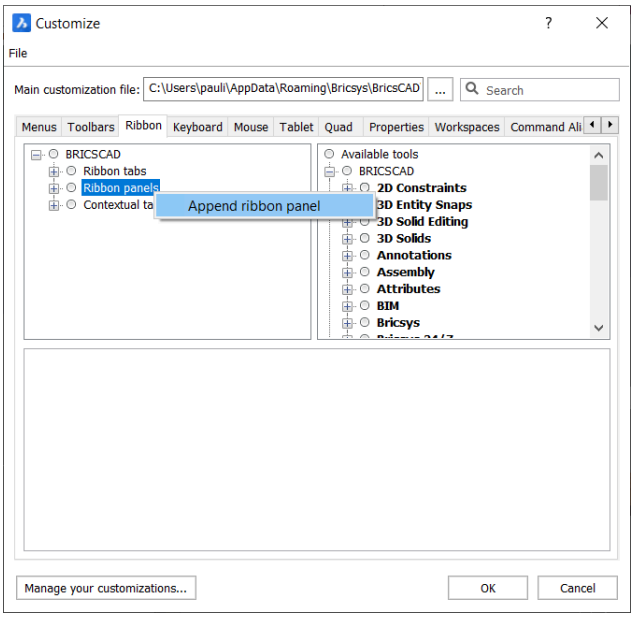
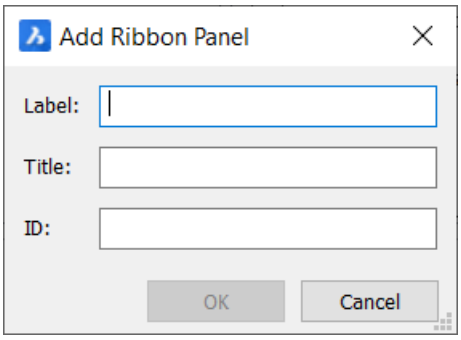
2. BIM Ribbon

A ribbon organizes tools in a series of panels (11) which are grouped in different tabs (12). Each tab contains panels and each panel contains a group of buttons and flyouts.

Nota: The content of the ribbon depends on the current workspace (see Editing In Workspace).

Nota: Use the RIBBON and RIBBONCLOSE commands to show or close the ribbon.

Creating a ribbon tab	Open the **Customize** Dialog box and click on the **Ribbon** tab. Right-click the **Ribbon tabs** group and choose **Append ribbon tab** in the context menu.  Fill out the fields in the **Add Ribbon Tab** dialog box:  **Label**: Caption on the ribbon tab. **Caption**: Used in the Select Ribbon Tab dialog when selecting ribbon tabs for a workspace. **ID**: A unique name to identify the ribbon tab.
Editing a ribbon tab	Open the **Customize** Dialog box and click on the **Ribbon** tab. Expand the **Ribbon tabs** group and you can edit each tab.

Creating a ribbon panel	Open the **Customize** dialog box and click on the **Ribbon panels**. Right-click the **Ribbon panels** group and choose **Append ribbon panel** in the context menu.  Fill out the fields in the **Add Ribbon Panel** dialog box:  **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.
Editing a ribbon panel	Open the **Customize** dialog box and click on the **Ribbon** panel. Expand the **Ribbon panels** group and you can edit each panel.

3. Working with Document Tabs

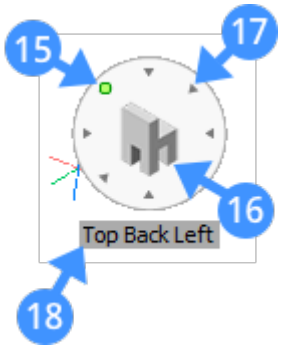
Document tabs allow you to:

- 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 **+** icon (14) at the end of the document tab to open a new drawing.

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.



5. UCS Icon

UCS stands for a User-defined Coordinate System. The UCS icon shows the nature of the current coordinate system.



The display of the icon is controlled through the UCSICON system variable and the position of the UCS icon is defined by the UCSICONPOSITION 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

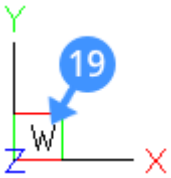
Each drawing has at least one layout and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale.

Paper space is a work environment that provides the model space view at a given scale, depending on the size of 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.

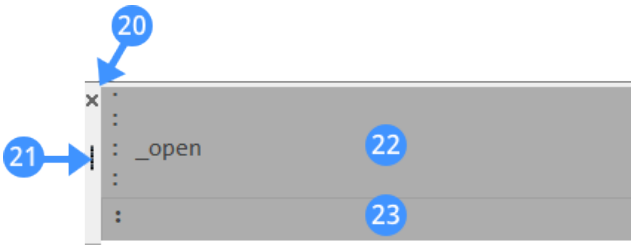


Nota: WCS stands for World Coordinate System.

8. Command line

The Command line 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 line is closed, this information shows in the Status Bar.
- The command history displays in the upper field of the Command line.

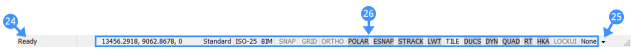


Close (20)	Click to close the Command line.
Grip (21)	Press and hold the left mouse button to drag the command bar.
Command history (22)	Shows the previously typed commands.
Command edit (23)	Type the commands and command options here.

9. 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 for the Status field (24). If you click the small down arrow button at the right end of the status bar (25) a list of all field displays. Click a field to toggle its display.

Right-click a field to display an options menu for this field.



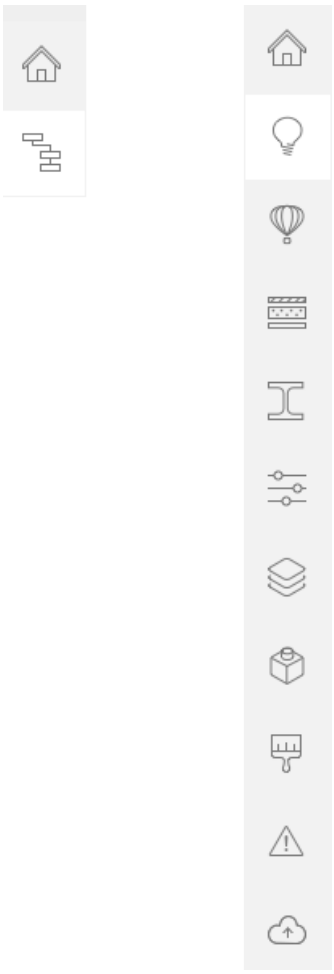
Status field (24)	Displays the status of the software.

List button (25)	Click the arrow to display a list of available Status Bar fields.
Fields (26)	Click a field to toggle it ON/OFF. Right-click to display a context menu.

10. Tool Panel

In BricsCAD BIM, the Tool Panel is displayed on 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.

By default the Tool Panel looks like the image below: on the left-hand side it displays the **Structure Browser** and the **Project Browser** and on the right-hand side it displays **Tips**, **Components**, **BIM Compositions**, **BIM Profiles**, **Properties**, **Layers**, **Mechanical Browser**, **Materials**, **Report** and **Bricsys 24/7**.



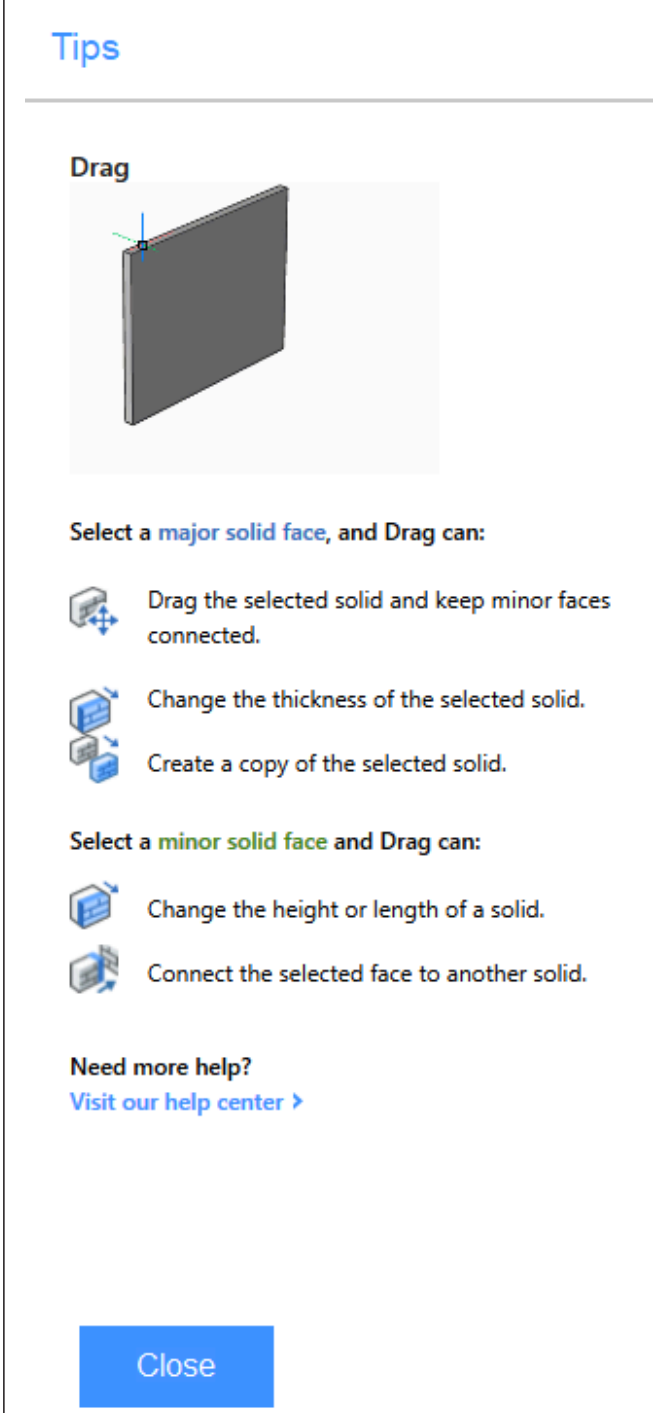
Nota: You can always add panels in the Tool Panel by right-clicking and checking those you want.

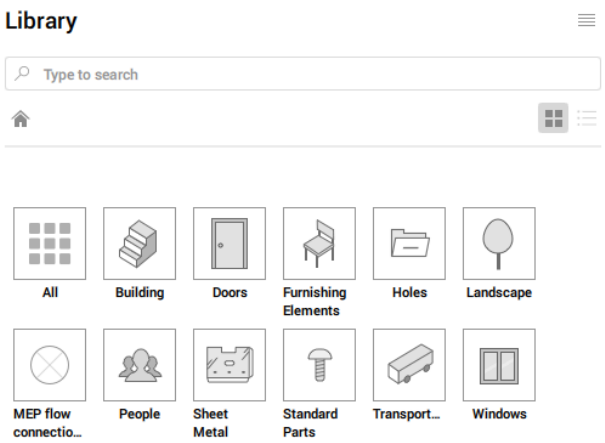
We will now have a look at the functions of the different dockable Panels from the Tool Panel:

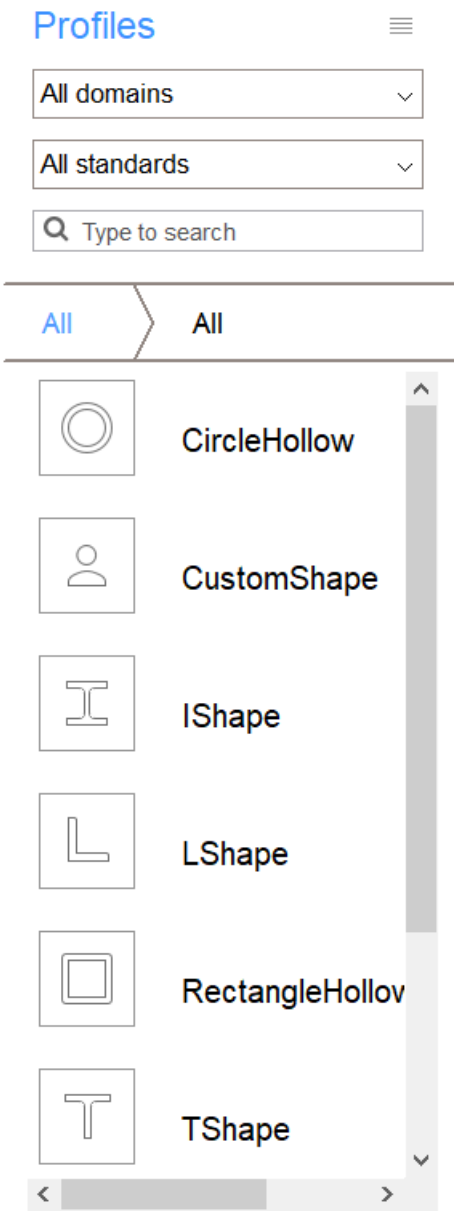
Project Browser	The BIM Project Browser allows you to easily navigate through the entire content of a project: models, sections, drawings, and schedules. See The **BIM Project Browser** article on how to use this panel.
Structure Browser	The **Structure Browser** is an interactive tree that displays 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**. See The **Structure Browser** article on how to use this panel.

Tips

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 DRAG command is active:

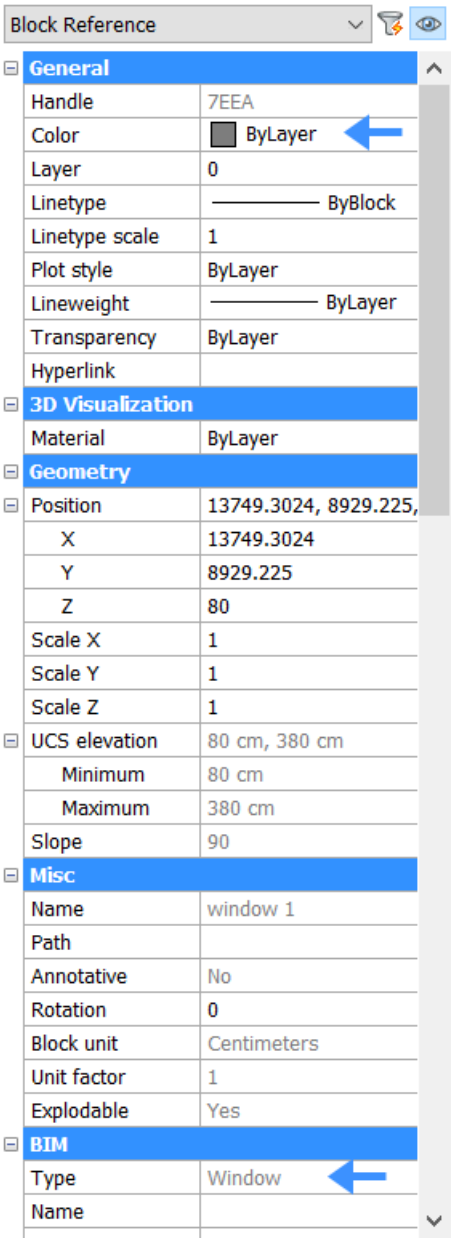


Library	The **Library** panel gives a list of components that you have by default. You can simply click and drag these components into your drawing.  At the bottom of the panel, you have the button . When you have made for example a drawing of a window, you can click on this button and add your own drawing to the component library. This way your window can be used as a component in all of your other drawings.
BIM Compositions	In the **BIM Compositions** panel, you find a library of predefined composition that you can assign to various solids of the BIM model by dragging them on. Assigning compositions to the elements in your model is a core step to increase the BIM model accuracy. Compositions can contain either multiple materials or a single ply. See The **BIM Compositions** article on how to use this panel in more detail

BIM Profiles	The **BIM Profiles** panel shows a library of predefined profiles that you can assign to linear solids by dragging them on.  See The **BIM Profiles** article to see how to work with this panel in more detail.

Properties

The **Properties** panel shows all the properties of a selected element. You always have some general info first, the other different tabs within this panel depend on the type of element that is selected.

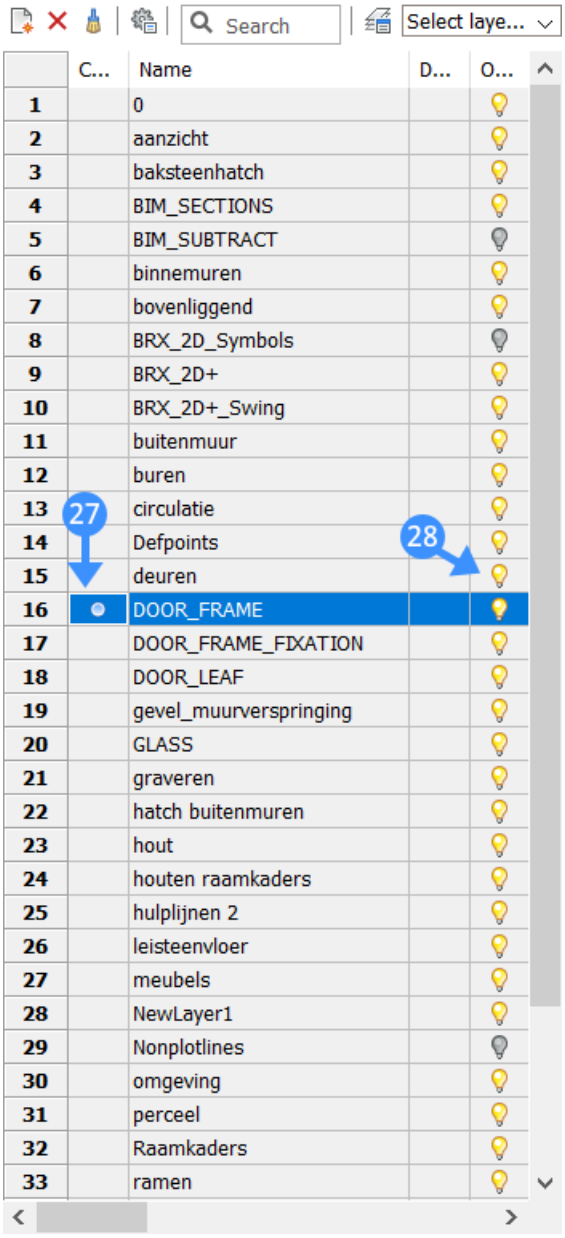


For example, here we can see that the selected item is a window and is colored by layer. You can adjust the properties as you like.

See the **Properties** article to see how to work with this panel in more detail.

Layers

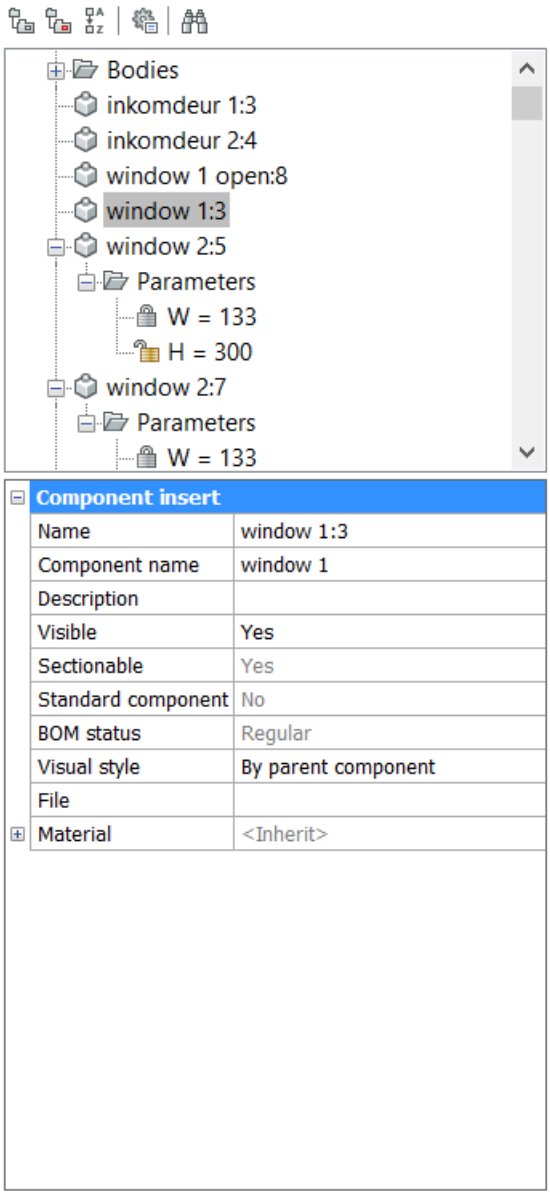
The **Layers** panel shows all the layers of your drawing.
It shows you the current layer (27), you can change it by clicking left of another layer. It also shows if a layer is ON or OFF by the state its lightbulb icon (28). Other layer properties such as color, line type,... can be found and adjusted by scrolling to the right.



In this example the current layer is **DOOR_FRAME** and it is turned ON.

Mechanical Browser

The **Mechanical Browser** panel displays the hierarchy of components, features, arrays, parameters, and constraints 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).

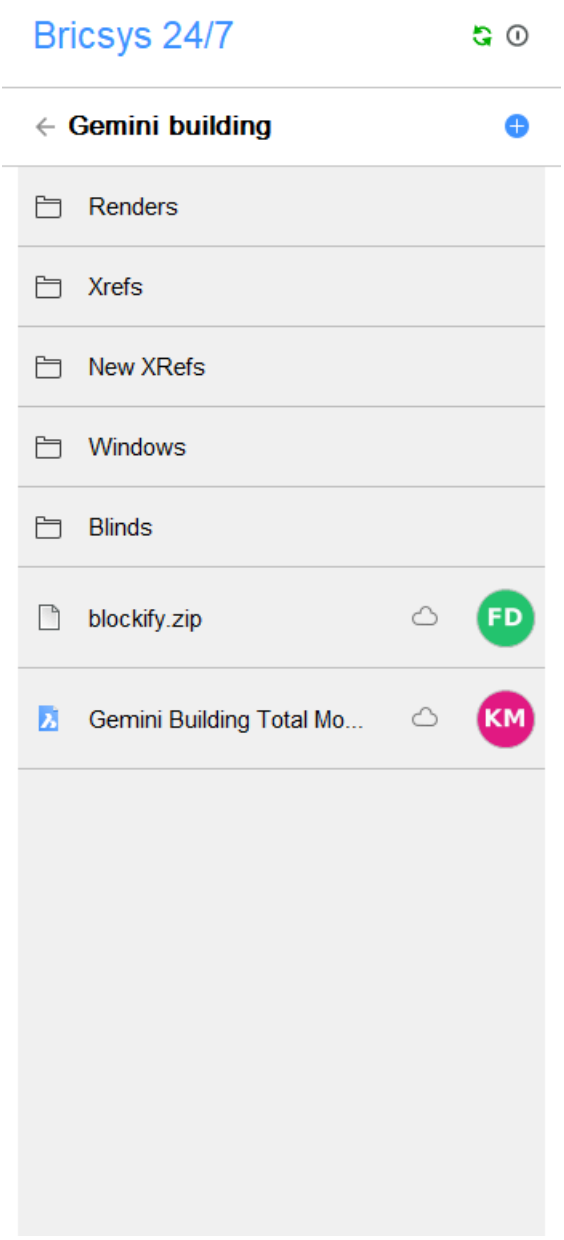


The **Mechanical Browser** allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right click on a particular node. For example, in the image above you could add a description for the selected window.

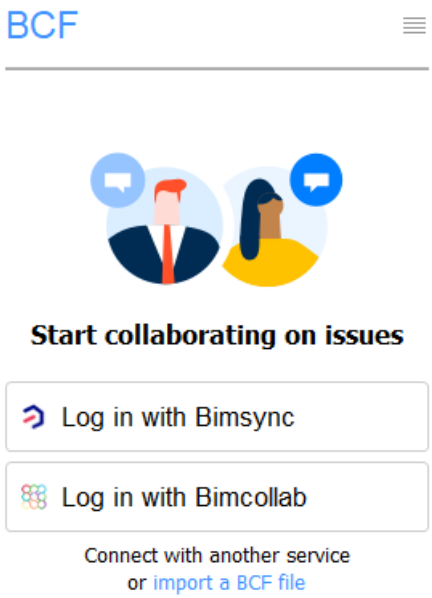
Render materials	The **Render materials** Panel contains a library of predefined materials that you can assign to solids by dragging them on. You can also see a list of all the materials that you are currently using in your drawing. See the **Materials** article to see how to edit and create materials.
Report	Some commands use the **Report** panel to report the command output; e.g. all SHEETMETAL (Sm*) commands, DMAUDIT and DMSTITCH.

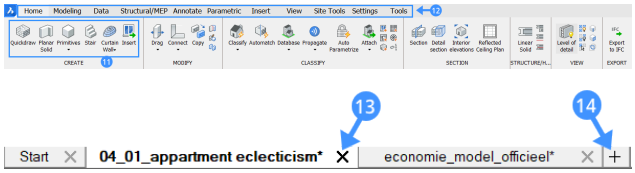
Bricsys 24/7

Login to your Bricsys 24/7 account here to collaborate on your project in the cloud. BricsCAD 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible.
When logged on, you can see an overview of your in the cloud Projects, Folders and Documents as shown in the image below:



See the **BricsCAD 24/7** panel article to see in detail how this panel works.

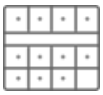
BIM BCF	Here you can import BCF (Bim Collaboration Format) files and connect to a BCF server. The BCF topics are listed in this panel.  The top part of the new **BCF** panel shows all the topics and can look similar to the **Library** panel. The lower part of the new **BCF** panel shows the currently active topic, with a bigger snapshot and the most common metadata: comment, author, status, etc. Double-clicking on an issue activates the topic.
Details	In the **Details** panel you can: • View saved details. • Organize saved details. • Propagate details. • Open details and adjust them in the model space.



20.2.3 Quad cursor

20.2.3.1 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 line, then choose an option.

To toggle the **Rollover Tips** on/off, do one of the following:

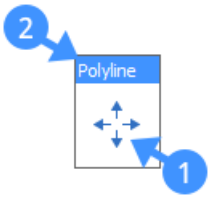
- Click the **RT** field in the Status bar.
- Type ROLLOVERTIPS in the Command line then set a value.

20.2.3.2 Quad layout

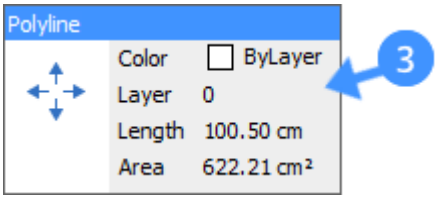
The **Quad** has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace.

- Based on the highlighted entity type, the **Quad** displays the most recently used tool (1) with the entity name (2).

  Nota: When multiple entities are under the cursor, hit the TAB key to cycle through the different entities.

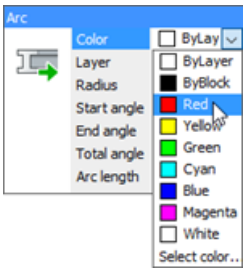


- When the **Rollover Tips** setting is on, a limited set of properties (3) of the entity under the cursor display in the **Quad**.



  The property values in the **Rollover Tips** can be edited.

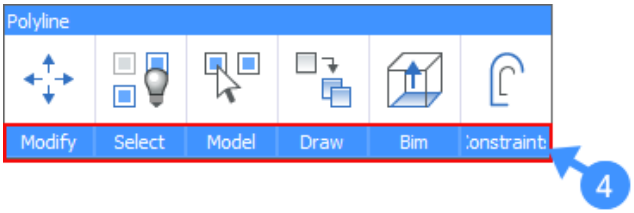
  The following figure shows the color of an arc being changed using the **Rollover Tips**.



Nota: To display the **Rollover Tips**, simply click the title bar of the **Quad** after expanding it.

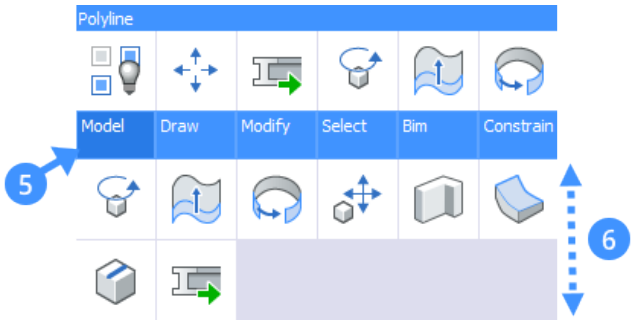
- When the cursor is moved on the **Quad**, a number of tool tabs display at the bottom (4).

  Nota: The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6).

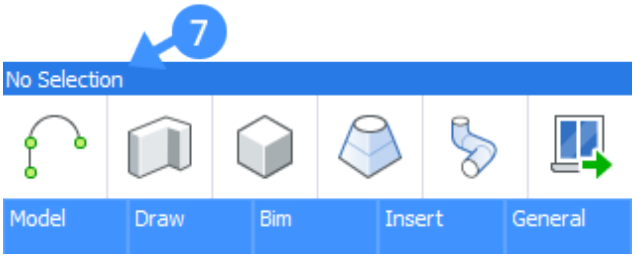


- When you move the cursor over the tool tabs title bar (5), the tool groups expand (6) under the cursor, which provides you with more tool options.

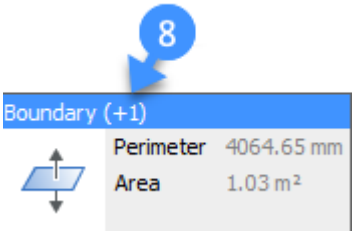
  Nota: The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor.



- 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.



- 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.

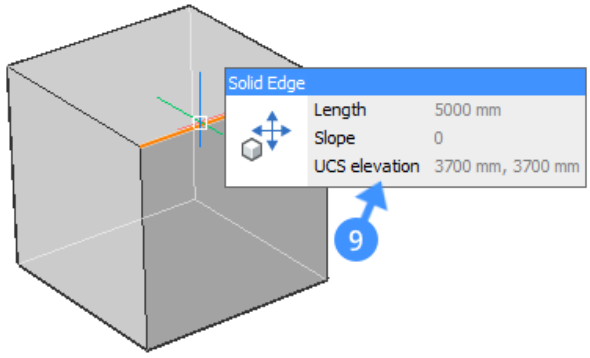


Nota: The **Quad** also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

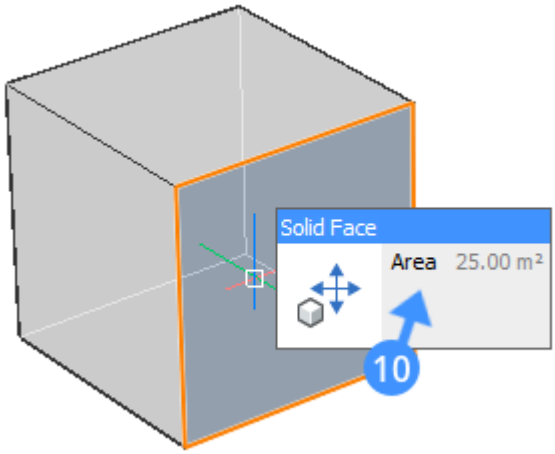
20.2.3.3 About: selecting entities

The **Quad** adapts to the type of entities that you have selected. If you select a surface, you will be provided with different tool options in the Quad, then selecting a solid or edge.

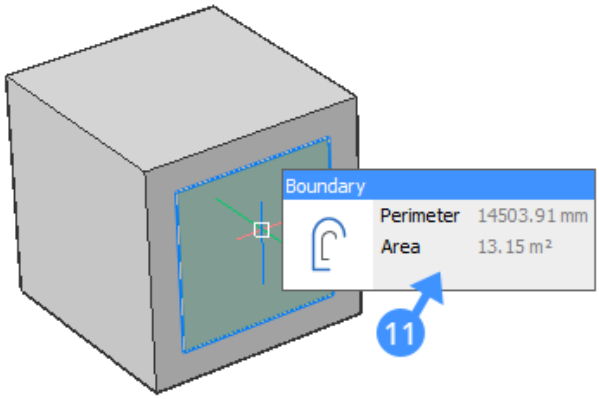
- Selecting an **edge**, the **Quad** appears showing the edge length, slope, and elevation (9).



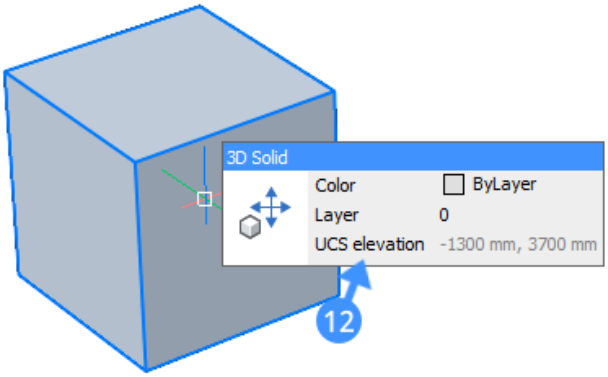
- Selecting a **solid face**, the surface area is shown in the **Quad** (10).



- Highlighting a **boundary**, the **Quad** displays the perimeter and area of this boundary (11).



- Highlighting a **solid**, the **Quad** displays the color, layer, and elevation of the solid (12).



20.2.3.4 Procedure: using the quad

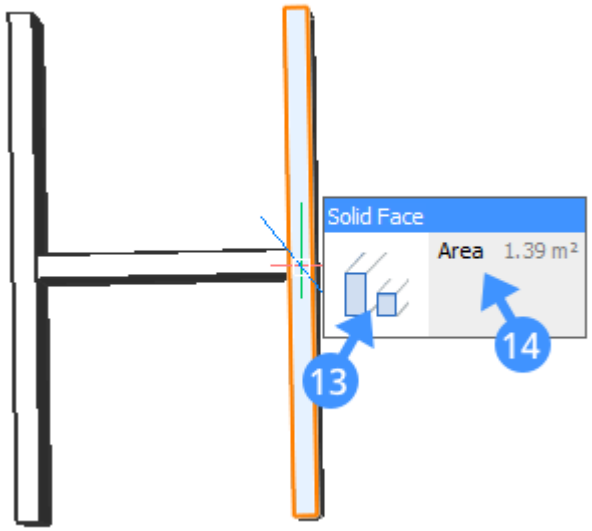
The following are general steps to use the **Quad** cursor.

Nota: The **Quad** and **Rollover tips** are both enabled.

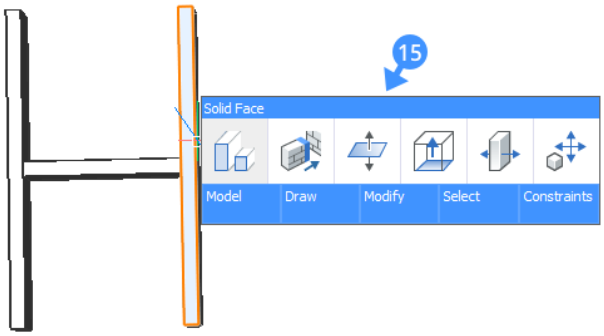
1 Hover the cursor over the entity you want to manipulate.

1 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.



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.



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.

20.2.4 Viewing entities

20.2.4.1 Commands

HIDEOBJECTS, ISOLATEOBJECTS, UNISOLATEOBJECTS

20.2.4.2 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.

20.2.4.3 Hiding entities

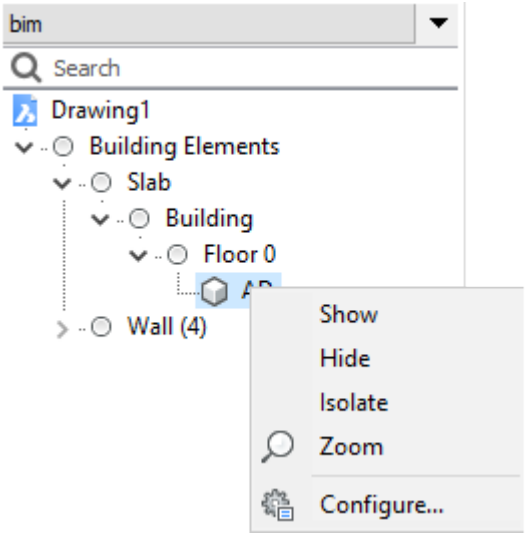
The **Hide** tool temporarily hides the selected entities in the view.



For more information about this command, visit the Command Reference article HIDEOBJECTS.

(**Optional**) **Structure Browser** can also be used to hide and isolate selected entities as well as revealing all hidden entities in 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).



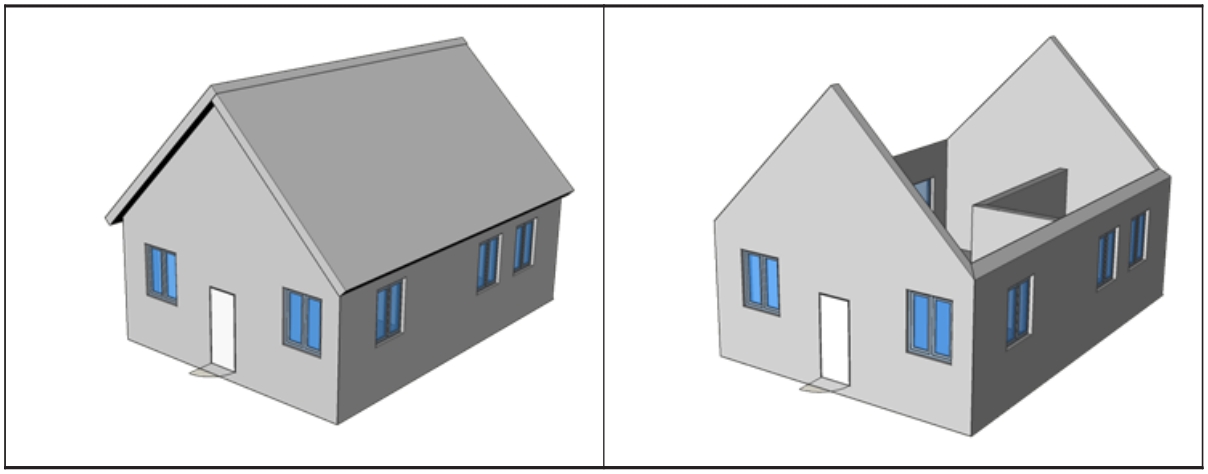
20.2.4.4 Procedure: hiding entities

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.

The following illustration shows an entire model on the left. In the second image, the roof was temporarily hidden.



20.2.4.5 Isolating entities

The **Isolate** tool temporarily hides all entities except those that have been selected.



For more information about this command, visit the Command Reference article ISOLATEOBJECTS.

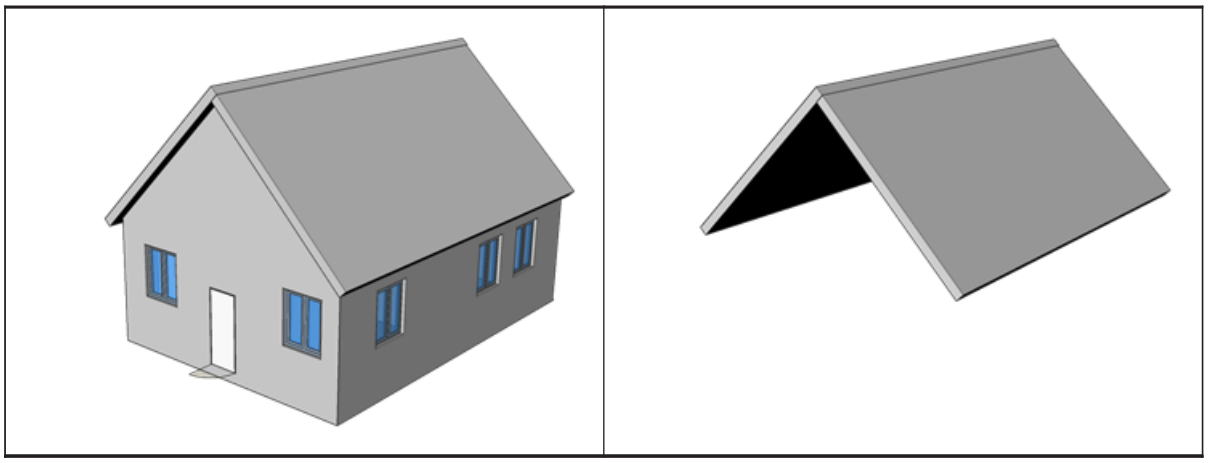
20.2.4.6 Procedure: isolating entities

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.

The following illustration shows the same building model from the below. In the second image, the roof was temporarily isolated.



20.2.4.7 Showing entities

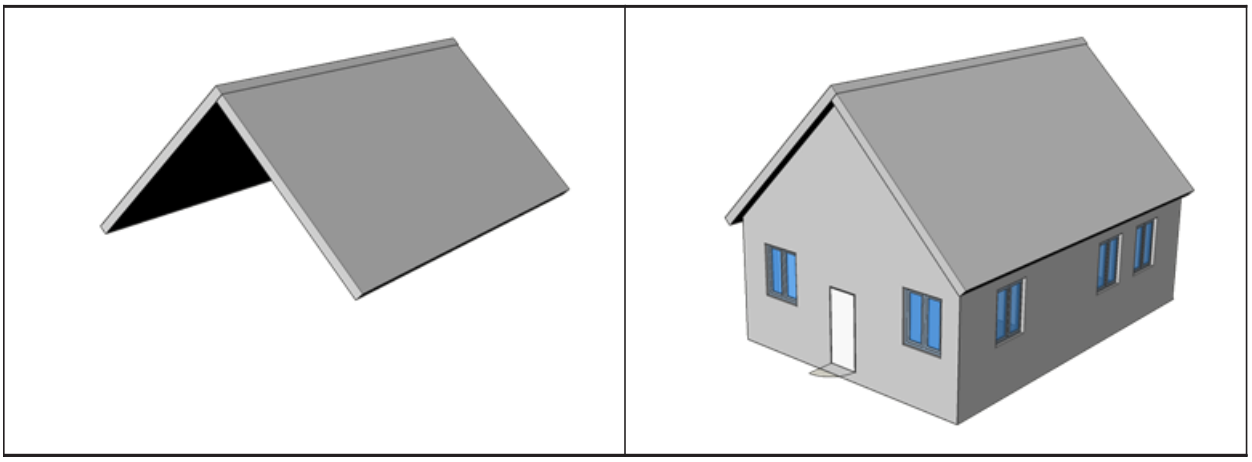
The **Show Entities** tool brings back all hidden entities in the view.



For more information about this command, visit the Command Reference article UNISOLATEOBJECTS.

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.



20.2.5 Selecting entities

20.2.5.1 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.

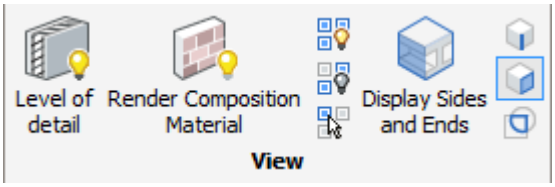
Selecting: you can select as many entities as you want.

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.

20.2.5.2 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** > **View panel** > **Selection modes**



Nota: To display the **Ribbon**, right-click on a toolbar, then choose **Ribbon** in the context menu.

Settings: **Program options** > **Selection** > **Selection Preview** > **Selection Modes**

Selection modes	0x0007 (7)
1	Select edges
2	Select faces
4	Select detected boundaries

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.

Nota: BIM Sides/Ends are only available on classified linear elements with the attached library profile.

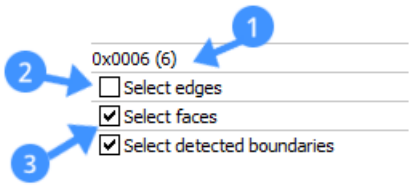
20.2.5.3 Changing selection modes setting

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:

Nota: The default value of the SELECTIONMODES system variable is 6, which means the detection of 3D solid edge is disabled, while the detection of 3D solid faces and boundaries are enabled.

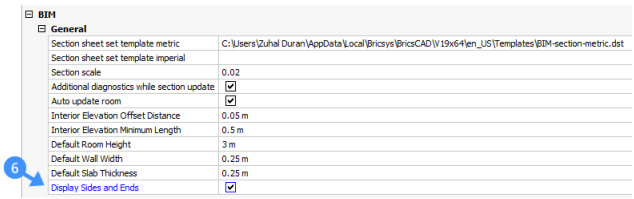
- Click the buttons on the **Settings** panel on the **Ribbon** and find **Selection Modes**.

  The following illustration shows the sum of the values (1) of the SELECTIONMODES 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).



- 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 entity.
- To display the BimSides/Ends for linear elements you can use DISPLAYSIDESANDENDS system variable in the **Settings** dialog box or set the DISPLAYSIDESANDENDS system variable to 1 in Command line.

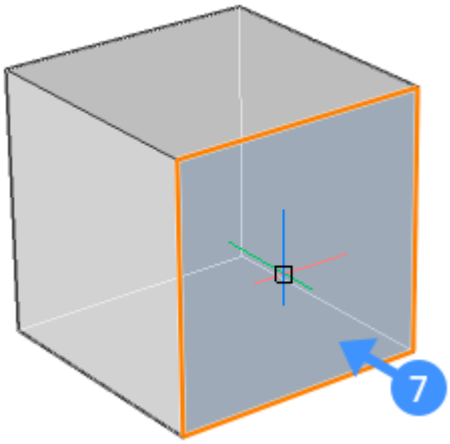
  The following illustration shows the **BIM** > **General** system variable when the DISPLAYSIDESANDENDS is enabled (6) after clicking the checkbox once.



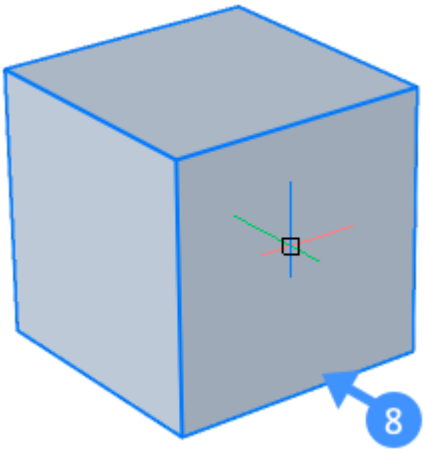
20.2.5.4 Selecting sub-entities

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default), do one of the following:

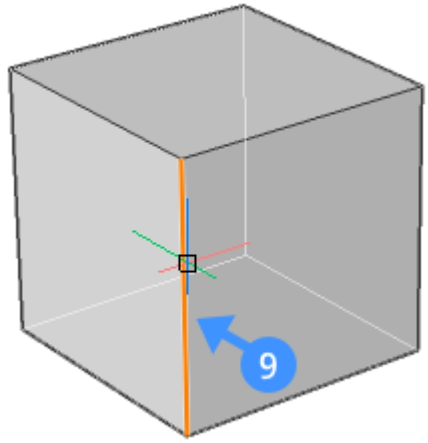
- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face displays in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.



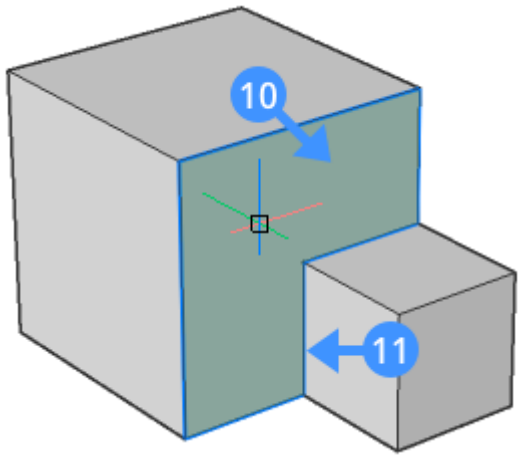
- To highlight a solid, hover over one of its faces while holding down the CTRL-key. The solid displays in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.



- 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.



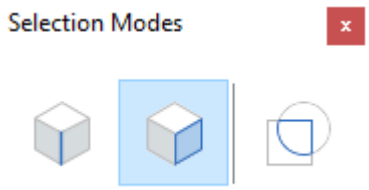
- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.
- When the detected boundary is highlighted, it displays with a color. By default, inside of the boundary displays in green (10) and the outer edge of the boundary displays in blue (11).



Nota: This setting can be changed by typing BOUNDARYCOLOR on the Command line (default value = 95).

20.2.5.5 Procedure: Modify the thickness of a solid

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the face of a solid

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

5 The selected face of the solid is dragged

6 The dynamic dimension field displays the relative displacement of the face (= with respect to its original position).

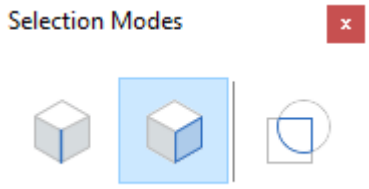
7 (**Optionally**) Press Tab to change the face selected as the reference face for the dynamic dimension.

Nota: 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 command to re-apply the composition to such solid.

20.2.5.6 Procedure: Modify the height of a wall

Modify the dimensions of a single wall

1 Make sure the **Select faces:** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the top face of the wall.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

The selected face moves dynamically with the cursor.

The dynamic dimension fields display the incremental height.

Do one of the following:

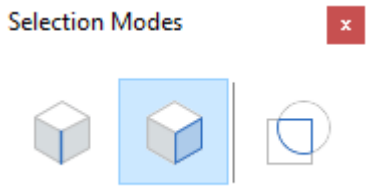
5 Type a value in the dynamic dimension field and press Enter.

6 Press Tab to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

Modify the height of multiple walls

Use the SELECTALIGNEDFACES 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.



2 Move the cursor over the top face of one of the walls.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose SELECTALIGNEDFACES in the **Solid/Face Selection** command group in the Quad.

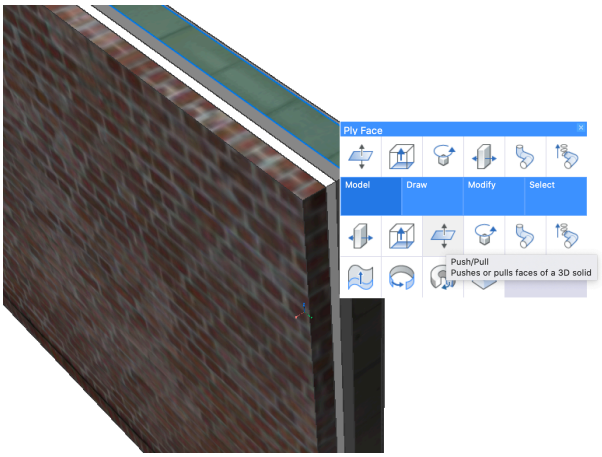
All faces lying in the plane of the selected face are selected.

5 Now follow the same steps as for a single wall.

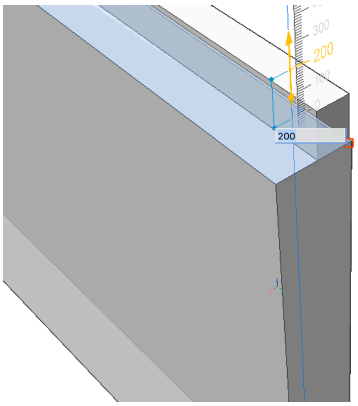
Modify the height ply-by-ply

When the **Display Composition** property of a solid is set (LEVELOFDETAIL system variable =2 and, optional, RENDERCOMPOSITIONMATERIAL system variable =1) and the **Select faces** option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

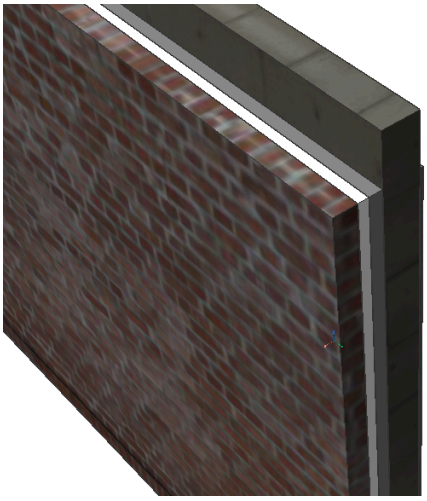
1 Move the cursor over the ply you want to edit and choose **Push/Pull** in the **Model** command group of the Quad when the ply highlights.



2 Type value in the dynamic distance field.



3 Press Enter to accept the distance.



20.2.5.7 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.

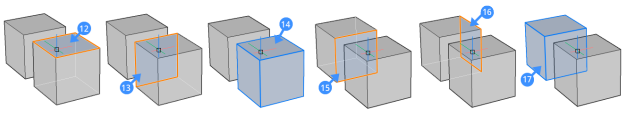
1 In the example below, the top face of the front solid is highlighted (12).

2 Press Tab key to cycle through the hidden entity.

3 Now, the back face of the front solid is highlighted (13).

4 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).

5 When the entity or sub-entity that you want to select is highlighted, left-click it to select.



20.2.5.8 Selecting multiple entities using the selection boxes

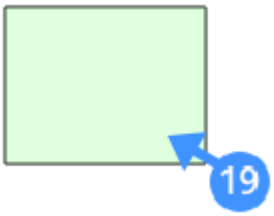
The selection boxes allow you to select one or more than one entity at a time. To use a selection box, left-click on an empty space in the drawing area, and move the cursor until the selection box covers the entities that you want to select.

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.



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.



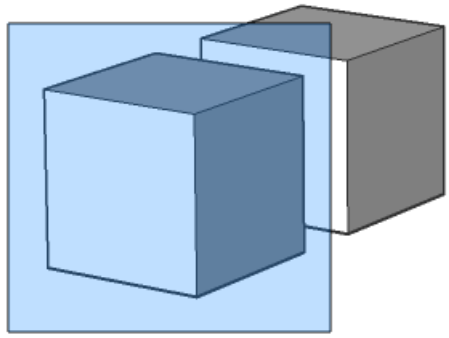
The following steps show how to select multiple entities using the blue selection box:

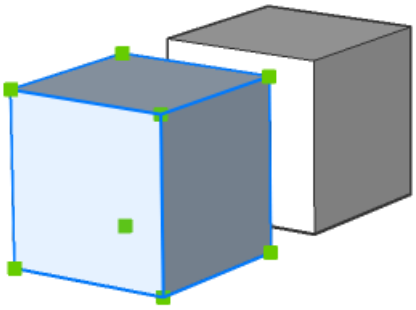
Nota: By default, the selection box only selects entire entities (e.g. solids, lines, polylines, blocks, …) but not sub-entities (e.g. solid faces, solid edges).

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.

1 The example below shows the blue selection box usage when the selection mode is set to **Solids**

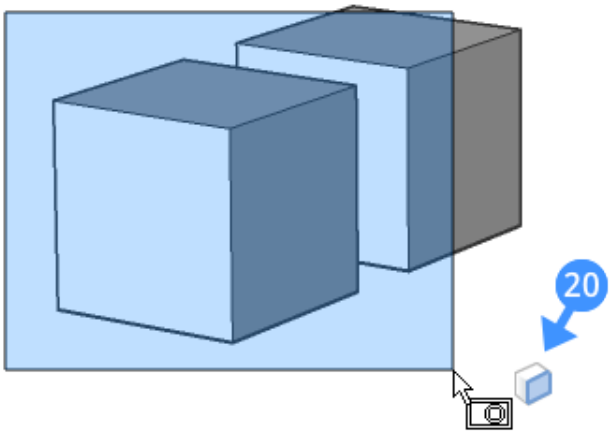
(default).

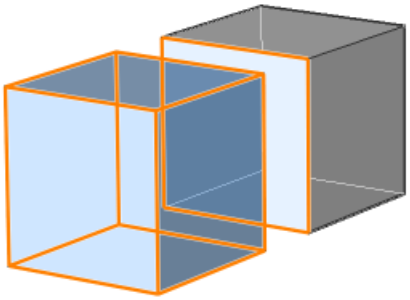




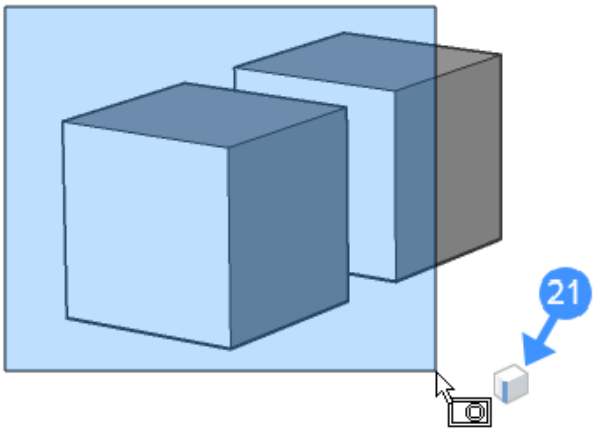
The blue selection box covers the left cube completely and thus only this cube is selected in the model.

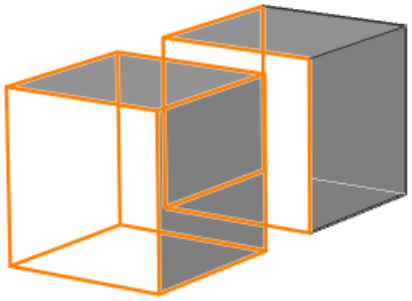
2 To select faces of 3D solids and surfaces, press CTRL key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to **Faces**.





3 To select edges of 3D solids and surfaces, press the 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**.

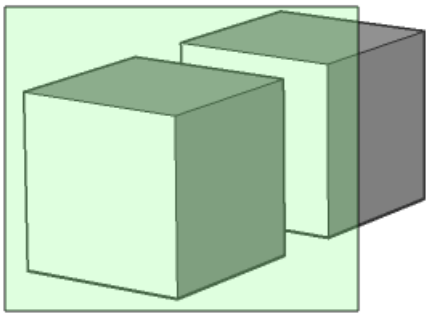


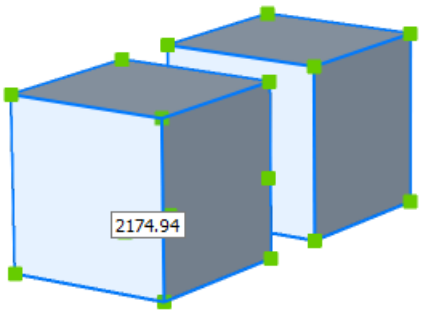


The following steps show how to select multiple entities using the green selection box:

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.

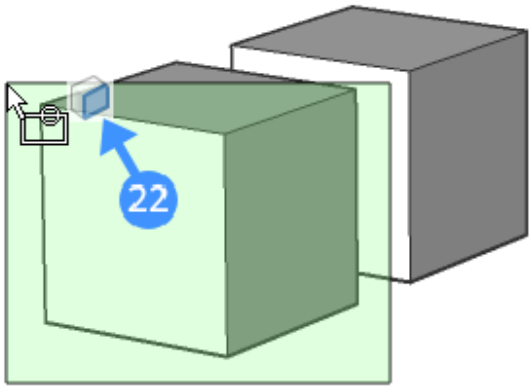
5 The example below shows the green selection box usage when the selection mode is set to default.

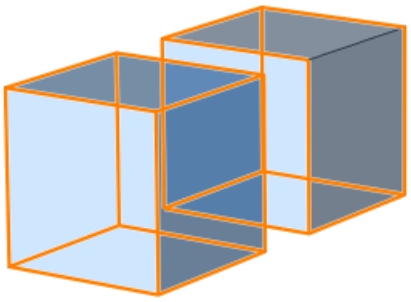




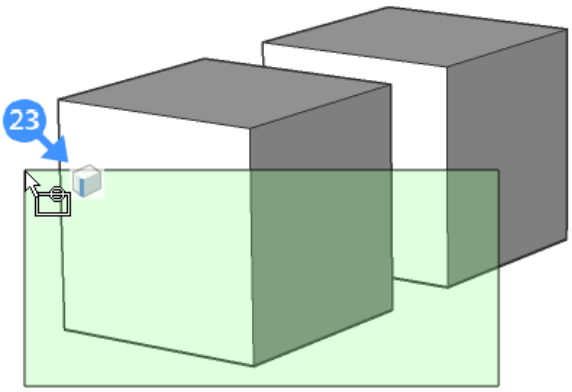
The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

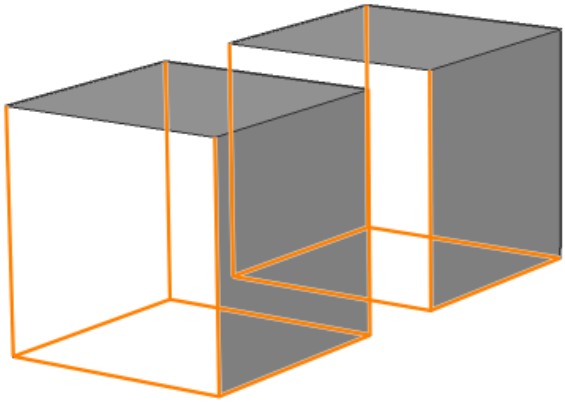
6 To select faces of 3D solids and surfaces, making use of the green selection box, press the CTRL key once during window selection. The ‘Faces’ icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected.





7 To select edges of 3D solids and surfaces, making use of the green selection box, press the 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.



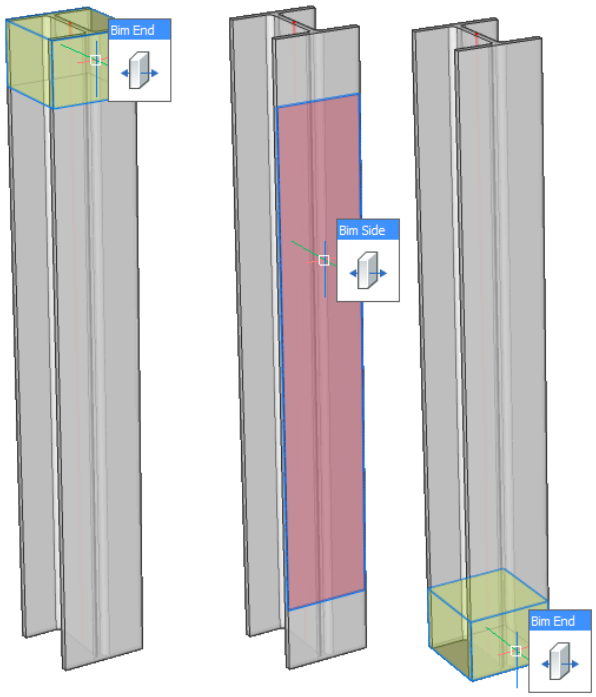


20.2.5.9 Selecting/Highlighting BIM sides and ends on profiles

1 Click on **Enable selection of Sides and Ends** icon in the BIM Toolbar (see below image).



2 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.



20.2.6 Level of detail

The RENDERCOMPOSITIONMATERIAL and LEVELOFDETAIL system variables control the display of materials and multiply compositions.

20.2.6.1 Level of detail



Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
 [0] Low	Multiply compositions do not display.
 [2] High	The individual plies of the multiply compositions are visible and editable.

20.2.6.2 Render composition materials



Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
 [0] Off	Materials are not rendered.

Option	Description
 [1] On	Materials are rendered. When Level Of Detail = 0 (Low), major faces of solids, that have a multiply composition attached, are rendered in the material of the underlying ply. Minor faces are rendered in the material of the exterior ply.

20.3 Técnicas de modelagem

20.3.1 BIM linear solid

20.3.1.1 Commands

BIMLINEARSOLID, BIMAPPLYPROFILE, BIMPROFILES, BIMADDECCENTRICITY, BIMRECALCULATE

20.3.1.2 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.



For more information about this command, visit the Command Reference article BIMLINEARSOLID.

For more information about **MEP modeling**, visit the help article **MEP Modeling** and **MEP Flow Connection Points**.

For more information about **Structural Steel**, visit the help article **Structural Steel**.

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 the Linear Solid tool.

Nota: You can also launch the **BimLinearSolid** command by entering it in the command line.

You are prompted: Set start point or [Follow/Rectangular/Circular/Library/select in Model/adJust profile] <Last point>:

2 Select a start point.

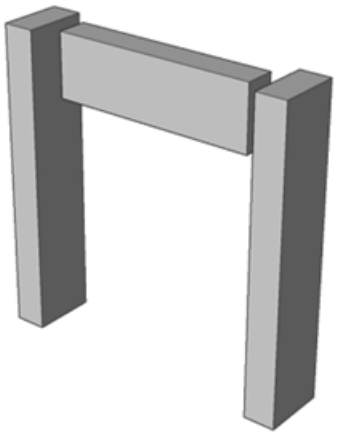
3 You are prompted: Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/ adJust profile]:

4 Move the mouse cursor to draw the beam or column.

5 Click to specify the endpoint or enter a value for the distance using the dynamic dimension field.

6 Press Enter key to accept it.

7 The following illustration shows a beam and two columns with the default rectangular profile.

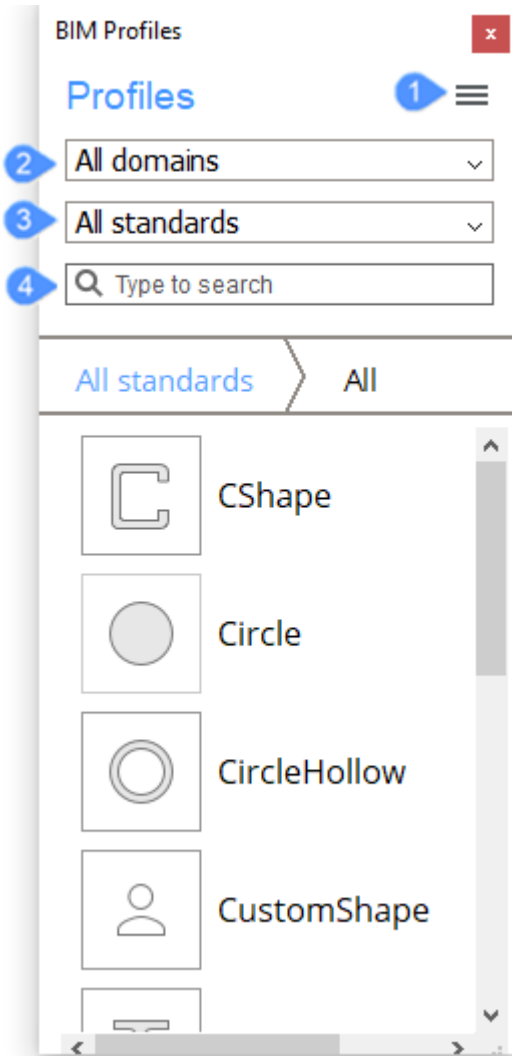


20.3.1.3 About BIM profiles

You can access the **BIM profiles** panel by clicking the I-beam on the right-hand side of your screen. 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, choose **Panels** in the context menu, then check **BIM Profiles**.

Nota: You can also find the **BIM profiles** panel by typing BIMPROFILES in the Command line.

The **BIM Profiles** panel will be displayed:



The drop-down box at the top (2) acts as a filter. There are different domains on which you can filter: **Generic**, **Structural Concrete**, **Structural Steel**, **HVAC**, **Piping** and **Electrical**.

Nota: 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 (3) allow you to filter the profiles by the country standards.

The search box (4) allows you to filter by name.

The menu (1) on the right-hand side allows you to access the **BIM Profiles** dialog box. Clicking the menu icon displays a context menu with a couple of options.

Select **Open profile dialog** on the menu to open the **Profiles** dialog box.

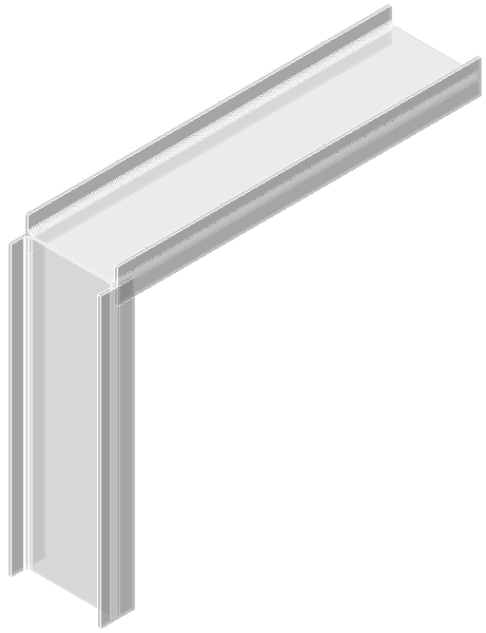
20.3.1.4 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

Nota: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



Nota: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

20.3.1.5 About BIMAPPLYPROFILE

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.



For more information about this command, visit the Command Reference article BIMAPPLYPROFILE.

How to apply profiles to an existing line or solid: with the BIMAPPLYPROFILE command

1 Select the line or solid you want to apply a profile to and launch the BIMAPPLYPROFILE command.

2 Press Enter and the **Profiles** dialog box, appears.

3 Choose a profile and press the **Select** button in the bottom-right corner.

4 Press Enter to end the command. The profile is applied to the line or solid in your model.

How to apply profiles to an existing line or solid: with the BIM Profiles panel

1 Open the **BIM Profiles** panel.

Nota: You can access the **BIM Profiles** panel by clicking the I-beam icon on the right-hand side of your screen.

2 Select all the entities you want to apply a profile to.

3 In the **BIM Profiles** panel, search for the profile you want apply to the selected entities.

4 Drag the chosen profile onto one of the selected entities.

5 The profile is applied to the line or solids in your model.

Nota: By applying a profile to a solid, the solid will be replaced by the chosen profile. In case of a line, the line will not be replaced, but a profile was added with his axis the same as the selected line.

6 Since the line is no longer needed, you can delete the line.

20.3.1.6 About BIMADDECCENTRICITY

BIMADDECCENTRICITY command allows you to control the relative position of the axis of a linear solid.



For more information about this command, visit the Command Reference article BIMADDECCENTRICITY.

1 Select an existing linear solid or profile in the drawing area.

2 Launch the BIMADDECCENTRICITY command by entering it in the Command line or select the command in the Quad (under the model tab) or in the Ribbon (in the **Structural/MEP** tab).

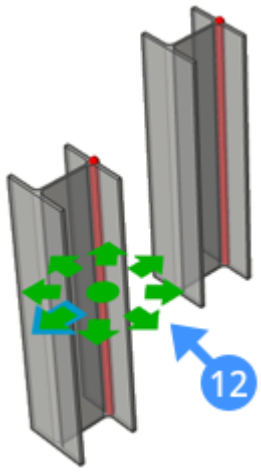
The eccentricity widget appears.

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.

5 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 .

6 Press Enter to accept changes. The following illustration shows the Eccentricity widget (12) on the linear solid profile.



20.3.1.7 About BIM recalculate axis

The Recalculate Axis tool recalculates the axis of linear building elements and repositions it back to the centerline of a linear element.



For more information about this command, visit the Command Reference article BIMRECALCULATEAXIS.

Recalculate the axis of linear building elements

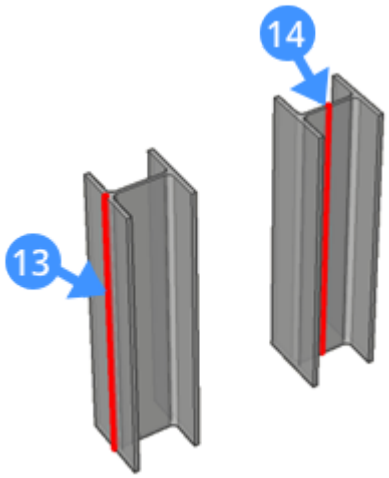
1 Select any linear building element in your model.

2 Launch the BIMRECALCULATEAXIS command.

3 Press Enter.

4 The axis of the element is moved to the default position.

5 The following illustration shows the result of using BIMRECALCULATEAXIS on the linear solid. The axis (13) is moved to the centerline (14) of the linear element.



20.3.1.8 Helpful settings when modeling Linear Solids

Display Axes

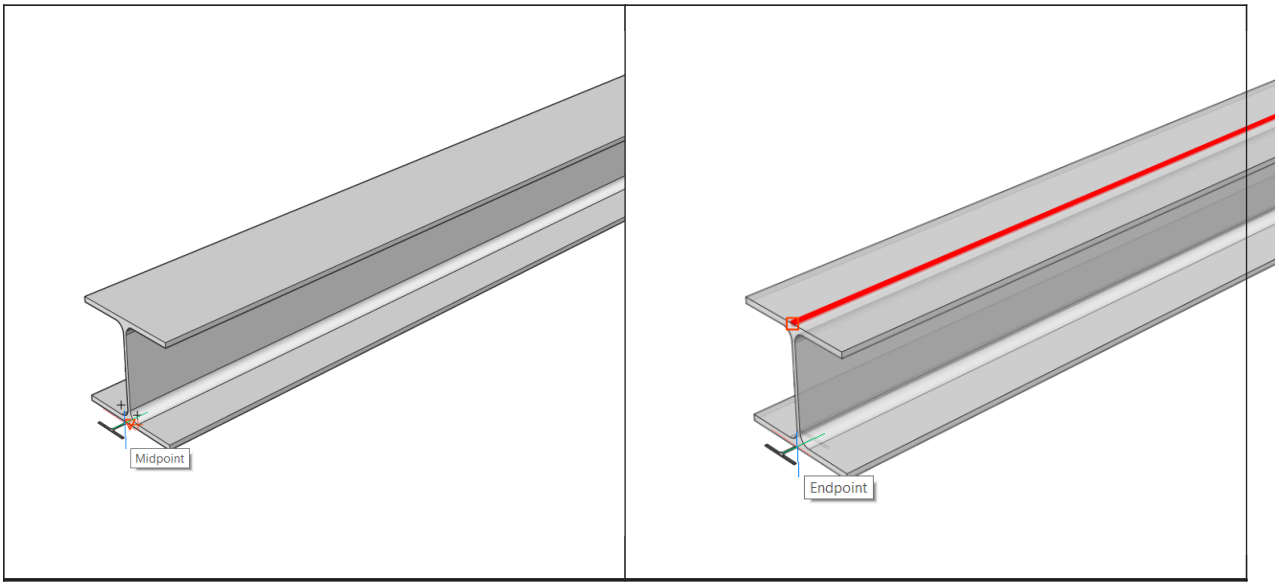
The DISPLAYAXES variable system can hide or view the Axes of linear solids in order to edit the structure more easily.



If DISPLAYAXES is toggled on, the axes of the linear solids are visible as a thick red line. In this case, you will only be able to snap the endpoint of the axis as a starting point to add more linear solids or profiles to your already existing structure.

When DISPLAYAXES is toggled off, no axis is visible in your drawing area and you're able to snap all specific points you have ticked in the context menu of ESNAP.

DISPLAYAXES OFF	DISPLAYAXES ON



BIMOSMODE

The BIMOSMODE variable system gives you the possibility to choose the level of detail you are snapping to.

1 Enter BIMOSMODE in the Command line or in Settings.

1 You are prompted: New current value for BIMOSMODE (0 to 3) <2>:

2 Choose a new value (type a number from 0 to 3).

Value	Snapping detail
0	Turns on all snapping.
1	Turns off all snapping to solids except for the axis of the solid.
2	Turns off all snapping to BIM Grid axes except for axis intersections.
3	Turns off all snapping to BIM Grid axes except for axis intersections. Turns off all snapping to solids except for the axis of the solid.

3 Press Enter

20.3.2 BIM window create

20.3.2.1 Commands

BIMWINDOWCREATE

20.3.2.2 About

In BricsCAD, the **Window Creator** tool allows you to turn a different set of design ideas into a window. You can use the tool on windows of all shapes and sizes, which enables you to utilize a significant amount of window styles in your BIM project.

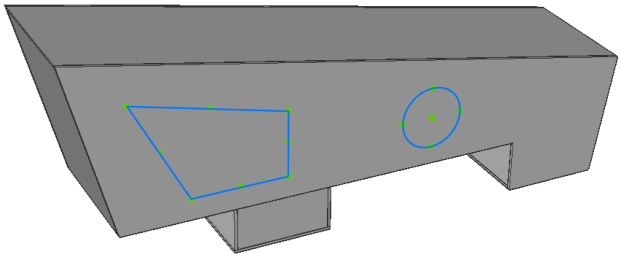
When you have a closed 2D entity or boundary on the face of a solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box and change the properties of this created window in a BIM model.



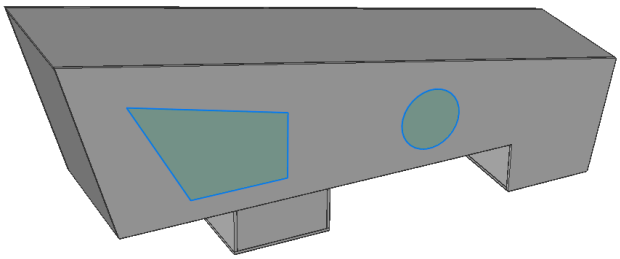
20.3.2.3 Creating a parametric window using boundaries

1 Draw a polygon on the face of a solid (wall).

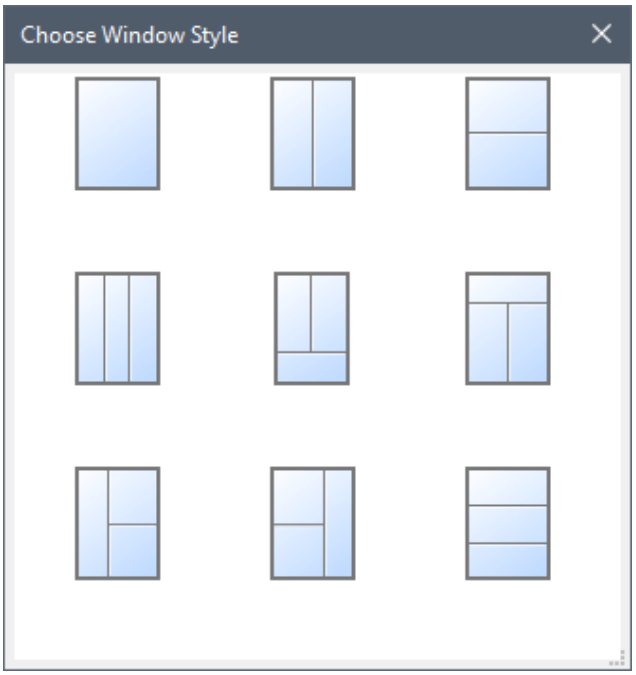
1 Based on this base profile, the window opening will be created.



2 Make sure **Boundary Detection** (select detected boundaries) is enabled into SELECTIONMODES system variable. Hover over the inside of the polygon and select the **Create Window** tool from the Quad.

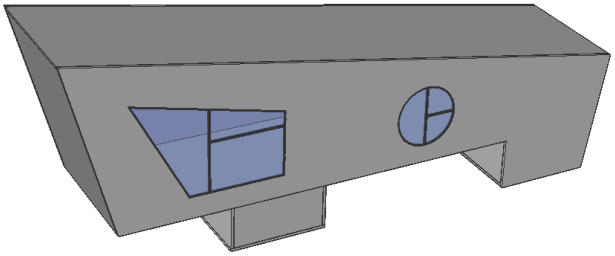


A **Choose Window Style** dialog box displays:

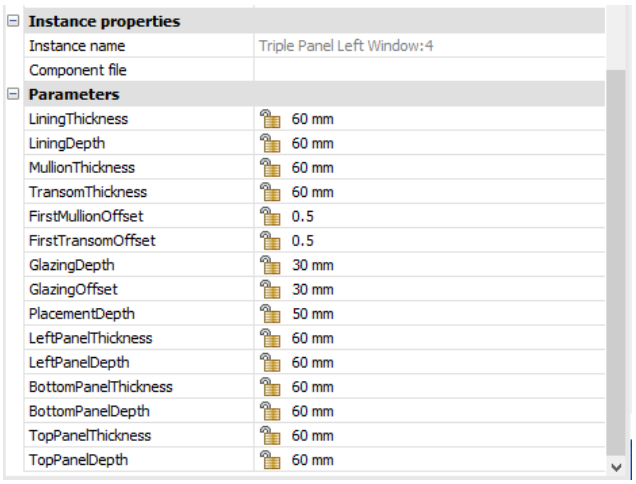


3 Select a window style in the dialog box.

4 The window is created on the wall with the selected style. The following illustration shows the triple panel left window style which was selected to create the windows on the face of the wall.



5 (**Optional**) You can refine the window design by changing the default values of the parameters. To do this, select the window and change the desired parameters in the **Properties** panel.



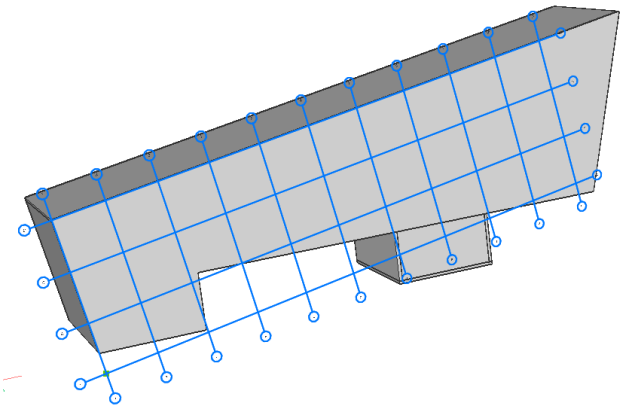
To change one of the default parameters of this window, type a new value in the property field.

Press Enter to accept the new value of the parameter. The window should be updated automatically.

20.3.2.4 Create windows based on a grid

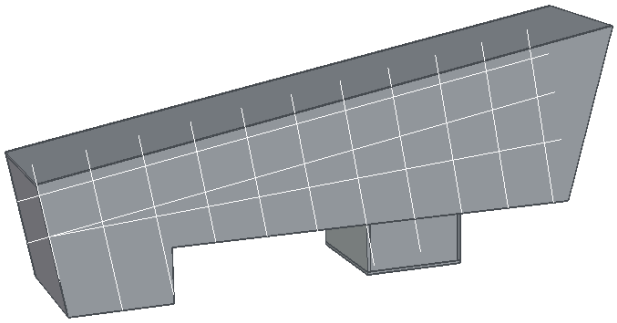
1 Use BIMGRID command to create a Grid on the face of the wall. Choose the basic dimensions in the Command line. Changing the **U-line** and **V-line** panel options allows you to specify the desired size and number of panels on the surface.

Nota: To draw on the face, make sure you first highlight the face and then click. Otherwise, the Grid will be created in the XY-plane.

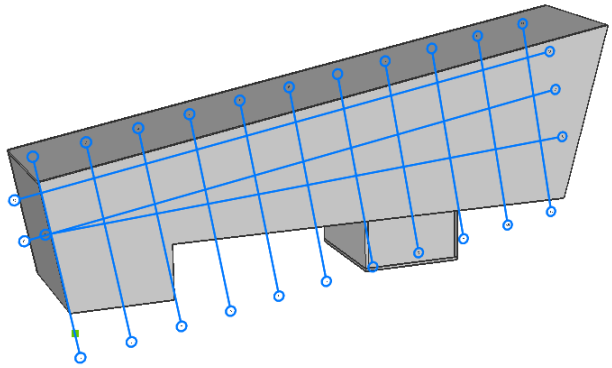


2 Use BEDIT or REFEDIT to edit the Grid.

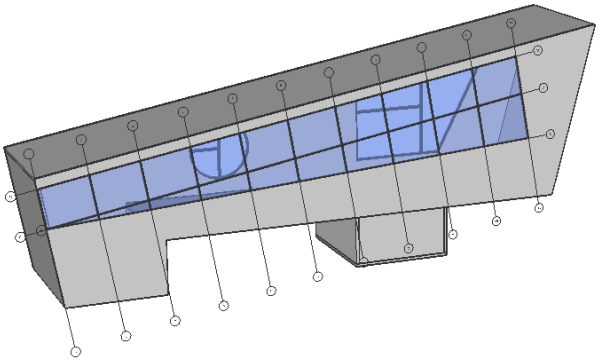
3 Use the Manipulator to change the position of the gridlines on the wall.



4 Select the Grid and select **Create Window** in the Quad.



5 The window is generated based on the gridlines.



Nota: You can edit the parameters in the **Properties** panel. Type a new value in the property field and press Enter. The window will automatically update itself.

Parameters	
FrameDepth	60 mm
GlazingDepth	30 mm
GlazingOffset	30 mm
PlacementDepth	50 mm

20.3.2.5 Creating window openings

BIM Opening Create creates an opening without window geometry. It uses the same command as BIMWINDOWCREATE. The command allows to create an opening at the location of the selected base profile. The base profile can be a boundary or a closed polyline.



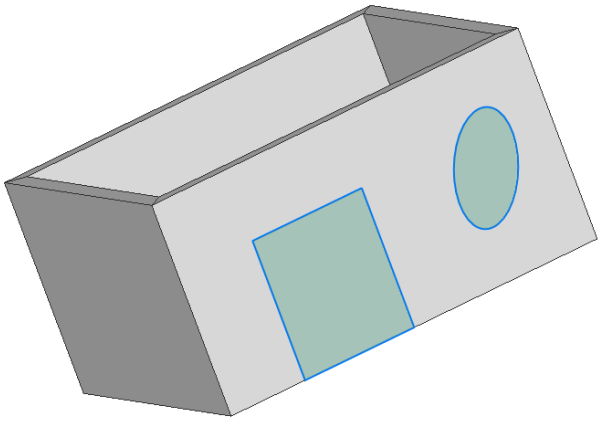
For more information about this command, visit the Command Reference article BIMWINDOWCREATE.

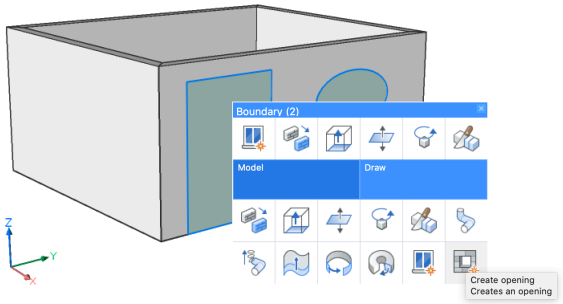
1 Draw one or more closed polylines on the face of a wall.

2 While boundary detection is on, hover over the polygon.

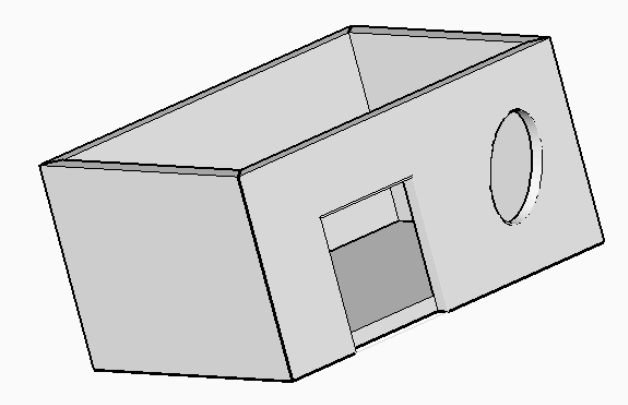
3 Or select the closed polyline.

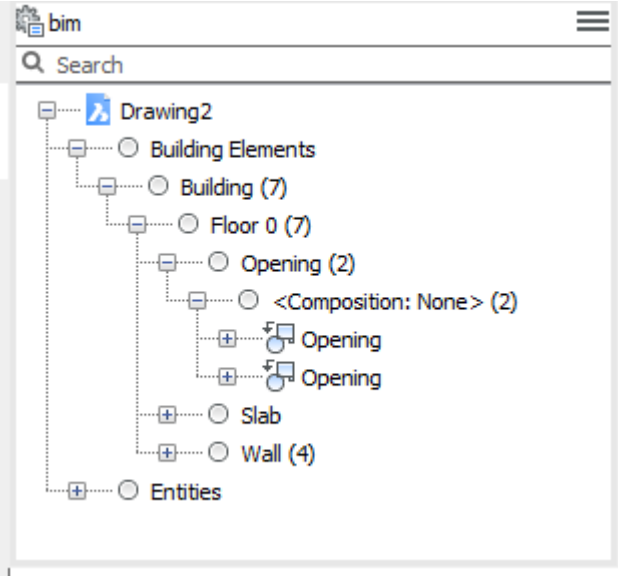
4 Select the **Create Opening** tool from the Quad.



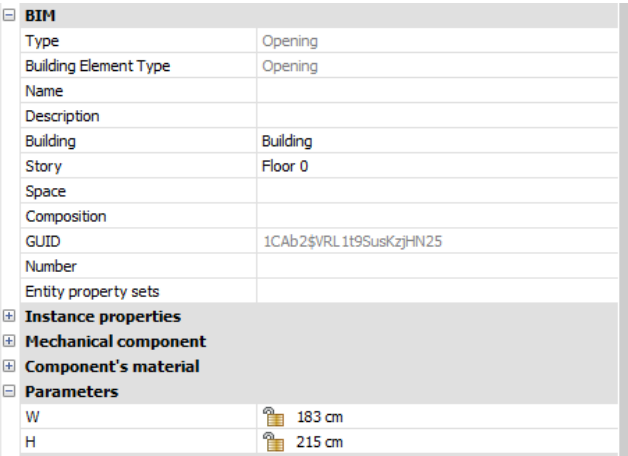


5 The opening is created based on the closed contour. In the **Structure Browser**, the opening is added as a block reference. The block reference is classified as an opening.





If you select the opening from the structure browser, you will see a set of properties related to this opening element in the **Properties** panel.



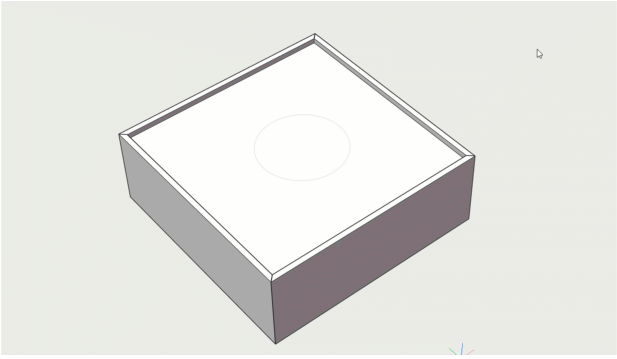
BIM	
Type	Opening
Building Element Type	Opening
Name	
Description	
Building	Building
Story	Floor 0
Space	
Composition	
GUID	1CAb2$VRL1t9SusKzjHN25
Number	
Entity property sets	
Instance properties	
Mechanical component	
Component's material	
Parameters	
W	183 cm
H	215 cm

Nota: If your closed contour has a rectangle shape, the opening element will include Width and Height parameters.

Nota: Opening elements contain subtractor solids. To display the subtractor solid, open the **Layers** panel and select the **BIM_Subtract** layer.

Nota: The associative tags can be generated using the BIMTAG command for the opening element in a sectioned drawing.

The following animated gif demonstrates how to create an opening based on a closed contour.



20.3.3 The library panel

20.3.3.1 Commands

LIBRARYPANELOPEN, LIBRARYPANELCLOSE, CREATELIBRARYBLOCK, -CREATELIBRARYBLOCK

20.3.3.2 About

The **Library** panel is a quick way to keep your BIM blocks at your fingertips or access predefined ones. You can add any CAD model to the **Library**. You can also fully customize the name and category to make it easier to find. You can then insert a library block in your drawing by dragging and dropping. Keep your library clean by deleting the items you no longer need.

20.3.3.3 Content of the Library panel

You can control the content of the **Library** panel in the **Option** menu.

Nota: When the COMPONENTSPATH system variable is empty (all paths deleted), the default **C:\ProgramData\Bricsys\Components** path is added automatically when you close and reopen the **Settings** dialog box.

20.3.3.4 Creating a library block

1 Open or create the drawing which contains the geometry for the new block.

2 Do one of the following:

- Click the **Home** icon ( ).

  This allows you to select the destination category; or you can create a new category.
- Click the **Category** thumbnail.

  The new item will be created in the selected category or in a subcategory of the selected category.
- Launch the CREATELIBRARYBLOCK command.
- The **Add block to library** dialog box displays.
- Select the entities or choose the **Entire drawing** option.

- Type a name for the block.
- (**Option**) Type a new category name or choose a category from the list.
- (**Option**) Click **Show Options** and set the block creation options.
- Click the **Create** button.

20.3.3.5 Managing library blocks

1 In the **Grid view** display, hover over the block icon.

2 Click the **Menu** icon (...).



3 (**Option**) Click the **File Open** icon (1) to open the block file.

Edit the file, then save the changes.

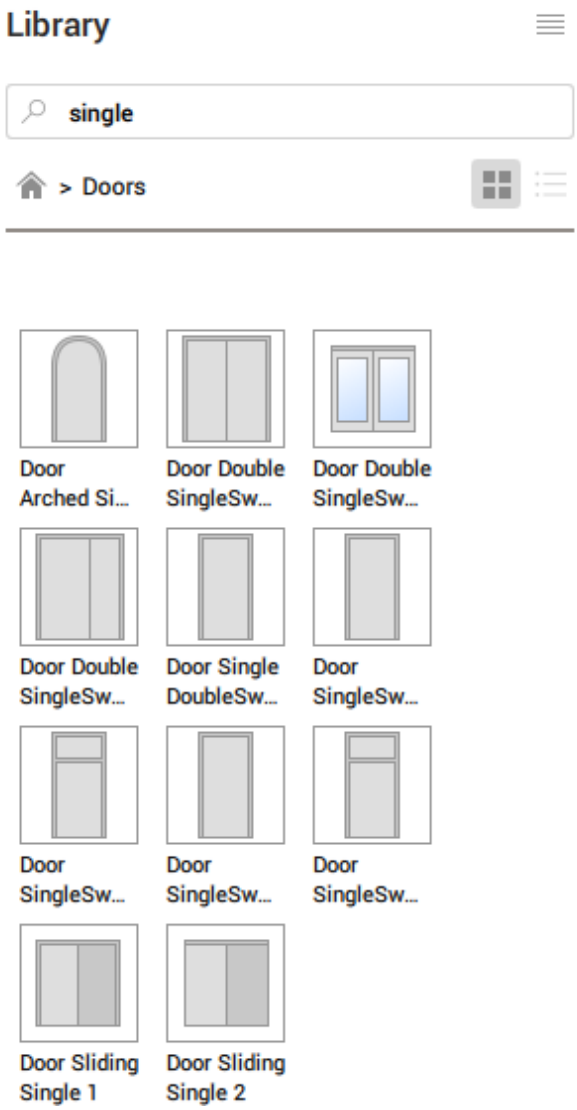
4 (**Option**) Click the **Delete** icon (2) to delete the block file.

Click **Yes** to confirm.

20.3.3.6 Searching the block libraries

Type a search string in the search field.

Only matching library blocks in the current category display.

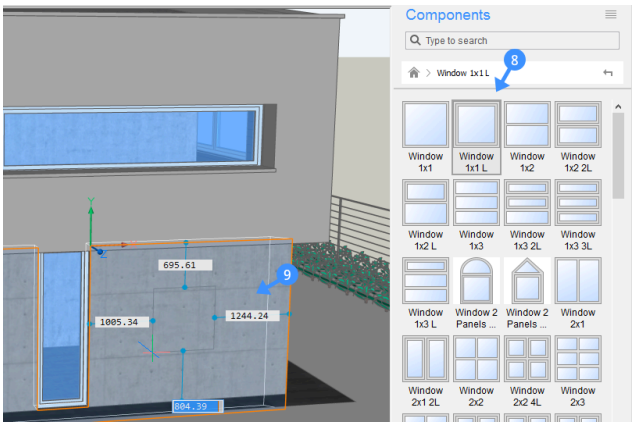


20.3.3.7 Inserting a BIM window into your 3D model

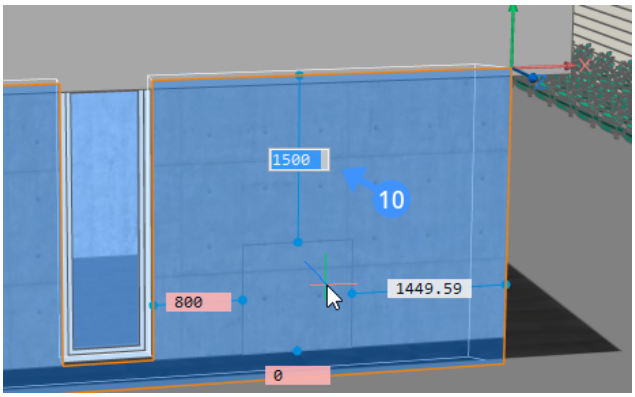
1 Click the **Windows** category tile in the **Library** panel.

2 Click a window (8).

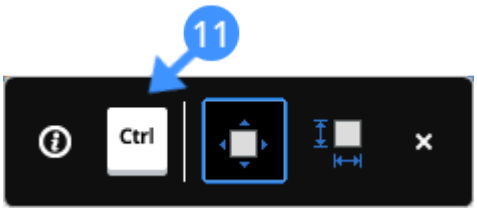
3 Drag and drop the window onto the wall (9).



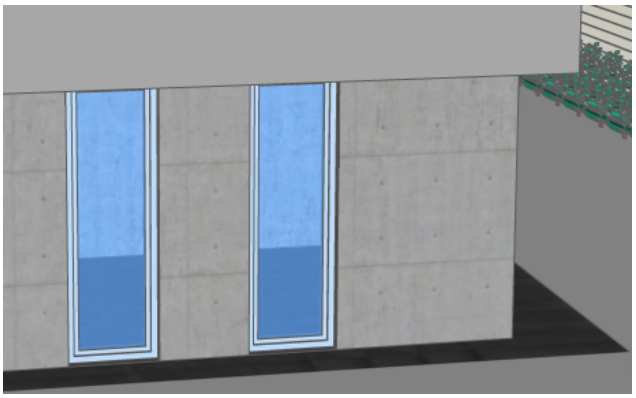
4 If Dynamic dimensions (**DYN**) is switched on, set the window location by entering the dimensions into the DYN fields (9). Press the Tab key to cycle through the DYN fields (10). A DYN field turns red to indicate the distance is locked.



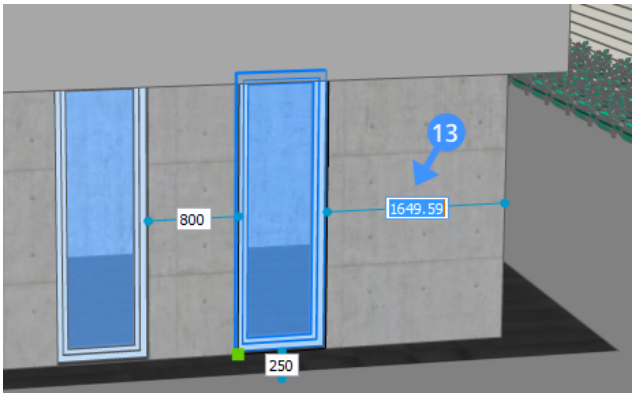
5 To edit the dimensions of the window press Ctrl (11) once and then use the Tab-key to cycle through the DYN fields (12).



6 Press Enter. The window is created. This same technique can be used for most of the blocks in the **Library**.



7 (**Optional**) Once you have a window you can adjust the position by selecting the window and then clicking on the dimension lines and editing the dimensions (13).



8 (**Optional**) To adjust the dimensions of the window after the window has been created, go to the **Properties** panel. Scroll down to the **Parameters** section and change the parameters as required. All windows and doors in the Library are parametric.

Parameters	
Name	Window_1x1_L:9
File	
Physical material	
W	800 mm
H	250 cm
Rebate	0 mm
PlacementDepth	90 mm
FrameThickness	60 mm
FrameDepth	50 mm
LeafThickness	45 mm
LeafDepth	60 mm
PaneOffset	20 mm
LeafOffset	20 mm

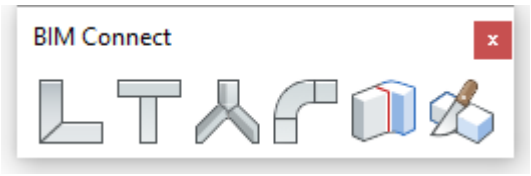
20.3.4 Connecting solids

20.3.4.1 Commands

LCONNECT, TCONNECT, BIMSTRUCTURALCONNECT

20.3.4.2 About

Walls, profiles, plies, etc. are all created as native 3D solids. So, any of the 3D Solids Editing commands can be used to modify them. Use **BIM Connect** commands to connect two solids in different ways.



The LCONNECT command creates an L-connection (bisector or parallel) between two solids.

The TCONNECT command creates a T-connection between two solids.

The BIMSTRUCTURALCONNECT command connects structural profile solids such as beams and columns.

The BIMFLOWCONNECT command creates a connection between flow segments, such as pipes or HVAC ducts.

The procedures below use the Quad cursor menu.

20.3.4.3 Procedure: creating L-connections

1 Make sure SELECTIONMODES system variable = 0.

2 Type **LConnect** in the Command line or select it via the Quad.

3 Select two solids or faces to connect.

4 The two selected solids are connected: the default connection type is L-bisector.

5 If HOTKEYASSISTANT system variable is ON (HKA is switched on), the Hotkey Assistant widget displays.



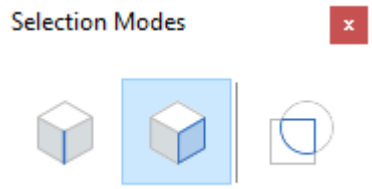
Use the CTRL key to cycle between the connection types **or**
You are prompted: Select base solid [Switch] <Accept>. Type **A**to use the selected connection, type **S** to switch to the desired connection type.
The 3D model updates dynamically when cycling the L-connection types.

Nota: L-connections can also be created between non-vertical solids such as roof slabs.

20.3.4.4 Procedure: Creating T-connections

The following procedure allows connecting minor faces of a solid to the major faces of other solids, regardless of what the orientation of the faces is. As a result, you can use TCONNECT command to T-connect solids.

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.

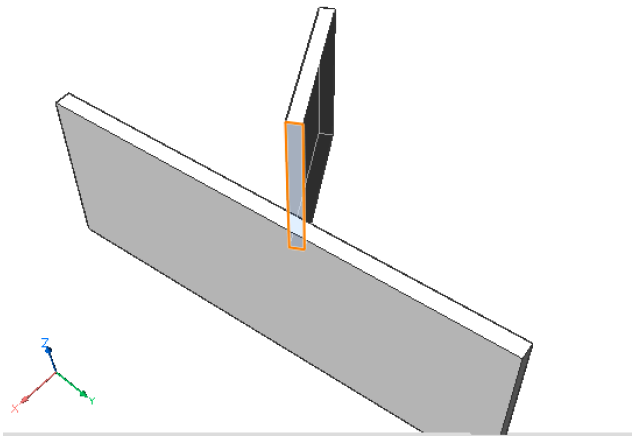


2 Type TCONNECT in the Command line or select it via the Quad.

3 Move the cursor over the minor face of the first solid at the end that needs to be connected.

4 Press Tab to highlight obscured faces.

Click when the face highlights.



5 (**Optional**) Repeat the previous step to select more faces, then press Enter.

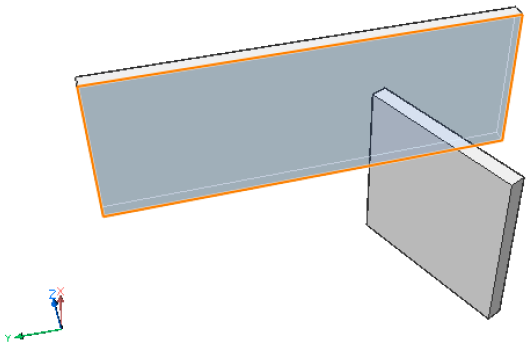
Dica: The SELECTALIGNEDFACES command in the **Select** command group in the Quad selects all faces in the 3D model that are in the same plane (coplanar) with a selected face.

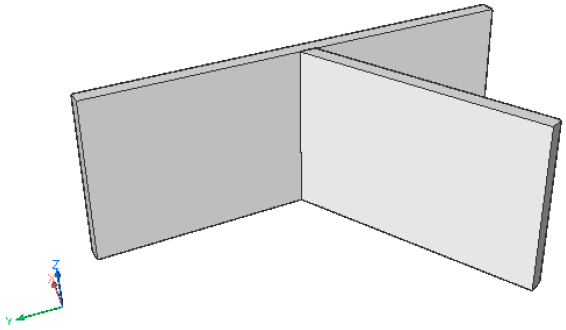
6 You are prompted: Select entities to connect to or [Connect to nearest/Disconnect/selection options (?)] <Connect to nearest>:

7 Do one of the following:

- Press Enter to connect to the nearest face(s).
- Select one or more faces.
- If necessary, press Tab to highlight an obscured face; click when the face is highlighted.

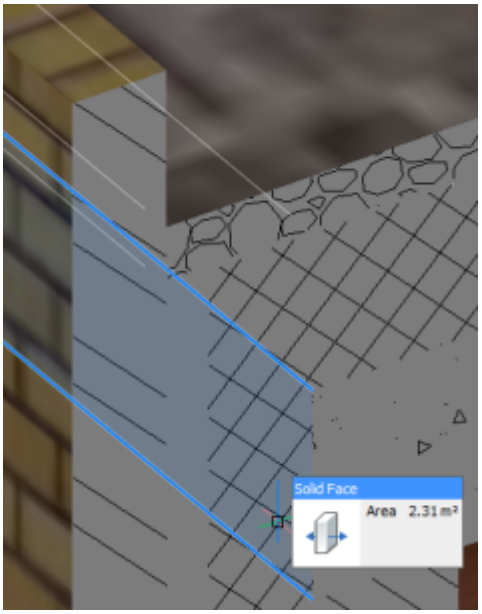
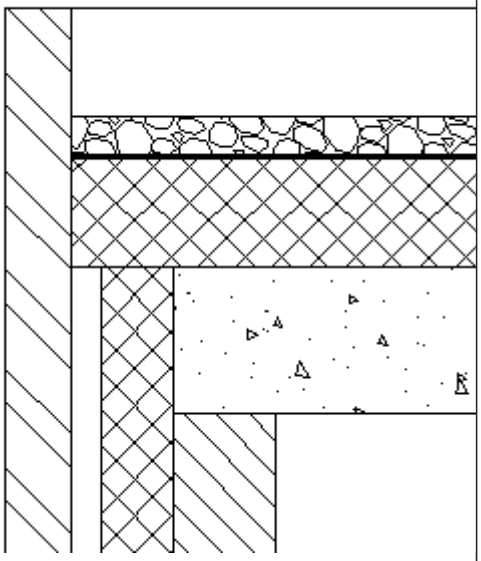
  Right click to stop selecting faces.





Dica: Hover over the minor face of the solid and choose **Connect with Nearest in the Model** command group of the Quad.

Creating T-connections ply-by-ply

	
3DModel	Calculated Section

To control the connection of composition plies between two building elements, you can **Push/Pull** a ply in a clipped BimSection (if Show Composition is ON). For more in-depth information, visit the BIM Ply Editing article.

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 Tab to select an obscured face.

2 When the ply face highlights, choose **Push/Pull** in 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.

5 The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

6 Optionally, use the BIMPROPAGATEPLANAR command to apply the connection to similar locations in the model.

20.3.4.5 Procedure: Setting the distance between parallel solids

1 Select the solid of which the position must be fixed.

2 Select the solid which will be moved when defining the new distance.

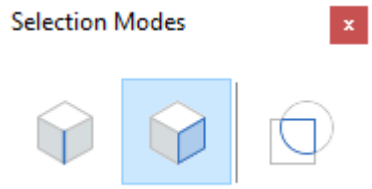
3 The nearest distance between the solids displays.

4 Type a value in the distance field and press Enter.

20.3.4.6 Procedure: Creating a copy of a solid

Creating a parallel copy of a solid

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over a face of the solid to be copied.

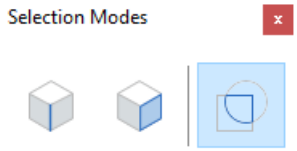
3 When the face highlights, choose **BimCopy** in the **Model** command group of the Quad.

A copy of the solid displays dynamically. The ruler indicates the current distance.

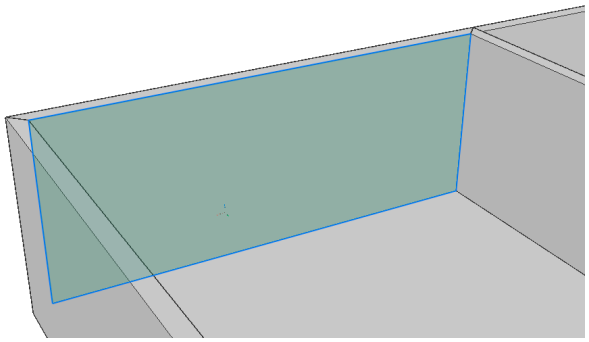
4 Type a value in the dynamic entry field and press Enter.

Creating a partial copy of a solid

1 Make sure the **Enable boundary detection** option of the SELECTIONMODES system variable is selected.



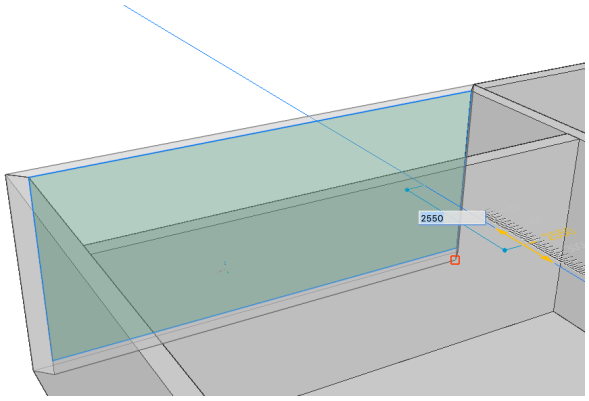
2 Highlight the part of the boundary for the part of the solid you want to copy.



3 When the face highlights, choose **BimCopy** in the **BIM** command group of the Quad.

A partial copy of the solid displays dynamically. The ruler indicates the current distance.

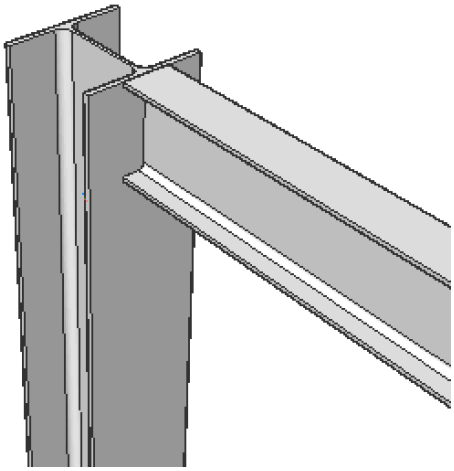
4 Type a value in the dynamic entry field and press Enter.



20.3.4.7 Procedure: disconnecting two connected solids

In some cases, you need to disconnect 2 solids of a node to do some further direct modeling.

You select two solids and it breaks the connection by separating the two solids physically and removes all end clippings at the side of the connection. As such, these solids can direct modeled upon again.



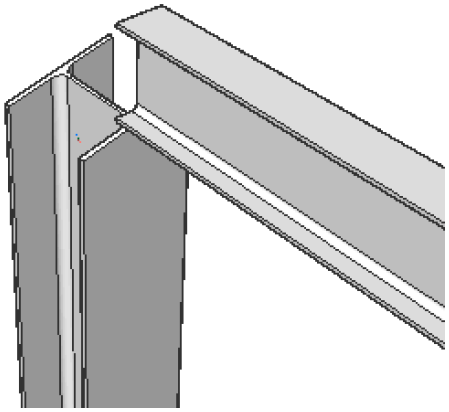
1 Type LCONNECT in the Command line.

2 Select the two solids.

3 The connection is automatically made, press CTRL until the disconnect icon is highlighted.



4 Press Enter.



Nota: You can use Propagate to disconnect similar nodes.

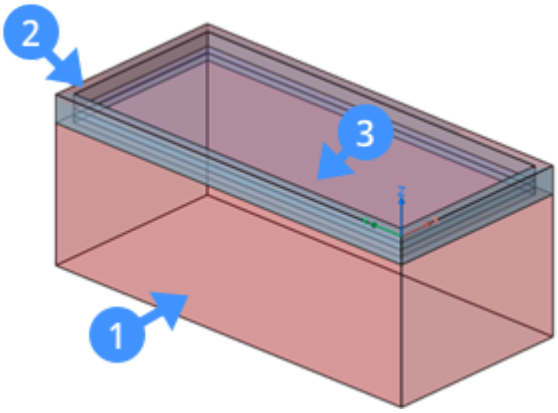
20.3.5 Creating a custom window

20.3.5.1 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 window at any time.

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)

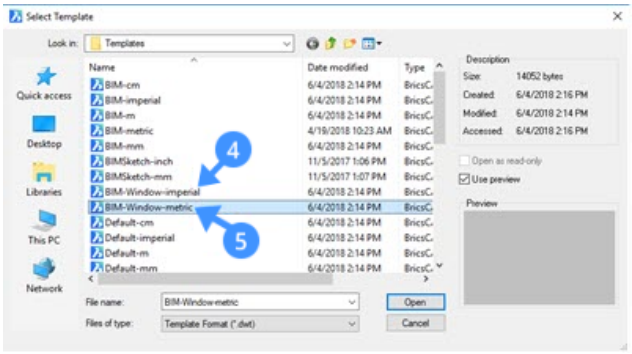


Step 1: Creating the file

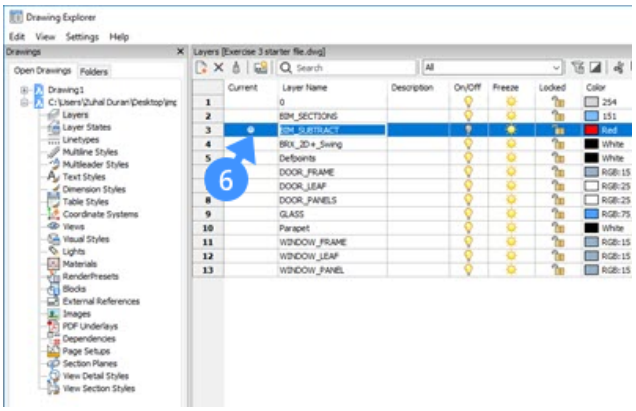
1 Click **New** in the **File** menu.

2 Select one of the pre-defined templates to create a simple window specifying the unit system of your drawing.

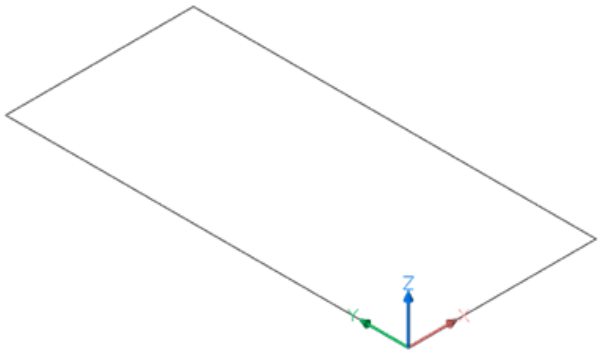
Nota: The pre-defined templates contain the layers needed to create the geometry of the window or door. These .dwt file format templates are **BIM-Window-imperial** (4) and **BIM-Window-metric** (5) which assure the required layers and properties are displayed in your drawing.



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.



4 Create a closed polyline (e.g. a rectangle) that represents the form of the opening for the insert.



5 Save the file in any location. The library of predefined windows and doors is located in the support folder.

6 By default this is: C:\Users<user_name>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support

\Bim\Components\Windows This path can also easily be found by typing SUPPORTFOLDER in the Command line.

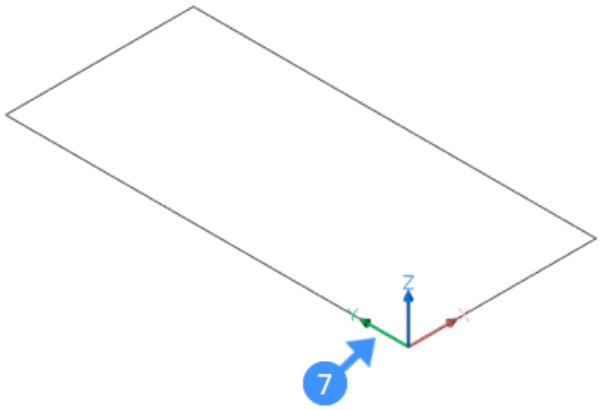
Step 2: Creating the subtractor solid

The subtractor solid will be used to create an opening.

Nota: **BIM Subtract** layer must be a current layer.

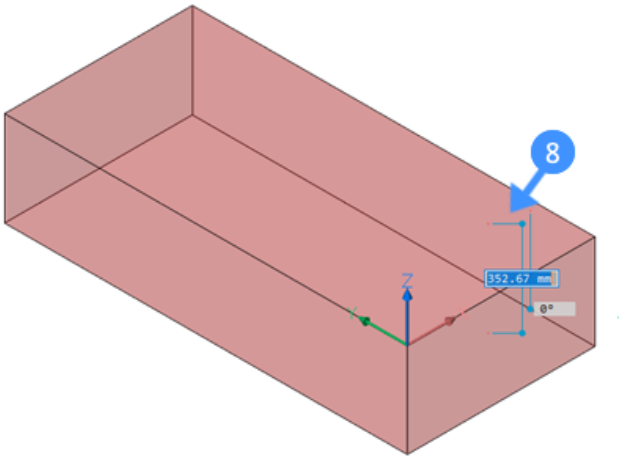
1 Use the outline of the solid which you have previously created (e.g. a rectangle).

1 The outline lies in the first quadrant of the XY plane of the World Coordinate System (WCS).



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.



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.

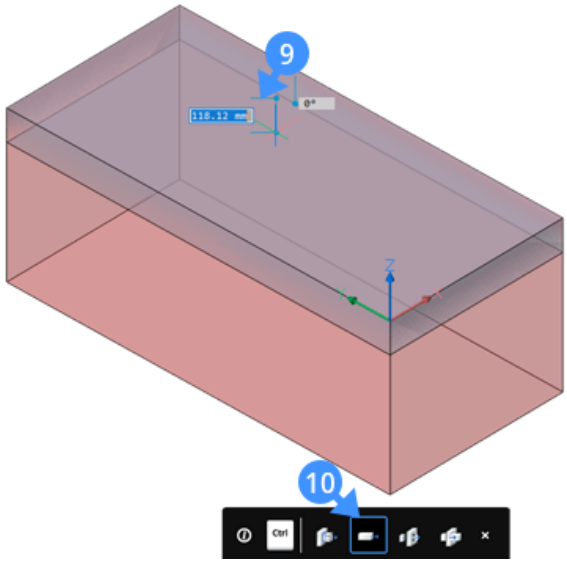
Step 3: Creating the fixed frame

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.

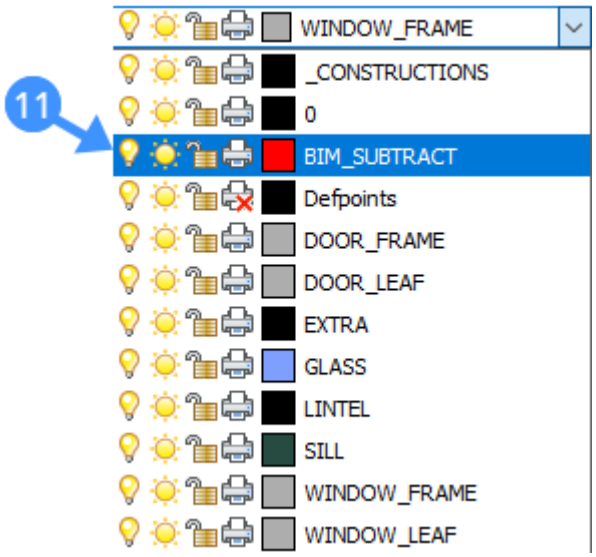
3 Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool 'Create' (10) option by pressing the CTRL key once.



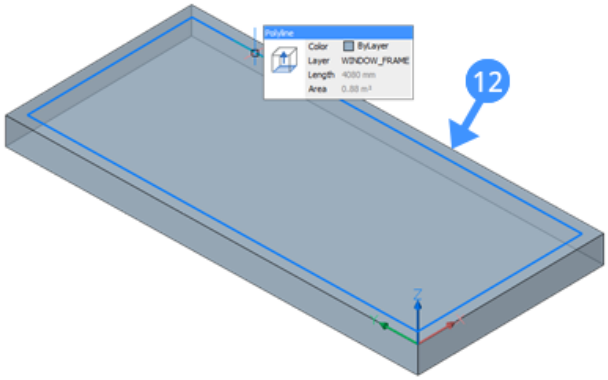
4 Type a value in the dynamic dimension field to specify the height of the fixed frame.

Nota: 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.

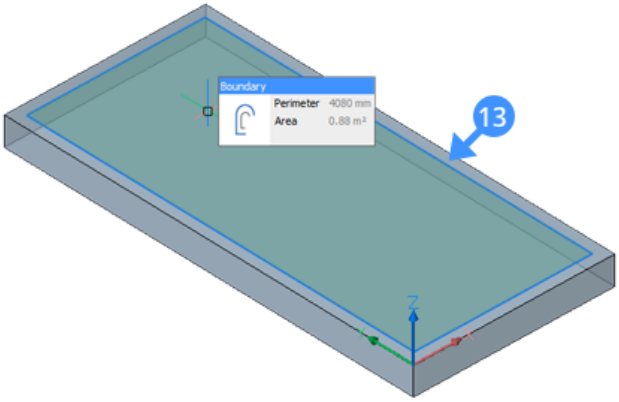


6 Use the **Offset** tool to create a parallel offset of the outline of the fixed frame (12).

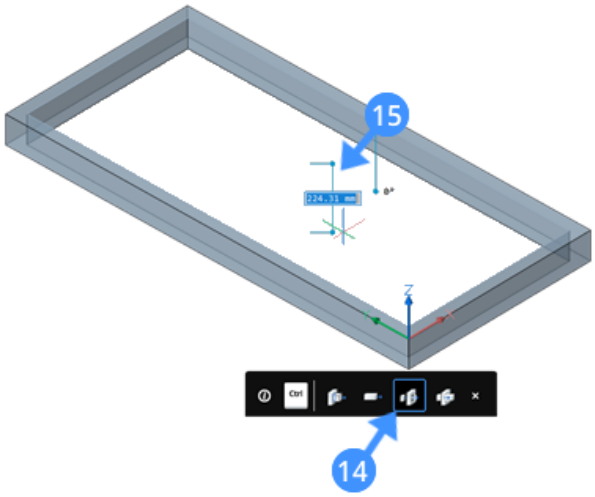


Nota: The distance of the offset equals the width of the frame.

7 Hover over the inside boundary of the parallel offset (13) and select **Extrude** tool from the Quad.



8 When the extrude mode is on the **Subtract** option (14) move your cursor downward (15) to create an opening in the fixed frame.



9 Do one of the following to finish the creation of the opening in the fixed frame.

- Click a point.

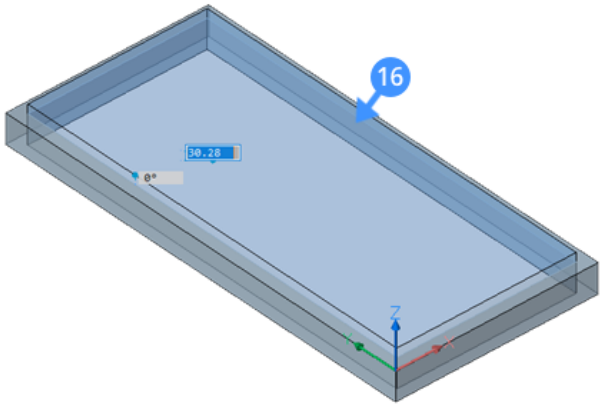
- Type a value in the dynamic dimension fields, then press Enter.

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.

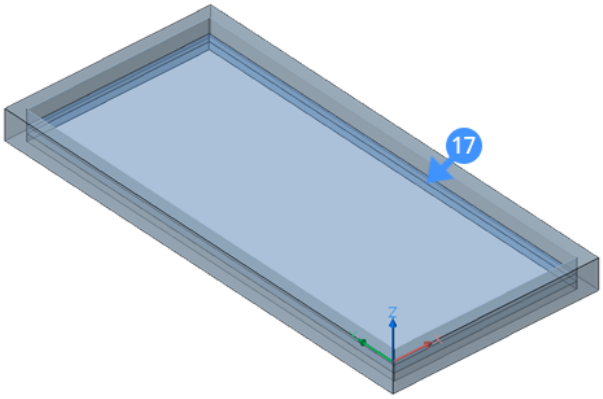
Nota: The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMExtrude to create a window glass pane.



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.

4 The following illustrations show the glass pane solid after moving it (17) down.



5 Optionally, before saving the drawing, classify the BIM as a window element.

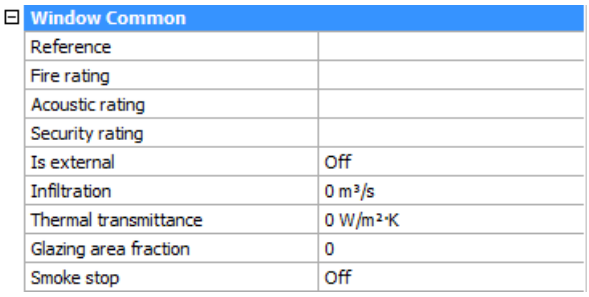
6 To do so, follow these steps:

- Type BIMCLASSIFY in the Command line.
- Press 'I' for window and press enter to accept it.
- Next, press 'D' for Drawing and press the 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.

7 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.



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

8 Save the drawing.

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.

20.3.6 Creating roofs

20.3.6.1 The roof creation workflow consists of the following steps:

1 Creating the roof footprint.

2 Extrude the footprint and create a hip roof (DMEXTRUDE command).

3 Adjust the slope of the roof faces if necessary (**Manipulator** widget).

4 Defining the thickness of the roof solid (**Body/Shell** option of the SOLIDEDIT command).

5 Create additional roof components.

20.3.6.2 Creating the roof footprint

You can create the roof footprint:

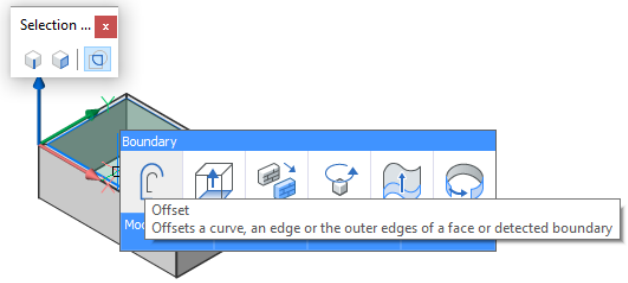
- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

To create the roof outline from the exterior walls boundary

1 Isolate the exterior walls. Isolating entities is easily done using the **Structure** Panel. See also the ISOLATEOBJECTS command.

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.

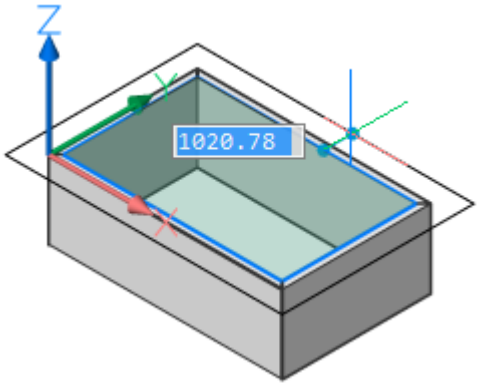
3 Set the **Enable boundary detection** ON in the **Selection Modes** toolbar or include the **Select detected boundaries** into SELECTIONMODES system variable.



4 Move the cursor to avoid area inside the exterior walls.

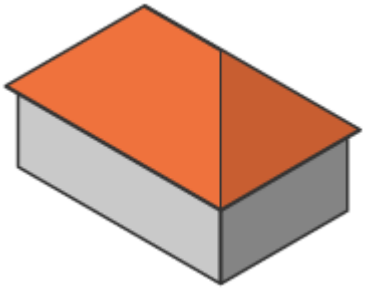
5 The inside boundary highlights.

6 Choose **Offset** in the **Draw** tab of the Quad.



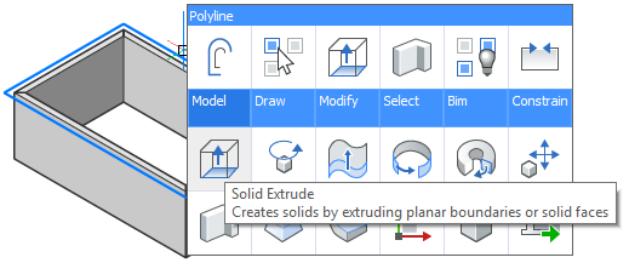
7 Do one of the following:
   - Type a value in the dynamic distance field.
   - Click to accept the value.

20.3.6.3 Creating a hip roof

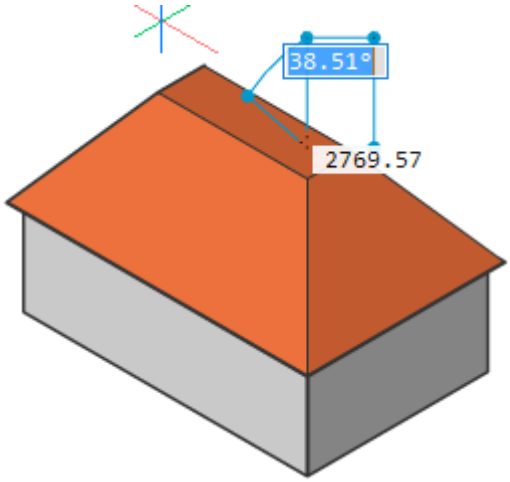


1 Create the roof footprint.

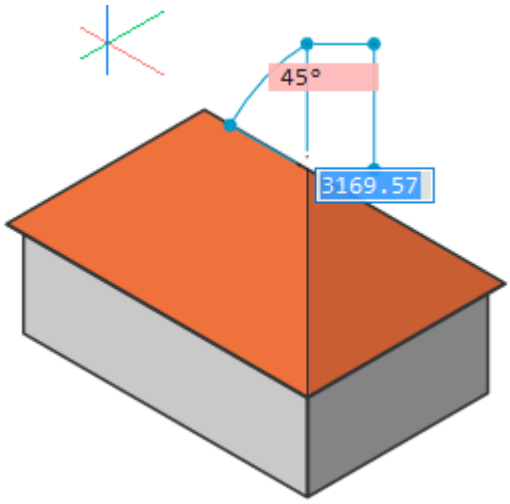
2 Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the Tab key.



4 Type a value in the **Taper Angle** field and hit the Tab key.



5 Do one of the following:

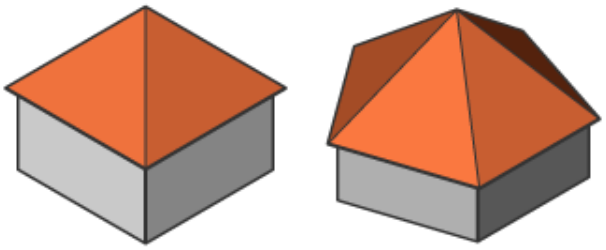
- Hit the Tab key and type another value in the **Taper Angle** field.

  The **Taper Angle field** is the complement of the inclination angle of the roof.

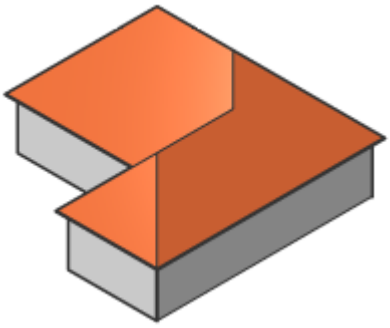
- Press Enter to create the roof.

6 Depending on the shape of the roof outline, the result is:

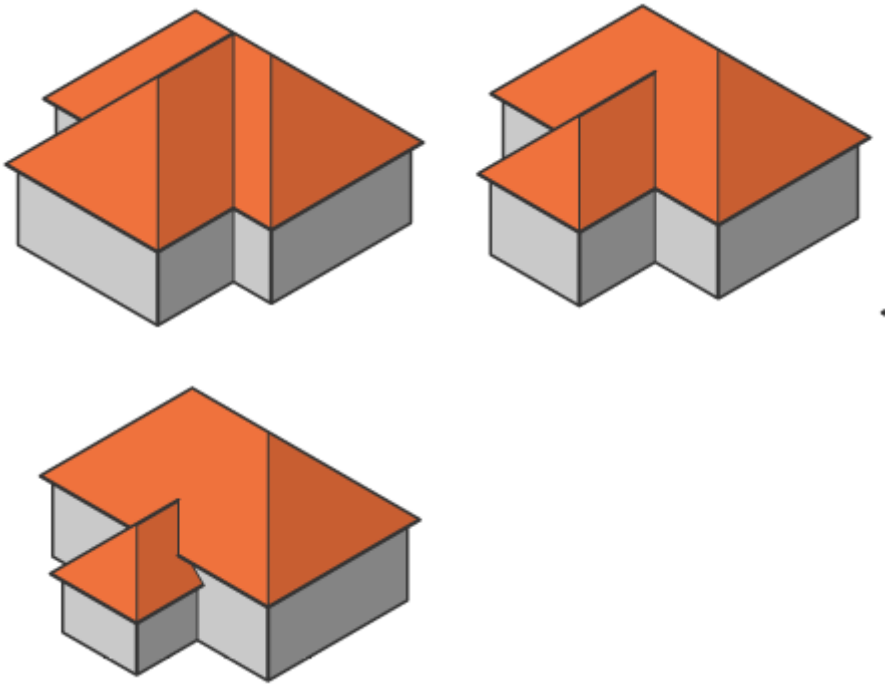
- Pyramid roof:



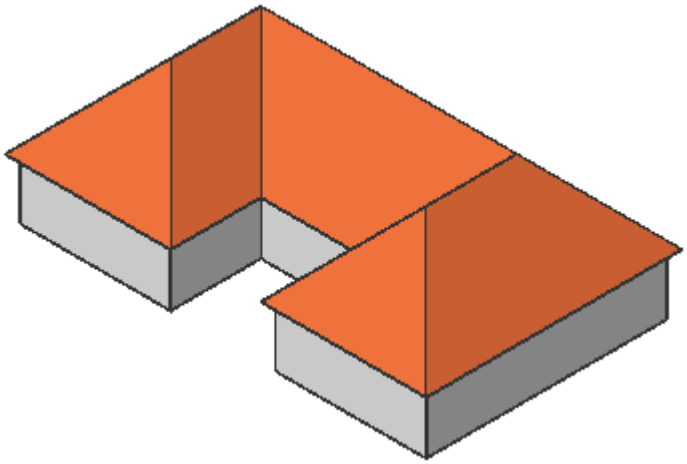
- L-shaped hip roof:

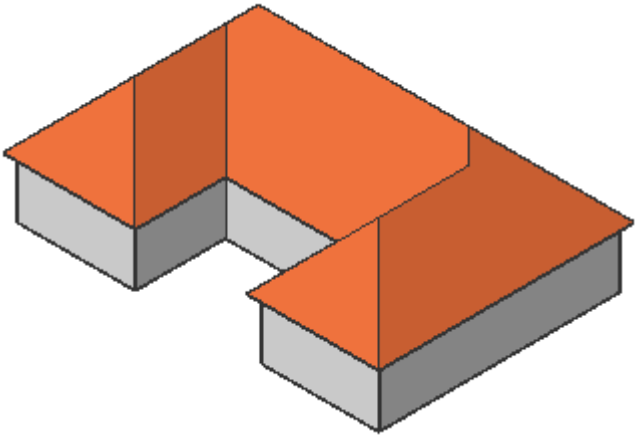


- T-shaped hip roof:

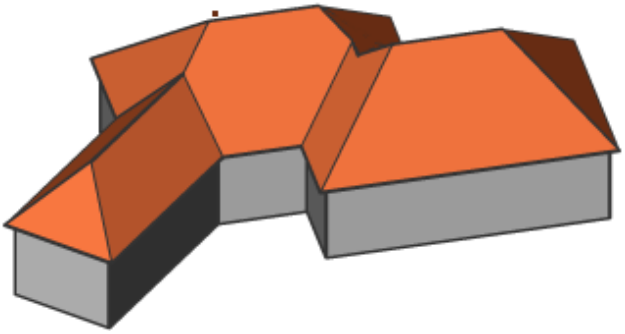


- U-shaped hip roof:





- Complex hip roof:



7 To define the thickness of the roof solid we will use the **Body/Shell** option of the SOLIDEDIT command.

Do one of the following:

a Choose **Shell** in the **Solid Edit | 3D Solid Editing** ribbon panel.
Or Choose **Shell** in the **3D Solid Editing** toolbar.

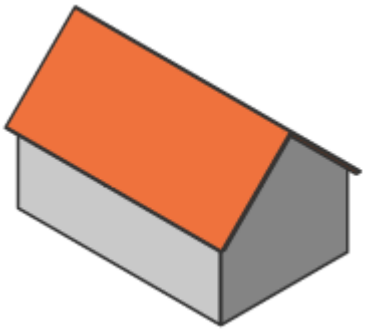
b You are prompted: Select 3D solid: Select the roof solid.

c You are prompted: Select face to remove or [Add/Undo/ALL]: Select the bottom face of the roof

solid.

d You are prompted: Enter the shell offset distance: Type a value to define the thickness of the roof solid.

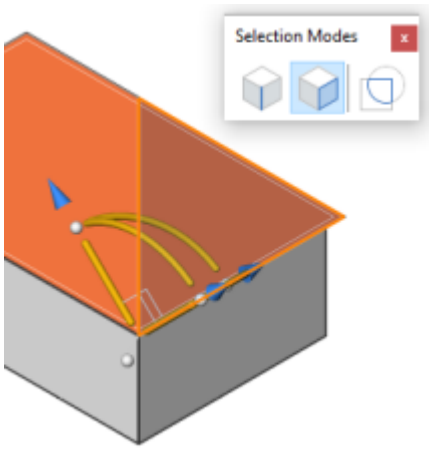
20.3.6.4 Creating a gable roof



1 Create a hip roof.

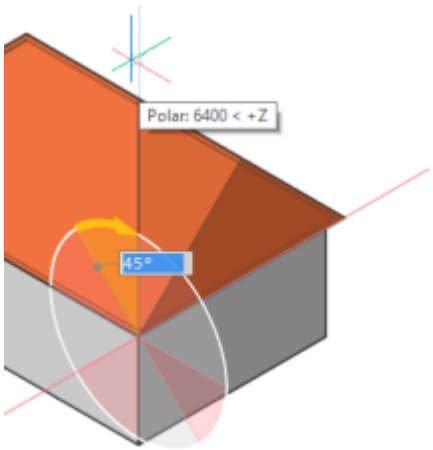
2 Make both end faces vertical:

- Move over the end face then hold down the left mouse button about half a second; the Manipulator widget displays (set the value of the MANIPULATOR system variable = 2).



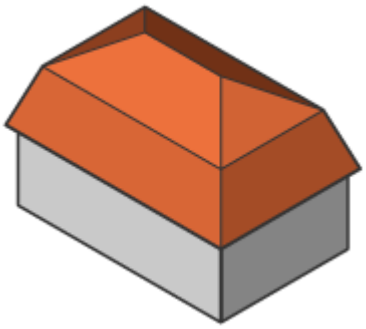
- Place the cursor over the vertical rotation arc. When the arc highlights, move the cursor.
- The selected face rotates dynamically.

  Click when the face is in a vertical position.



3 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

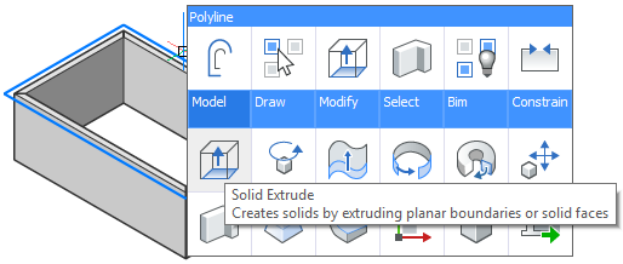
20.3.6.5 Creating a mansard roof



A mansard roof is composed of two solids, which we will create as two separate extrusions.

1 Create a roof footprint.

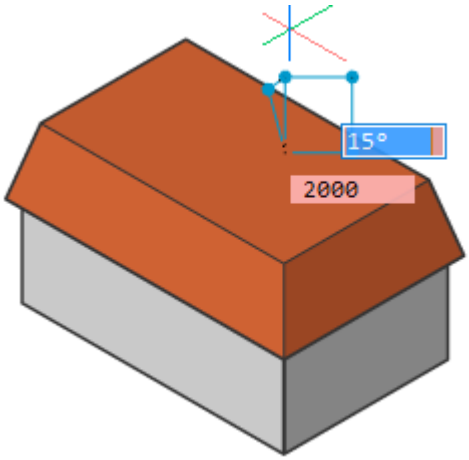
2 Move the cursor over the roof outline and choose to **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the Tab key.

4 Type a value in the **Taper Angle** field then hit the Tab key.

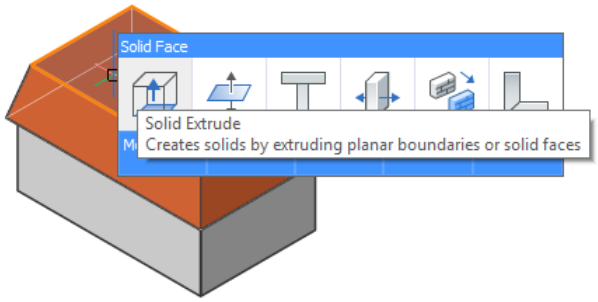
The **Taper Angle** field is the complement of the inclination angle of the roof.



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.

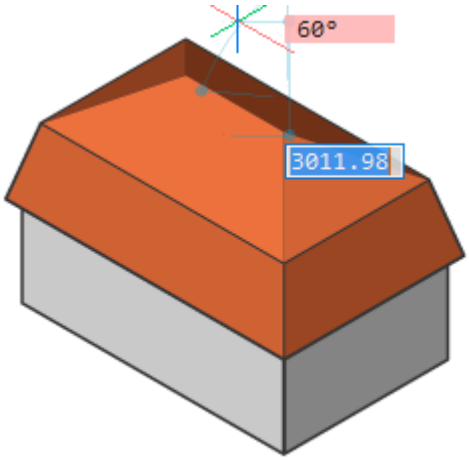
6 Optionally hit the Tab key to go back and forth between the entry fields and press Enter when you are satisfied with the result.

7 Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad.



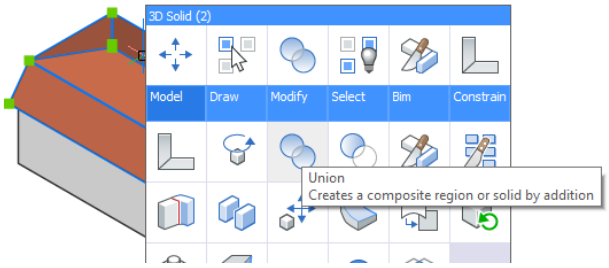
8 Move the cursor vertically, then hit the Tab key.

9 Type a value in the **Taper Angle** field then hit the Tab key.



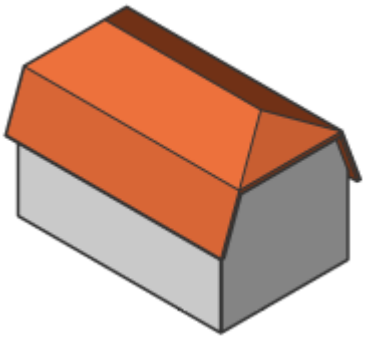
10 Do one of the following:
- Hit the Tab key and type another value in the **Taper Angle** field.
- Press Enter to create the first solid.

11 Select both solids and choose **Union** in the **Model** tab in the Quad.



12 Repeat step 7 of the Create a hip roof procedure.

20.3.6.6 Creating a jerkinhead roof



A jerkinhead roof is composed of two solids, which we will create as two separate extrusions.

1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

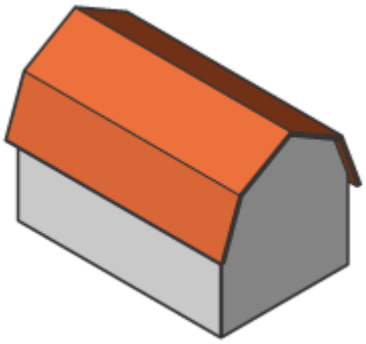
2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

3 Execute steps 7 to 11 of the Creating a mansard roof procedure.

4 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

20.3.6.7 Creating a gambrel roof

A gambrel roof is composed of two solids, which we will create as two separate extrusions.



1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

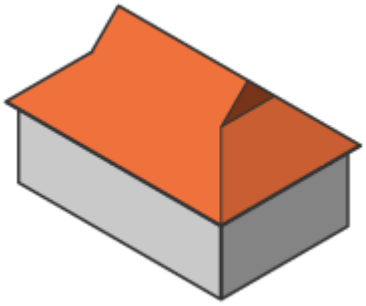
3 Execute steps 7 to 10 of Creating a mansard roof procedure to create the second solid.

4 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

5 Execute step 11 of Creating a mansard roof procedure.

6 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

20.3.6.8 Creating a dutch gable roof

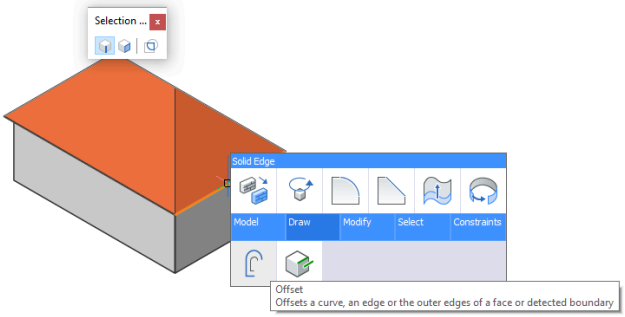


A Dutch gable roof is based on a hip roof.

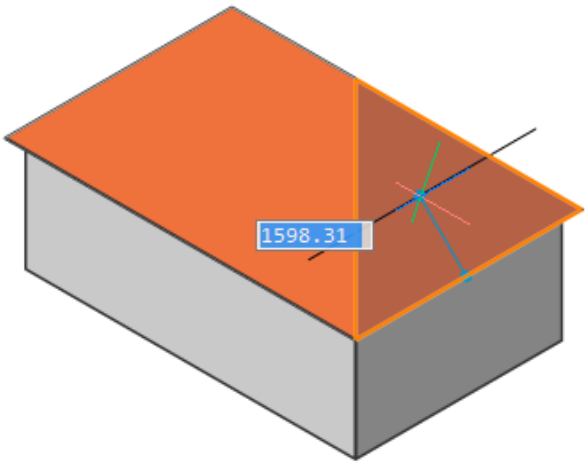
1 Create a hip roof.

2 Set the **Select Edges** option of the SELECTIONMODES 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.



4 Move the cursor and type a value in the dynamic distance field or click a point.



5 Do one of the following:
   - Choose **Imprint** in the **3D Solid Editing** toolbar.
   - Choose **Imprint** in the **Solid Edit | Solid Editing** ribbon panel.

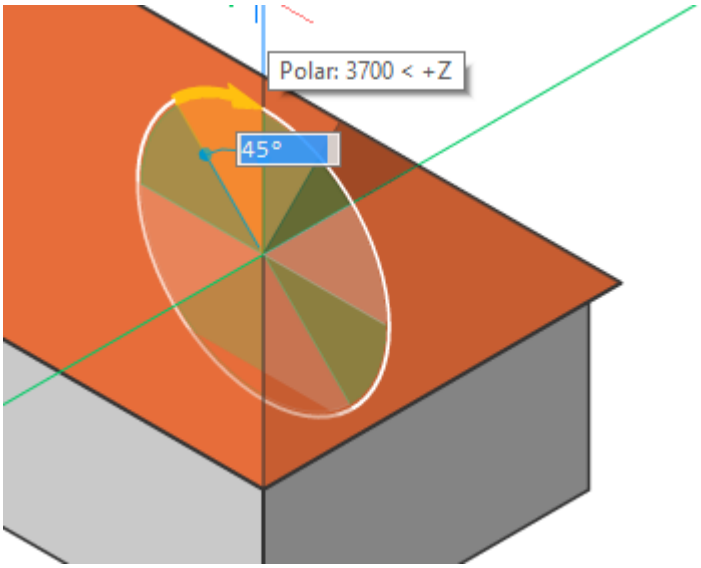
6 You are prompted: Select 3D solid, surface or region: Select the roof solid.

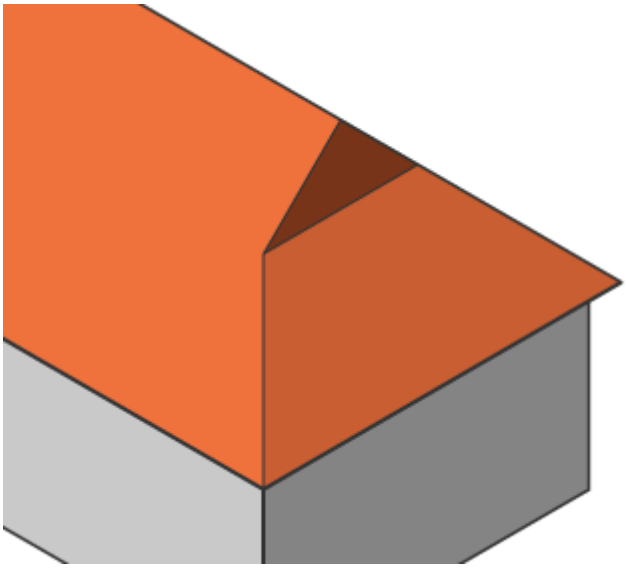
7 You are prompted: Select an entity to imprint: Select the offset roof edge.

8 You are prompted: Delete the source object[Yes/No] <No>: Choose Yes.

9 A triangular face is created.

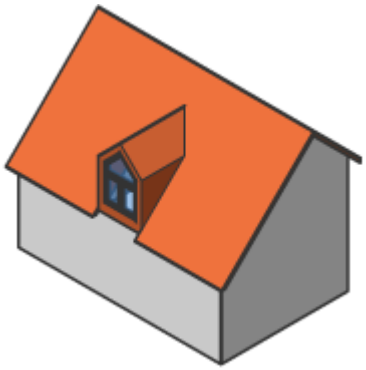
10 Execute step 2 of Creating a gable roof procedure to set the triangular face vertical.





11 Execute step 7 of the Create a hip roof procedure to set the thickness of the roof solid.

20.3.6.9 Creating dormers



The procedure to create a dormer 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 a dormer roof.

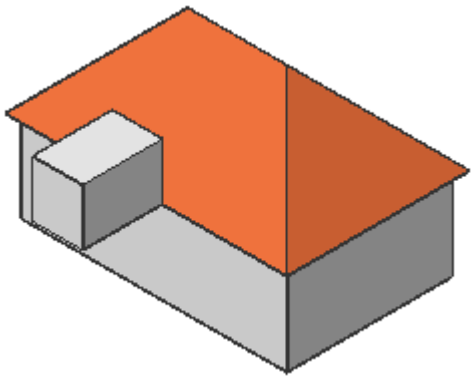
1 Right-click, then choose **Box** in the **Model** tab of the Quad.

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.

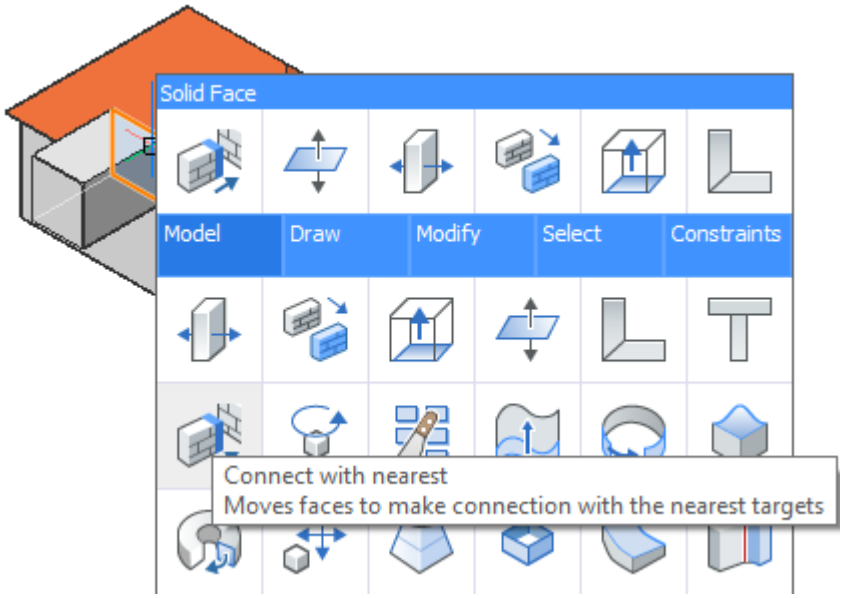
3 You are prompted: Set opposite corner or [Cube/Length of side]:

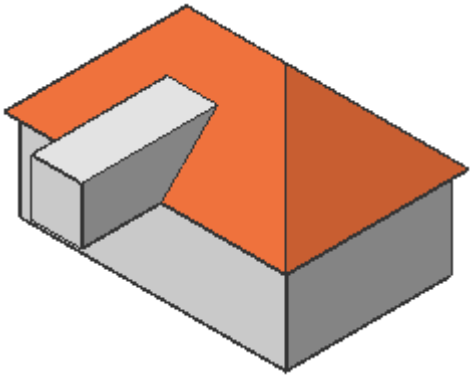
4 Click a point, then define the height of the cube: either click a point or type value.



5 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).

6 Choose Connect with nearest in the Model tab in the Quad.



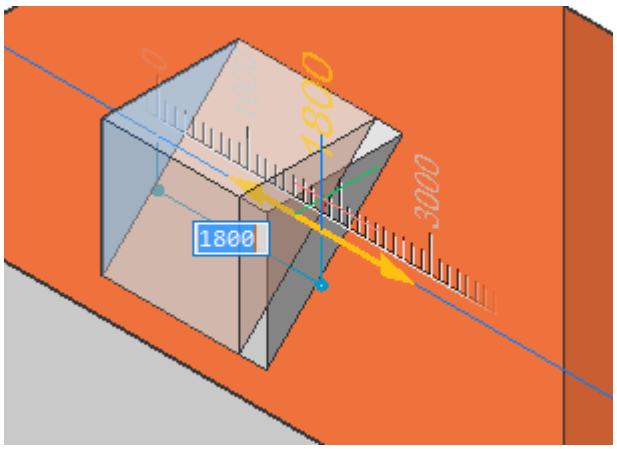


7 Select the front face of the box, choose **Push/Pull** in the **Model** tab in the Quad and move the face to the roof plane.

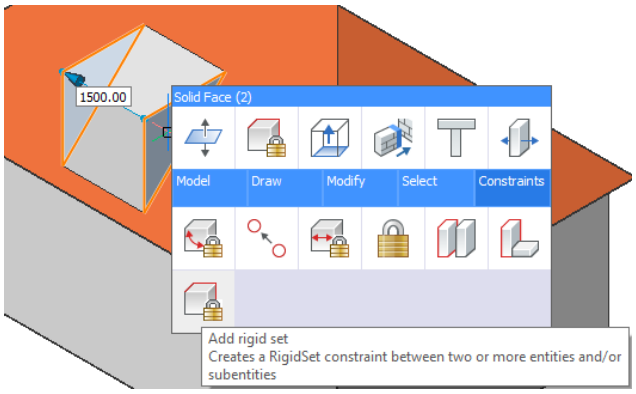
8 Use PUSHPULL command on the front, top and side face of the dormer solid to define the dimensions

and the position of the dormer on the roof:

- Select a side face, then hit the Tab key to define the width.



- Select both side faces, then choose to **Add rigid set** in the **Constraints** tab in the Quad to fix the width.



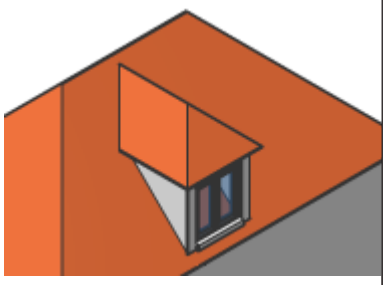
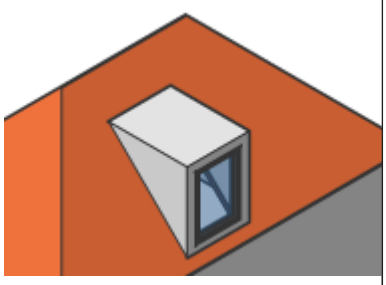
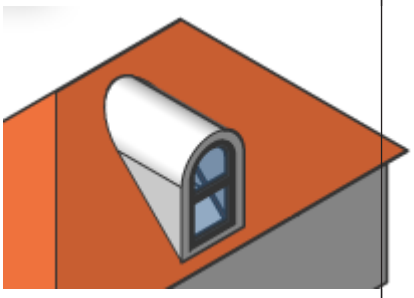
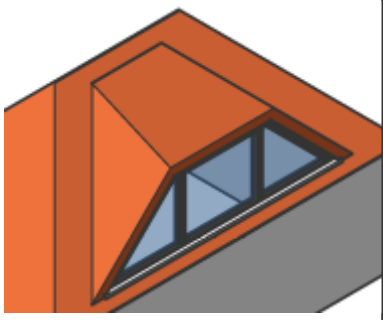
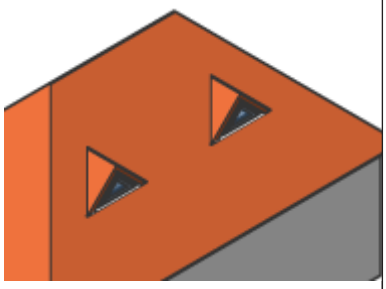
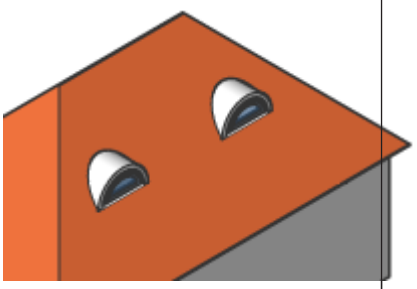
- Drag one of the side faces to position the dormer. Do one of the following:
  - Type a distance in the dynamic entry field.
  - Click to accept the value.
  - Move the cursor over a face that is parallel to the face being dragged and hit the Tab key. The distance field now shows the distance to this 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.

9 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 backward until the front face is triangular.

		
Hip	Flat	Vault
		
Special	Triangle	Eyebrow

20.3.7 Creating slabs

20.3.7.1 Commands

DMEXTRUDE, POLYSOLID

Nota:

The DMEXTRUDE command allows to extrude a closed 2D entity such as polylines, circles, ellipses and extrude a detected boundary, enclosed by walls and/or 2D entities.

The POLYSOLID command allows to create pitched roofs.

20.3.7.2 Creating floor slabs between walls

Make sure that:

- The **Enable Boundary Detection** option of the SELECTIONMODES system variable is ON.
- The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system.
- If necessary, move the origin of the coordinate system.
- Dynamic dimensions are ON.

1 Move the cursor inside the surrounding walls.

2 When the boundary highlights, choose **Extrude** in the **Model** command 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.

20.3.7.3 Creating a pitched roof slab

Make sure that:

- **Dynamic UCS** is ON.
- **Dynamic dimensions** (DYN) is ON; it is recommended to have the **Tracking dynamic dimensions** option of the DYNMODE system variable set.
- **Entity Snap Tracking** (STRACK) is ON.
- **Endpoint entity snap** is ON.

1 Launch the POLYSOLID command.

1 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.

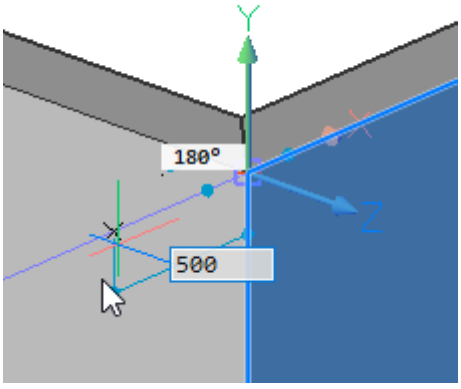
3 When the face highlights, hit the Shift key to lock the **dynamic UCS**.

4 Do one of the following:

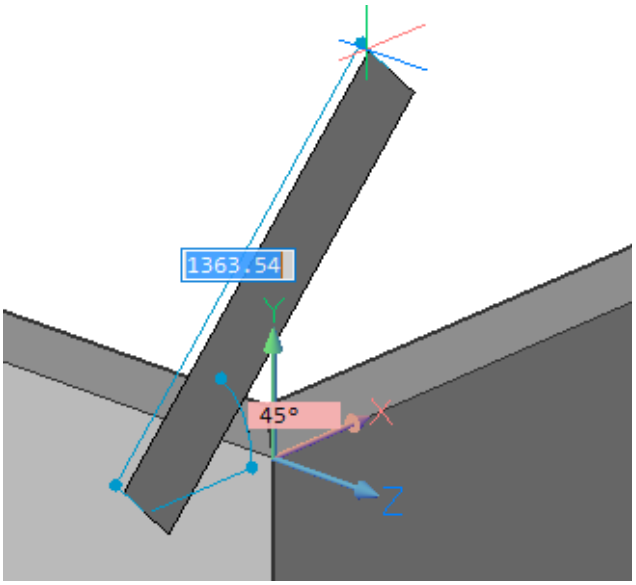
5 To create the roof at the top edge of the wall, snap to the exterior top corner of the wall.

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.



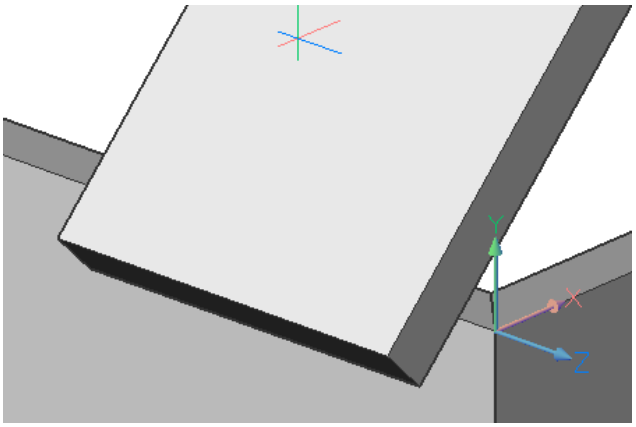
6 Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.



7 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.

8 The roof slab is extruded dynamically.

9 Specify a point to define the length of the extrusion.



10 Repeat the previous steps to create more roof slabs.

11 Use CONNECTWITHNEAREST to connect the walls to the roof slabs.

20.3.8 Creating walls

20.3.8.1 Commands

POLYSOLID, LCONNECT, TCONNECT, BIMQUICKDRAW

20.3.8.2 About the Polysolid tool

The **Polysolid** tool allows you to create wall solids by selecting a start point, an endpoint and entering a height. Note that, in BricsCAD BIM, you always start by creating the geometry first and adding materials or compositions later. This way there is no need to worry about details when modeling a preliminary design.

The use of Dynamic Dimensions and Polar Tracking is recommended. Dynamic Dimensions can be used to specify the length of the wall and Polar Tracking controls the direction.

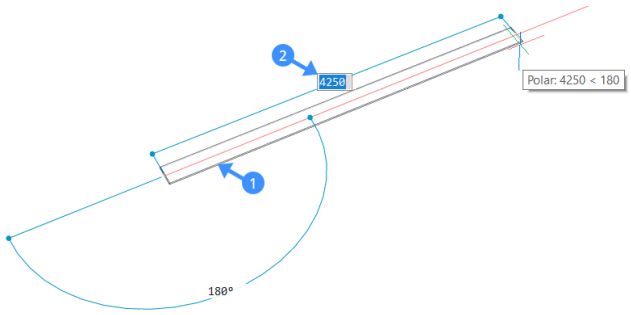
For more information about this command, visit the Command Reference article POLYSOLID.

20.3.8.3 Procedure: Create a simple wall

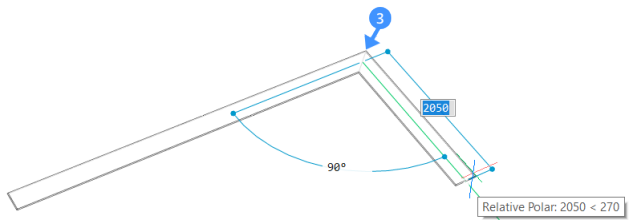
1 Launch the **Polysolid** tool.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

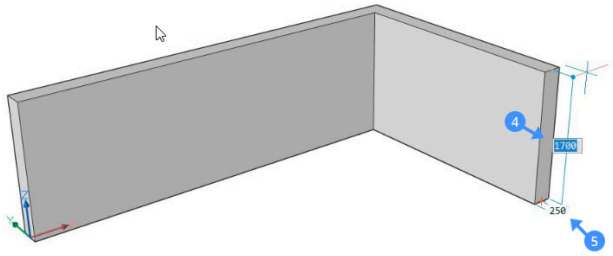
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



4 Select an endpoint or enter a value in the dynamic input field and press Enter. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.



5 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing Tab.



6 To change the justification of the wall hit CTRL while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.



Nota: The width and height of the previous wall will be the new default values of the next Polysolid.

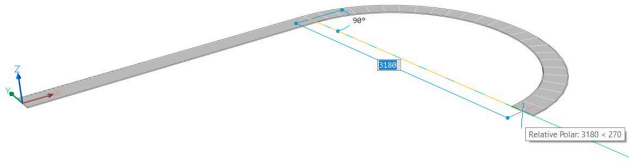
20.3.8.4 Procedure: creating a curved wall

1 Launch the **Polysolid** tool.

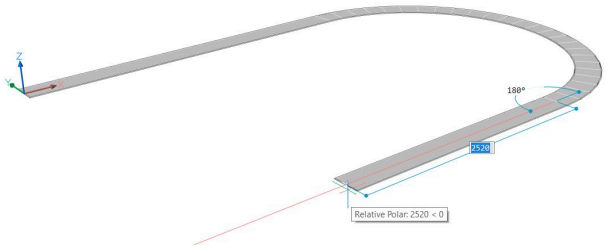
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press Enter.

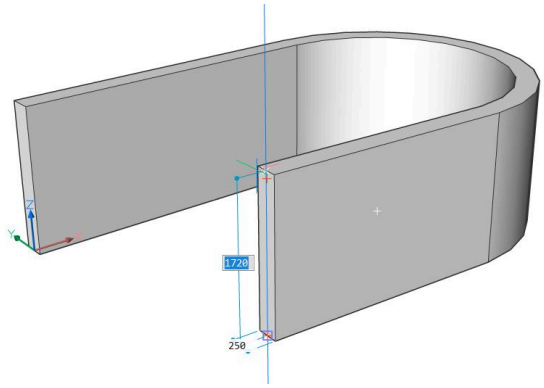


5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.



6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width

field. You can switch between the two dynamic input fields by pressing Tab.



20.3.8.5 Procedure: using an existing 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 prompts menu. Using the **Entity** option in **Polysolid** enables you to create 3D geometry from a 2D layout.

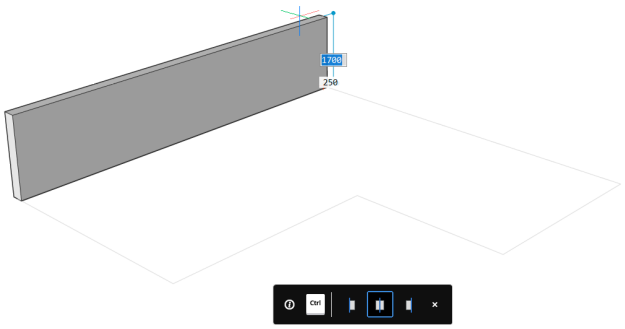
2 Choose a 2D linear entity which will be the Polysolid base.

3 Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 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.

5 The current width of the wall appears in the Width Field. Press Tab switch between the Height and the Width field.

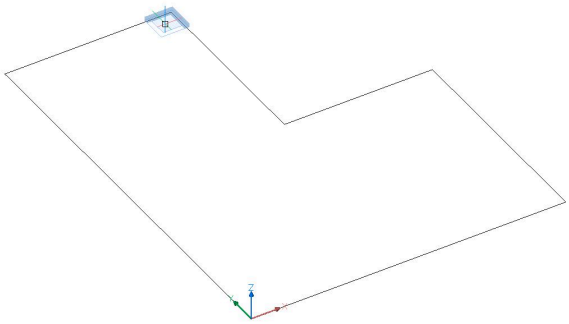
6 The Hotkey Assistant appears and displays the possible justification options. Press CTRL to cycle through the wall justification options.



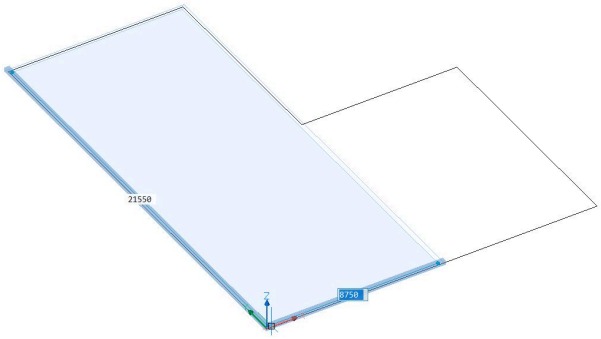
It is also possible to use an existing 2D plan to create walls using the **BimQuickDraw** tool:

1 Open the **Quickdraw** tool from the Quad or type Quickdraw in the Command line.

2 The cursor automatically snaps to the lines of your 2D drawing. Place your cursor where you want to place your first corner and left-click.



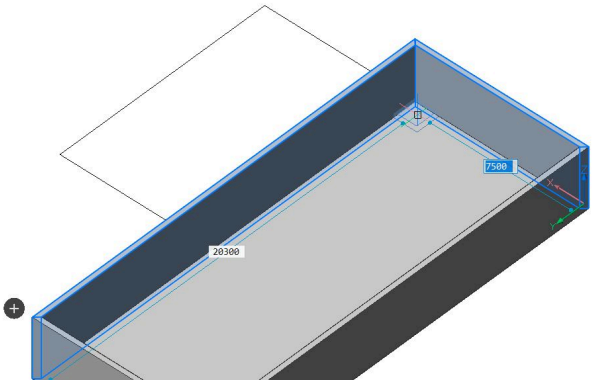
3 Hover your mouse to the opposite corner and left click again or type the values of the lengths of the sides in the respective fields and press Enter.



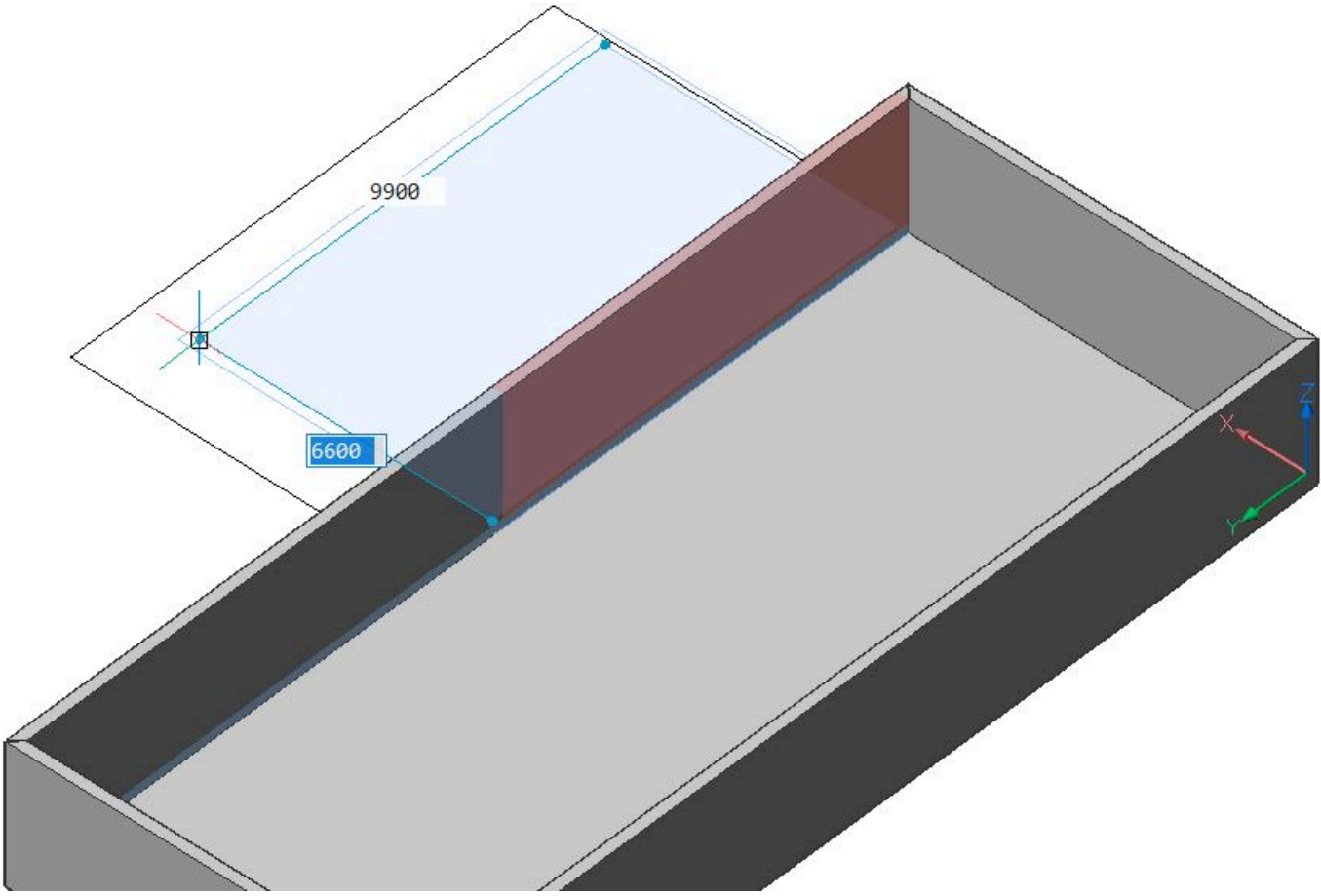
4 The **Quickdraw** tool always uses the default height and width settings.

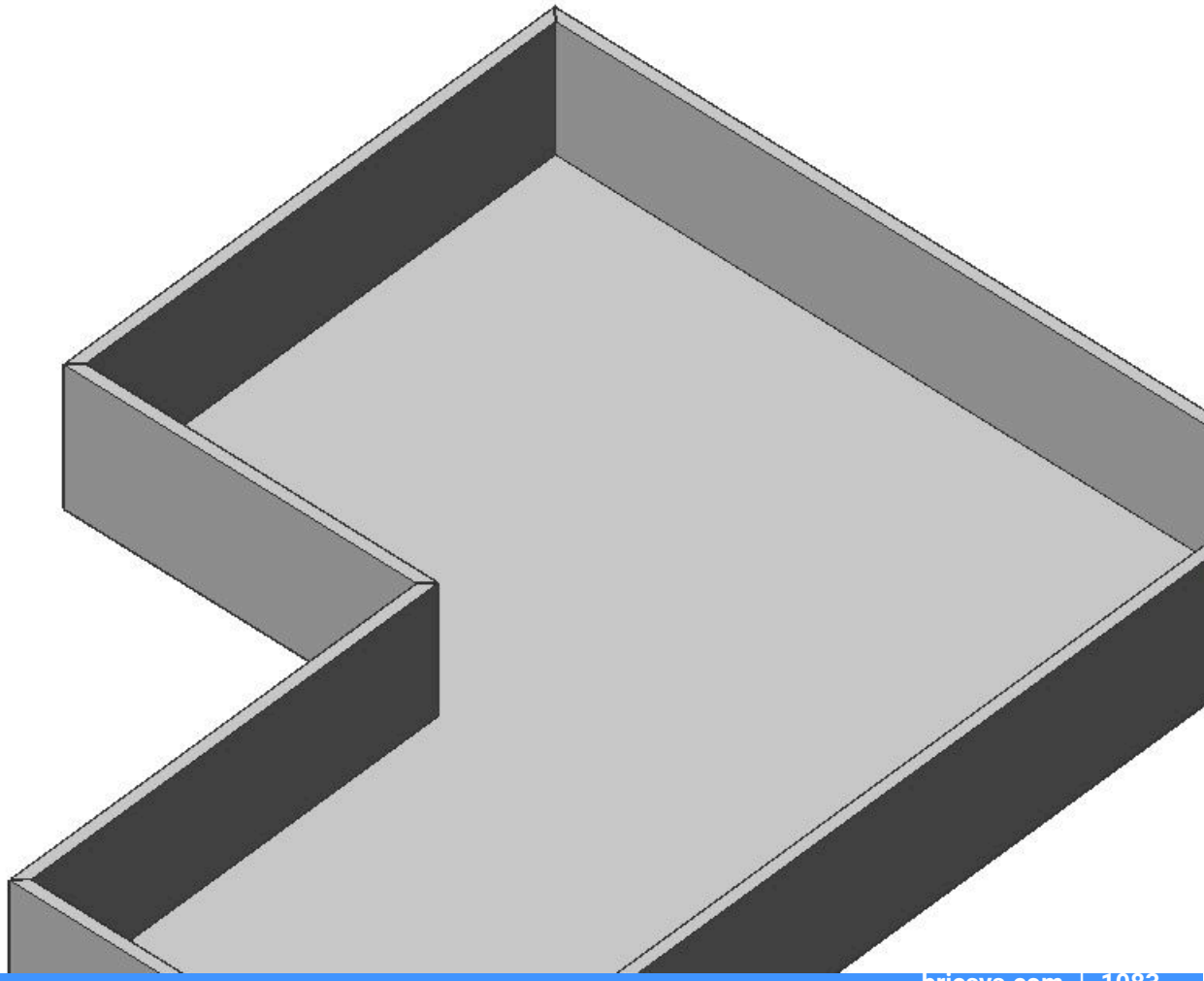
To change these values type S and press Enter to open the Quickdraw Settings. Change the values in this menu.

5 To finish the L-shaped form, you can draw a new rectangular on the second part of the figure. The common wall of the two rectangles will automatically be removed if you draw a second rectangular starting from the interior side of the first rectangular as indicated below:



6 While drawing the second rectangular, you can see the wall that will be deleted is displayed in red.


9900
6600
Z
X
Y



20.3.8.6 About the BimConnect tool

The **BimConnect** tool allows you to connect one wall to another element.

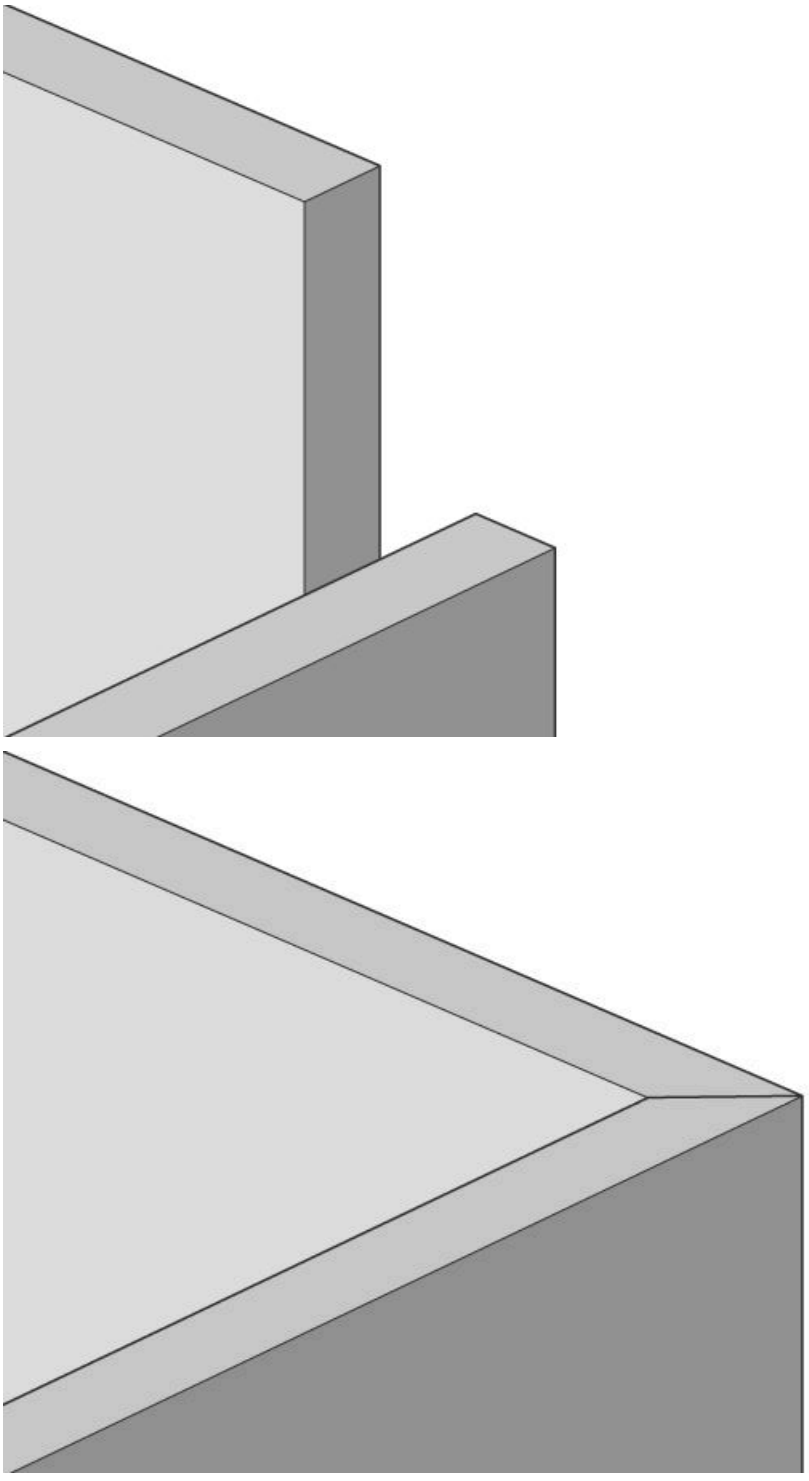


For more information about this command, visit the Command Reference article LCONNECT.

20.3.8.7 Procedure: connecting two walls

1 Select two wall elements and apply the **LConnect** tool.

When two walls intersect, a mitered connection will be created by default.



2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

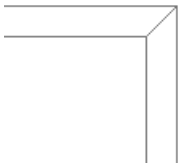
3 The selected connection type is indicated by a blue frame.



The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.
Switch between the options by pressing CTRL.

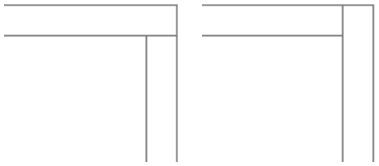
- **Bisector L-connection (miter)**

  When the **bisector L-connection** is selected, the walls are connected at an angle of 45°.



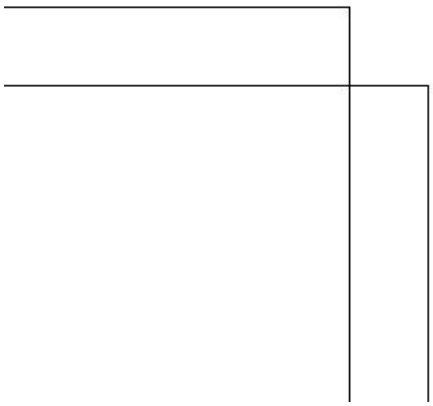
- **Parallel L-connection 1 Parallel L-connection 2**

  When the L parallel type 1 or type 2 connections are selected, the walls are connected at an angle of 90°.



- **Disconnection**

  When the disconnected option is selected, the existing connection between the walls is removed.



Press Enter to accept the current connection type or press CTRL to cycle through the possible connection options.

20.3.8.8 Connecting walls to other elements

The **Connect with Nearest** tool allows you to connect the minor face of a wall to the major face of another element, regardless of the orientation of the faces. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or the side face of a slab to a wall.

To connect the face to the nearest object, select the face of the object that you would like to connect with the nearest object and choose the Connect With Nearest tool in the Quad.

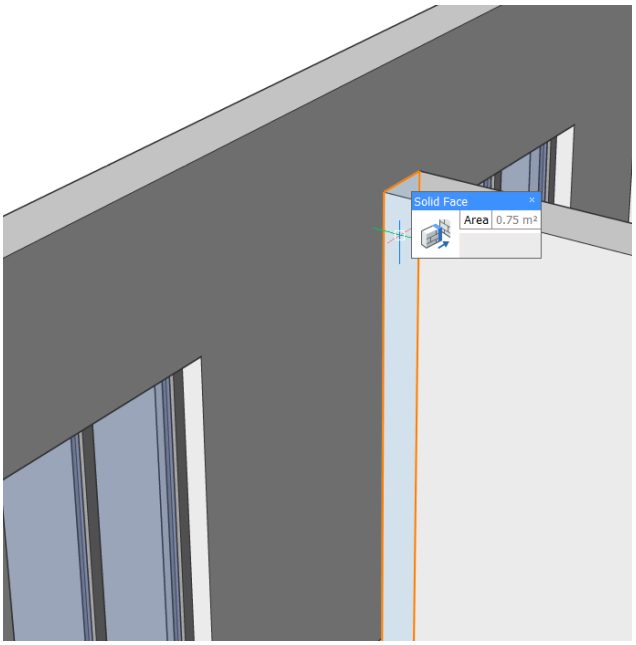
Nota: Before using the **Connect with Nearest** tool, make sure the **Select Faces** option is enabled in the selection mode settings.



For more information about this command, visit the Command Reference article TCONNECT.

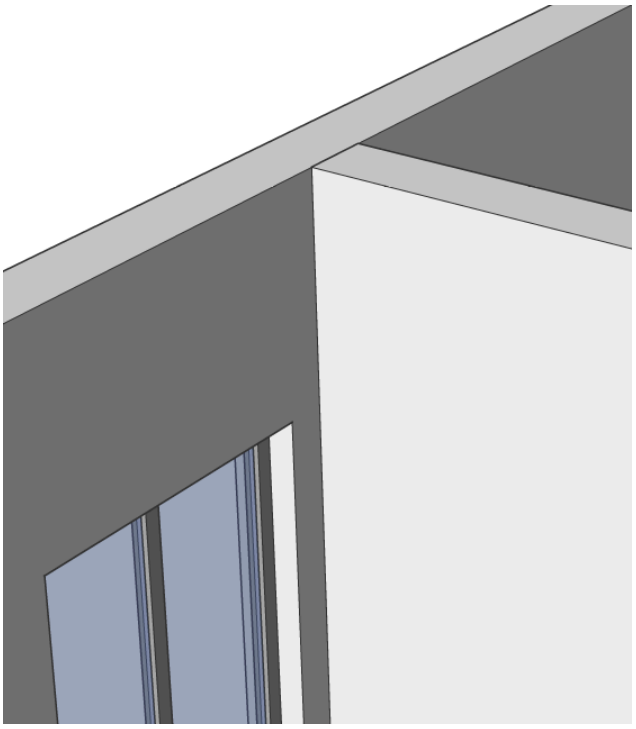
20.3.8.9 Procedure: creating T- connections between two walls

1 In the drawing area, highlight the face you want to connect to another wall.



2 Use the **Connect with Nearest** tool from the Quad.

The **Connect with Nearest** tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



20.3.8.10 About the select aligned faces tool

The **Connect with Nearest** tool can also be used to connect multiple faces of a wall to the nearest face of another element. To do this use the **Select aligned faces** tool. This tool enables you to highlight all faces that are coplanar with the current selection.

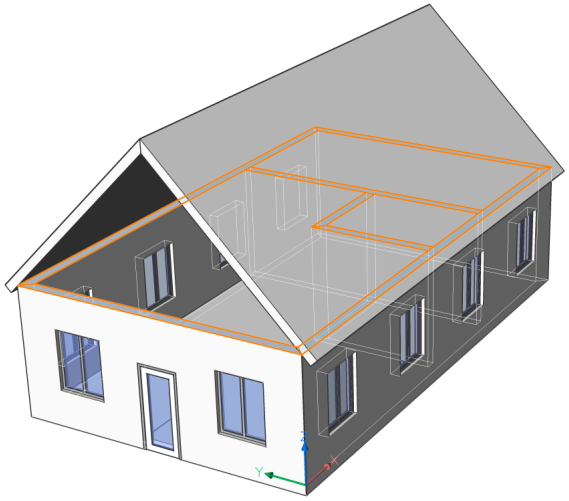


For more information about this command, visit the Command Reference article SELECTALIGNEDFACES.

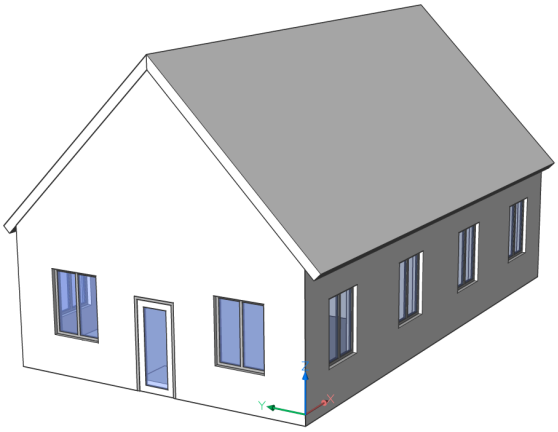
20.3.8.11 Procedure: connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click

the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



20.3.9 Curtain wall

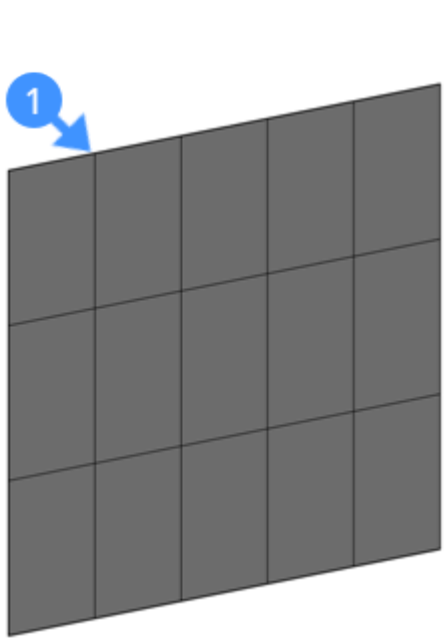
20.3.9.1 Commands

BIMCURTAINWALL

20.3.9.2 About

In BricsCAD, curtain walls are created from the face of a 3D solid or surface as a block by using the **Curtain Wall** tool.

- Curtain walls can be created from any planar and curved surface.
- Window Frames are placed on a BIM Grid when creating a Curtain Wall.
- Changing the U-line and V-line panel options allows you to specify the desired size and number of panels on the planar or curved surface.



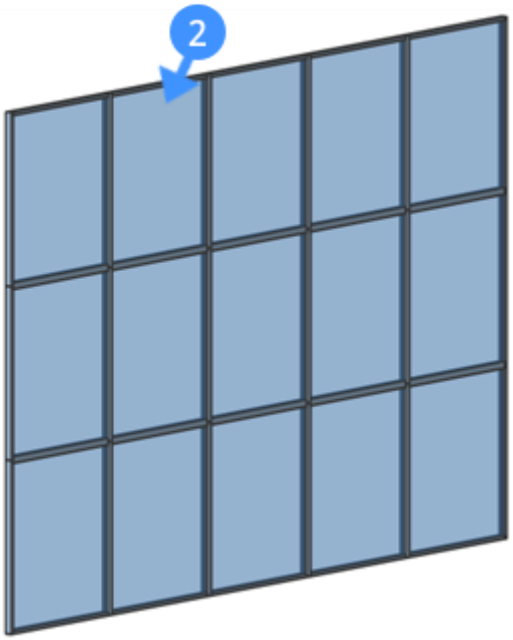


Figure 1, Curtainwall grid (1).

Figure 2, Curtainwall with glazing (2).

The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall.

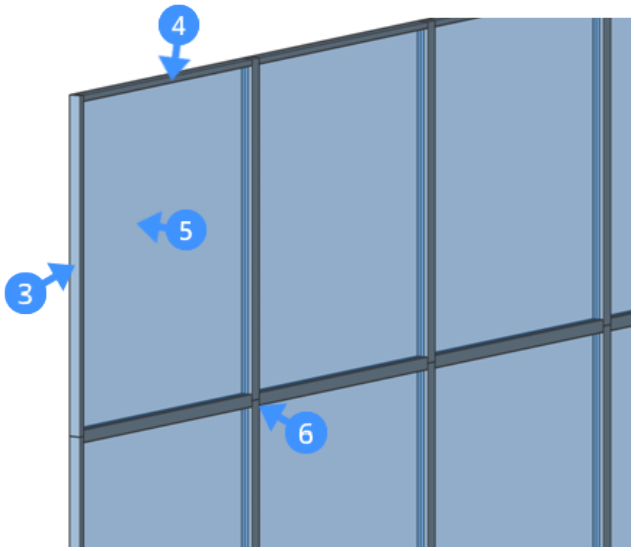
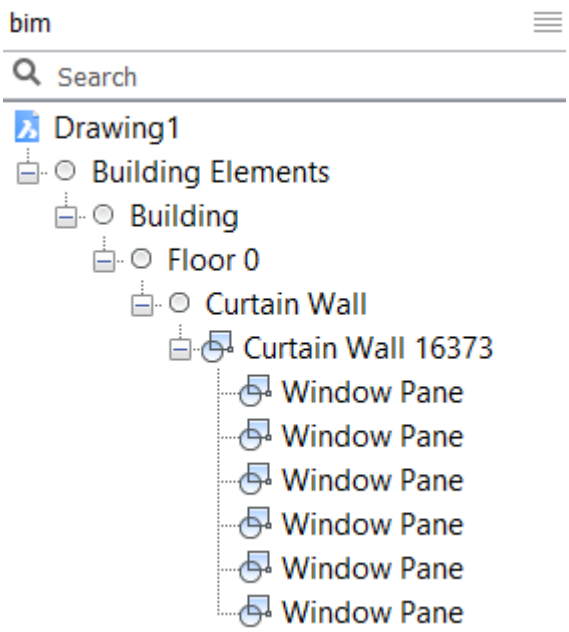


Figure 3, A curtain wall. The frame that coincides with the grid vertical (U) panels (3), the frame that coincides with the grid horizontal (V) panels (4), and glass in-fill (5) are shown. By default, the straight connection (6) is made between the beams.

Curtain walls are automatically classified as a **Curtain wall** in the **Structure browser**.

If you explode the Curtain wall, the sub-entities are classified as windows and as frames. The frames are linear solids. This allows you to calculate the length of a curtain wall.



20.3.9.3 Procedure: creating a curtain wall

1 Launch the BIMCURTAINWALL command in the Command line.

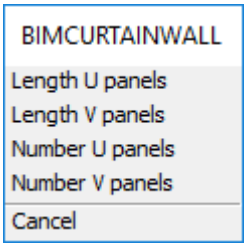
1 You are prompted: Select a face:

2 Select a face in the drawing area to create a curtain wall.

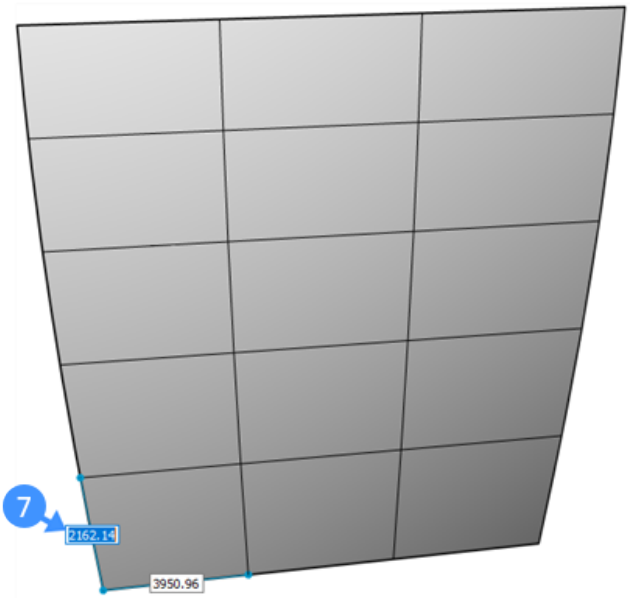
3 The curtain wall grid is automatically generated on this face.

You are prompted: Create grid or [Lenght U panels/Length V panels/Number U panels/Number V panels]:

4 Change the length of the U or V panels and number of U or V panels using the prompt menu and the Command line.



Or, if Dynamic dimension (DYN) is on, the length of the panels can be changed using the DYN fields (7).



5 Hit the TAB key to switch between dynamic dimension boxes.

6 Or Press Enter to accept the current value.

Nota: 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 (8) will appear and the Command line prompts a message (9).



The prompt menu and the Command line allow 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".

7 Press Enter to accept the default value.

8 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

The prompt menu:



(**Optional**) To change the frame width, select the **Width** option in the prompt menu or type "W" in the Command line.

Default 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.

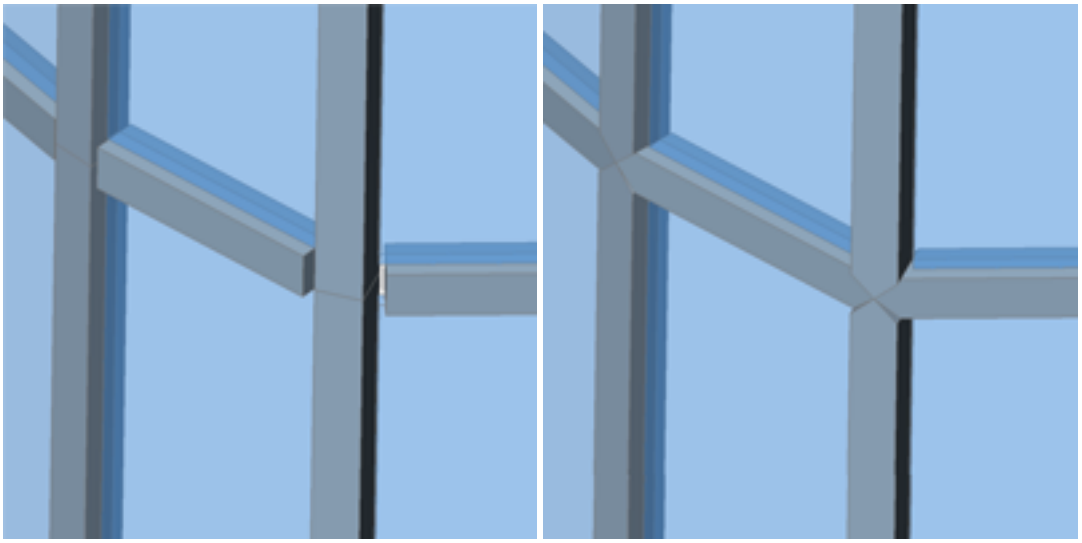
Default frame depth = 60.000000

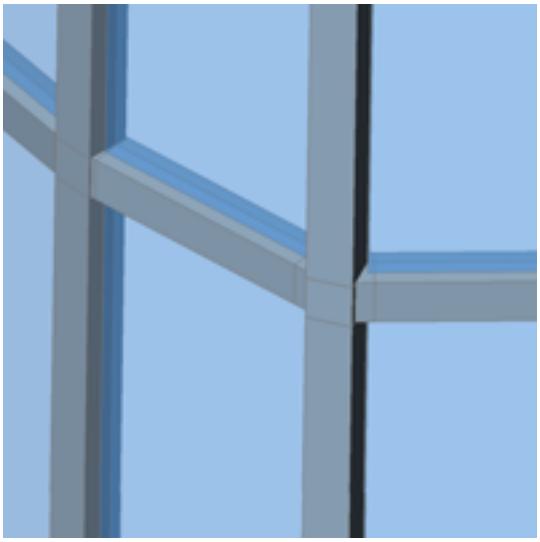
(**Optional**) To change the glass thickness, select the **Glass thickness** option in the prompt menu or type "G" in the Command line.

Default glass thickness = 20.000000

(**Optional**) To choose a connection type between, select the **Connections type** option in the prompt menu or type "C" in the Command line. For smooth connections, select the **Smooth connection** option or type "S" in the Command line. For nodes connection, select the **Nodes connection** option or type "N" in the Command line.

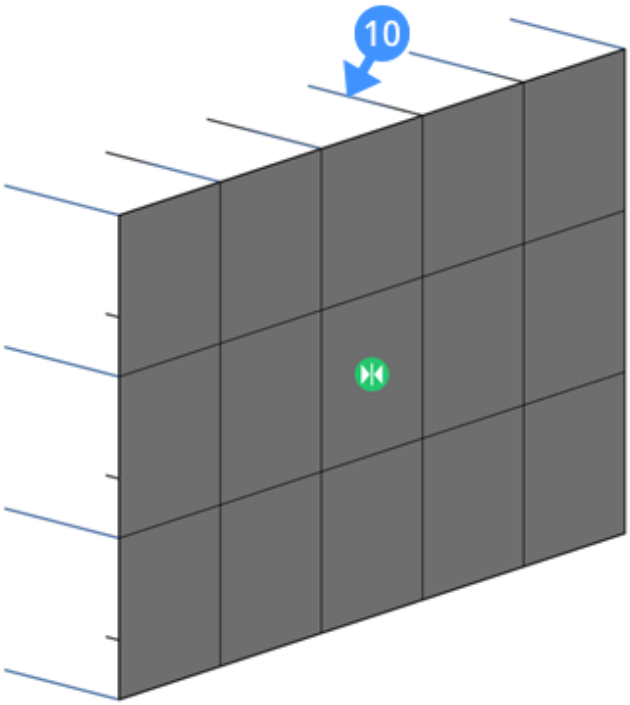
The default connection is straight.





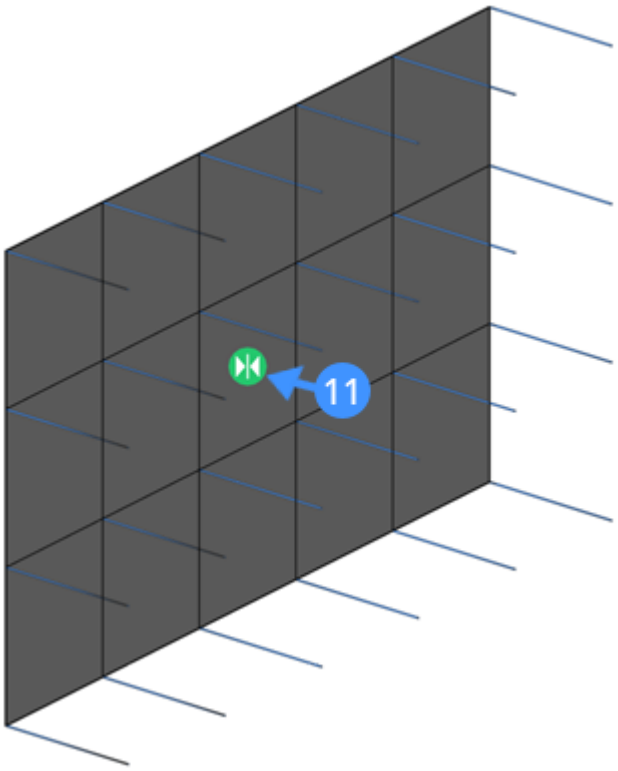
There are three types of connection. From left to right: Straight - Smooth - Nodes.

9 Blue lines (10) appear on the face to show the side a curtain wall will be created.



Nota: Step 6 and step 7 are skipped when the face is selected from a solid. In this case the inside normal is chosen by default.

10 Use the widget (11) to flip the curtain wall.

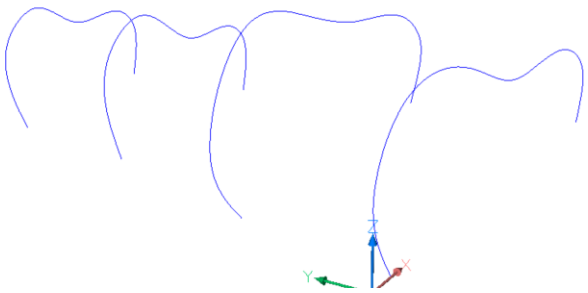


11 Press Enter to accept.

Nota: The surface or solid used to create the curtain wall will be deleted by default. If you don't want this, set the **DELOBJ** value to '0' in **Settings**.

20.3.9.4 Procedure: Creating a curtain wall from a free form surface

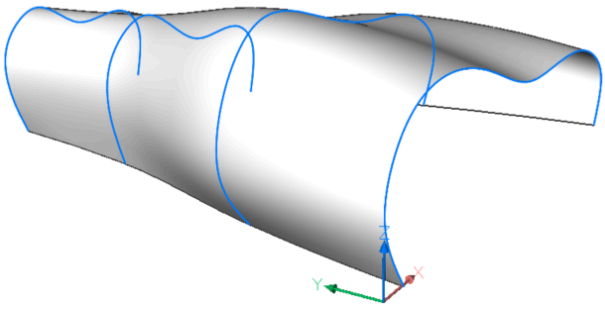
1 Start with 4 splines.



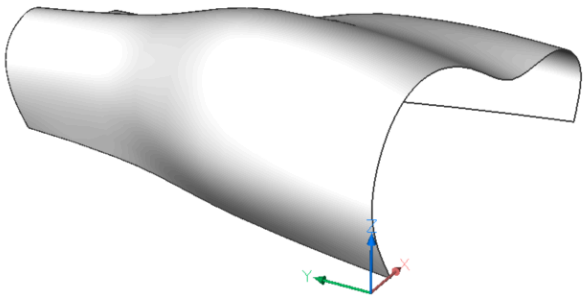
2 Launch the LOFT command in the Command line.

3 You are prompted: Select cross-sections in lofting order [MOde]:

Select the 4 splines in lofting order.



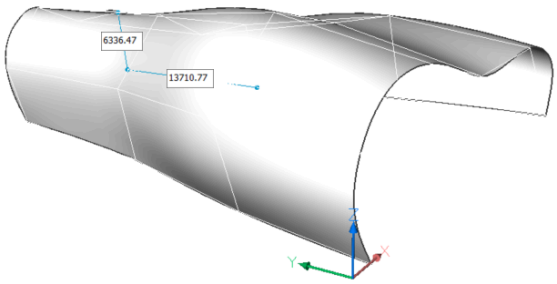
4 Press Enter twice to end the command.



5 Launch the BIMCURTAINWALL command in the Command line.

6 You are prompted: Select a face

Select the lofted surface.



7 A curtain wall grid is automatically generated from this surface.

8 You are prompted: Create grid or [Lenght U panels/ /Length V panels/Number U panels/Number V panels].

Change the number of the U and V panels in the prompt menu or the Command line. In this example: U = 20 and V = 8.

9 You are prompted: Planarize [Yes/No] <Yes>

10 Click "Yes" to planarize the grid cells.

11 You are prompted: Enter target deviation <0.00001>:

12 By default, the target deviation from planarity is set to "0.00001".

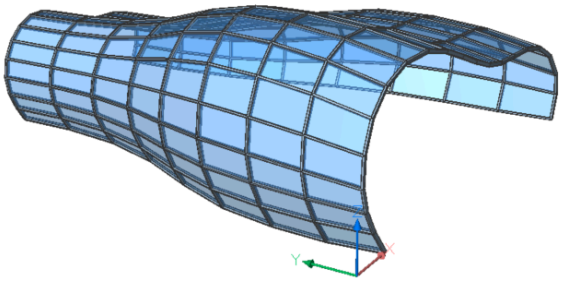
Press Enter to accept it.

13 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

14 Change the parameters to:

- Width: 150
- Depth: 300
- Glass thickness: 20
- Connection type: Smooth

15 Press Enter to accept. A curtain wall is created.



20.3.10 DmExtrude

20.3.10.1 Commands

DMEXTRUDE

20.3.10.2 About

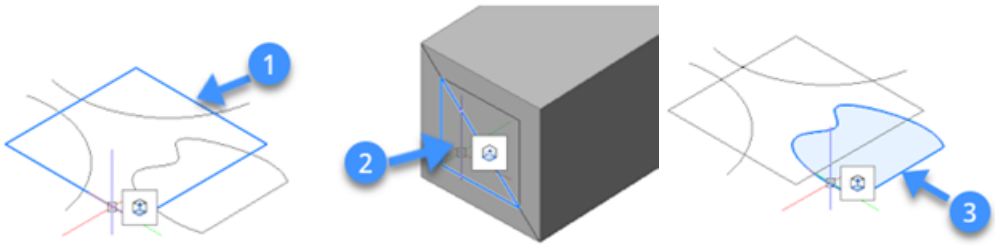
The **Extrude** tool allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The **Extrude** tool enables you to cut the solid model partially by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model.



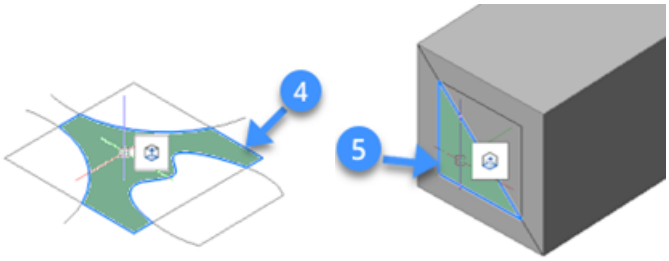
For more information about this command, visit the Command Reference article DMEXTRUDE.

The following illustration shows the selected entities to be extruded.

Closed entities (1) (2) are shown. A region object (3) is shown in the figure.



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.



The closed boundaries are recognized automatically in BricsCAD, when the **Enable Boundary Detection** option on the **Selection modes** toolbar is active.

Nota: As a closed entity only closed polylines, circles, and ellipses are accepted.

20.3.10.3 Procedure: using DMEXTRUDE in solid mode

The following are general steps to add and remove volume from the solid(s) using the **Extrude** tool.

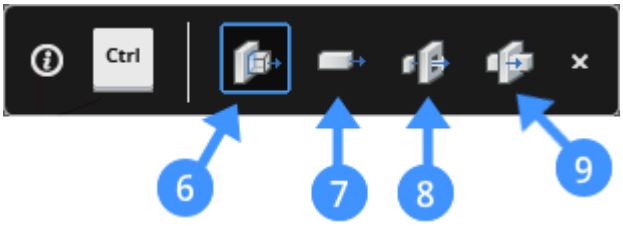
1 Hover the cursor over an entity or sub-entity in the drawing area.

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>.

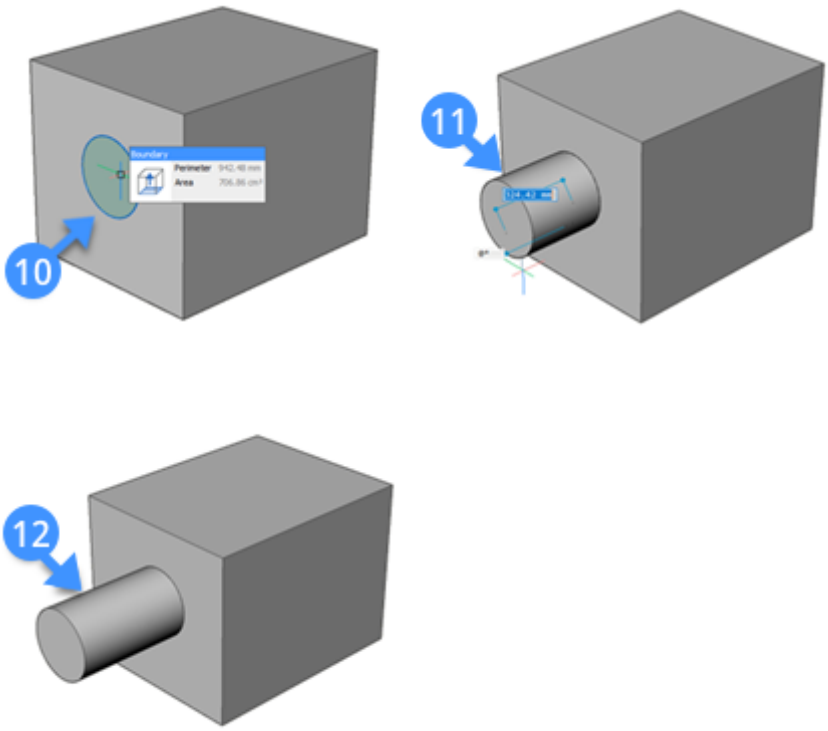
3 The “DmExtrude Hotkey Assistant” appears at the bottom of the screen with a set of extrusion options.

Nota: The default mode is indicated by a blue frame.

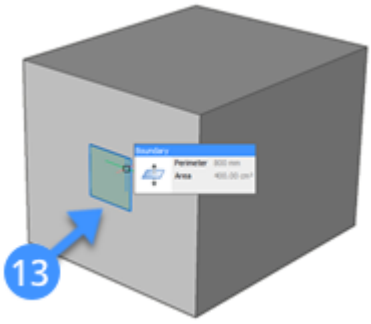


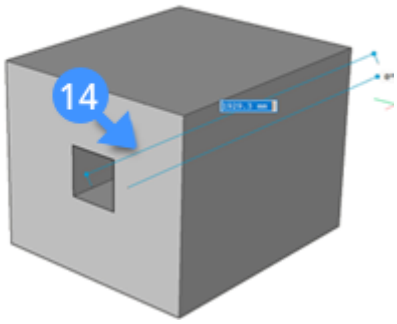
- **Auto (6)**: The result of the auto option depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable value. Note: By default, the value of the DMEXTRUDEMODE is set to “3”. The following illustration shows the extruded entity from the main solid while the default value is set to “3”. When you highlight an entity (10) and extrude it

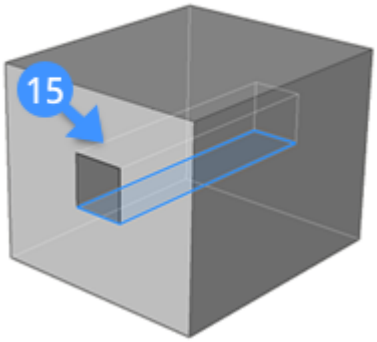
to the outward (11), a new volume is added (12) to the main solid model.



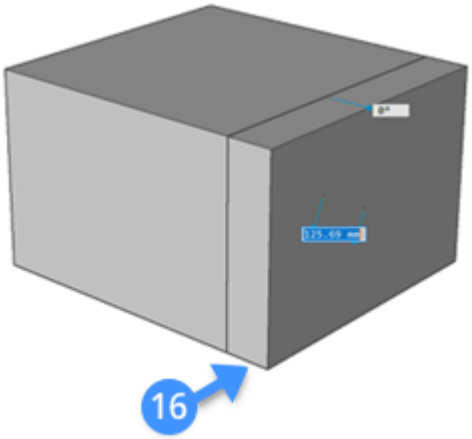
The following illustration shows the subtracted entity from the main solid. When you highlight an entity (13) and extrude it to the inward (14), a volume is subtracted (15) from the main solid.





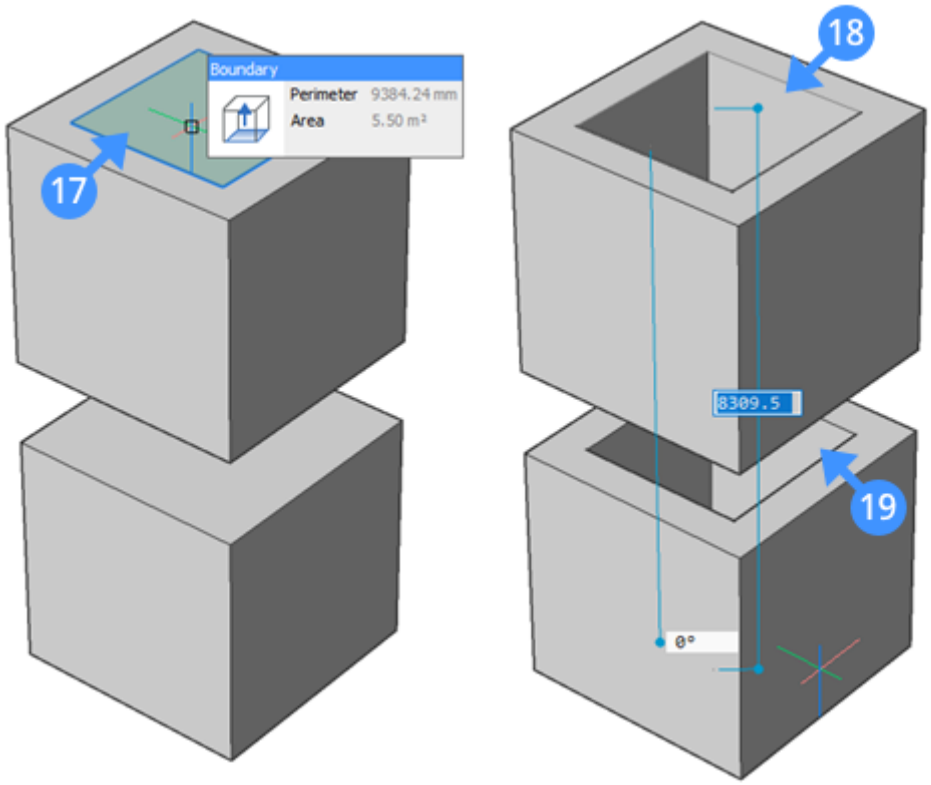


- **Create (7)**: Regardless of the extrusion direction, a new volume is created (16) from the existing solid.



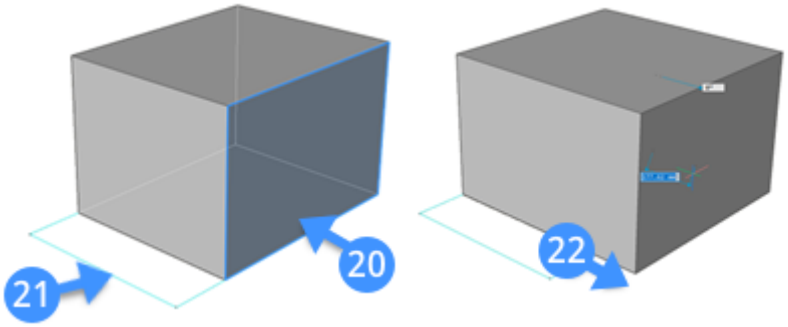
- **Subtract (8):** The 3D solid is subtracted from each interfering existing solid.

  The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17).

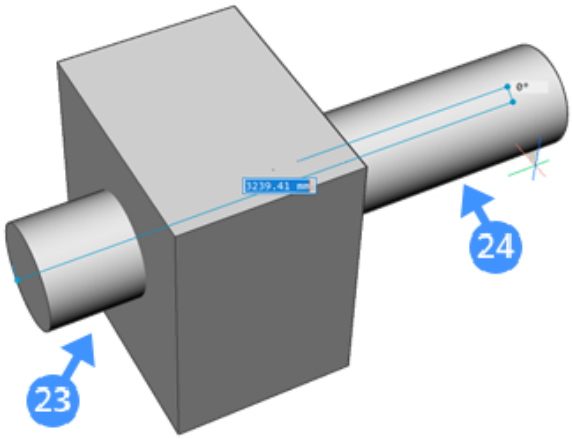


- **Unite (9):** The new volume is unified with each interfering existing solid.

  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.



- **Both sides**: New symmetric volumes are created (23), (24) in both directions from the main solid.



To create symmetric volumes from the existing solid, type “B” on the 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.

20.3.10.4 DMEXTRUDE in Surface Mode

The **extrude** tool allows you to create 3D surface(s) by extruding 2D entities such as lines or polylines.



For more information about this command, visit the Command Reference article DMEXTRUDE.

20.3.10.5 Procedure: using the extrude tool in surface mode

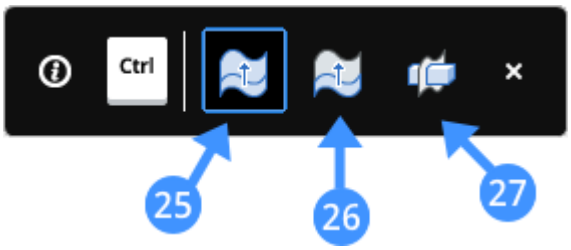
The following are general steps to create 3D surface(s) using the **Extrude** tool in the Surface mode.

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.



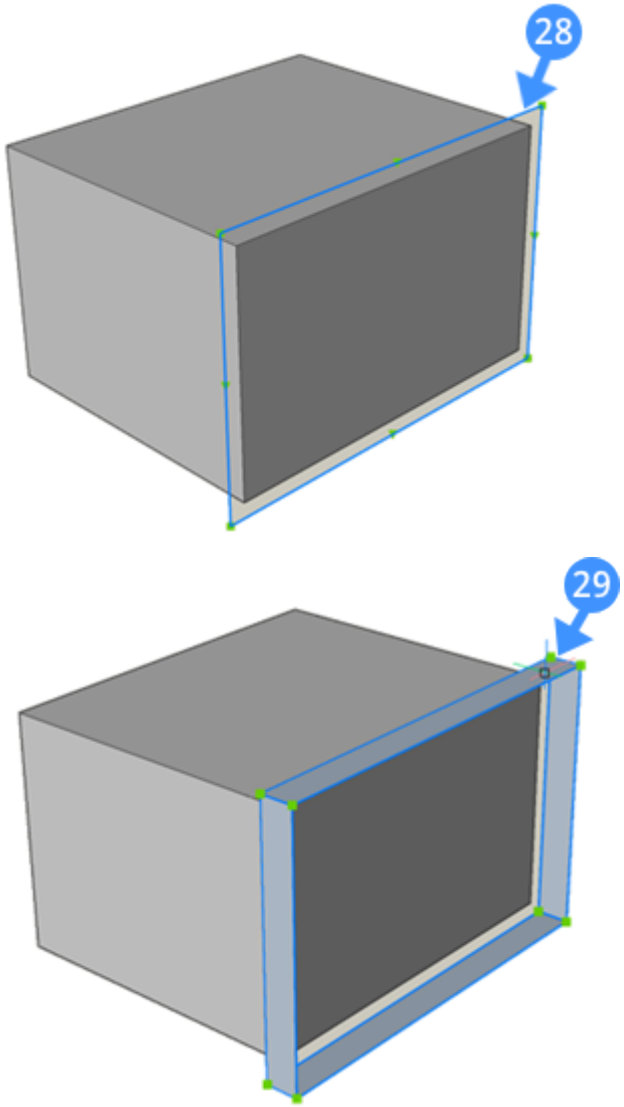
Nota: The default mode is indicated by a blue frame.

- **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 with the base surface.
- **Create (26):**Create new surfaces.
- **Slice (27):**Cut through solids with the extruded surface. This option is much like the SLICE command.

4 Select one of the options and move the mouse to give the extrusion direction, entering a value in the dynamic dimension field.

5 Press Enter or right-click to accept the extrusion for the surface.

6 The following illustrations show the created surface (29) by extruding the selected 2D entity (28) in the drawing.



20.3.11 DmPushPull

20.3.11.1 Commands

DmPushPull

20.3.11.2 About

The **Push/pull** tool allows you to add and remove volume from solids by moving highlighted faces.

Nota: **Push/pull** tool applies to planar, cylindrical, spherical, conical and toroidal faces of a solid.



For more information about this command, visit the Command Reference article DMPUSHPULL.

20.3.11.3 Procedure: using Push/Pull tool to add or remove volume to the solid

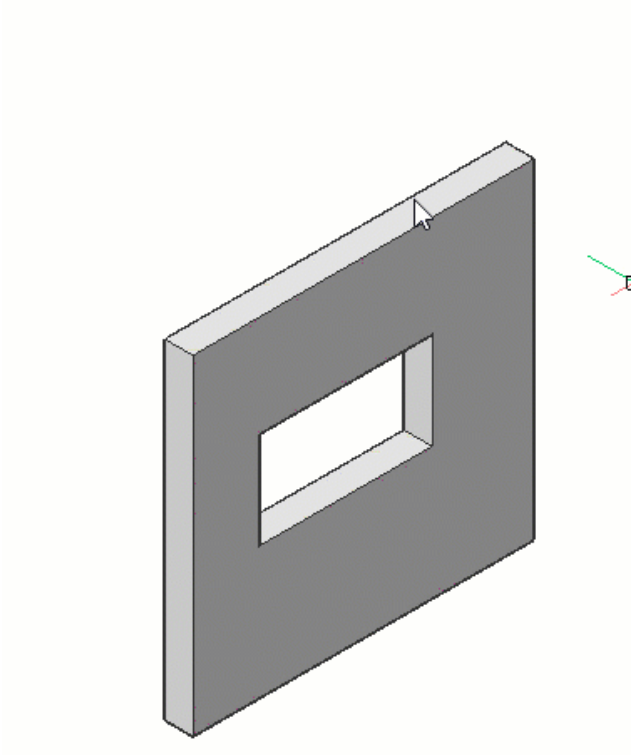
The following are general steps to add or remove volume from a solid(s) using the **Push/Pull** tool.

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 to pull the highlighted face.

4 The following movie shows the highlighted face behavior. The selected face moves dynamically, adding or removing volume.



5 Specify a point in the drawing or enter a value in the dynamic dimension field, then right click to accept changes.

20.3.12 Editing grids

20.3.12.1 Commands

BIMGRID, BEDIT, NUMBER, PROPERTIES

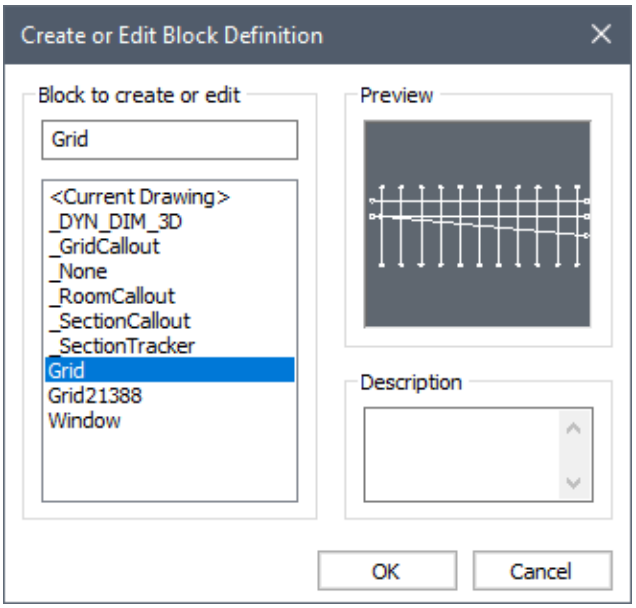
20.3.12.2 About

BIM grids are created as a block. The BEDIT (Block Edit) command allows to edit a BIM grids.

20.3.12.3 Editing a rectangular grid

1 Double click the grid.

1 The **Create or Edit Block Definition** dialog box displays.



2 Select the grid, then double click or click the OK button.

3 A **Block Edit** session is started.

4 Edit the grid axes.

5 You can move, rotate, copy, stretch, grip-edit or delete grid axes.

6 Renumber grid axes or edit axis labels if necessary (see the procedures below).

7 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

20.3.12.4 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.

2 Select all vertical axes.

3 Start the NUMBER command and choose the appropriate number style.

4 Make sure to overwrite the existing numbers.

5 Repeat the previous steps for the horizontal axes.

6 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

20.3.12.5 Editing a grid label

1 Double click the grid to start a **Block Edit** session.

2 Double click an axis.

3 The properties of the axis entity (line or arc) display in the **Properties** panel.

4 Select the **BIM/Axis number** property.

5 Type a value or name.

6 Click the **Bclose and Save** icon () on the **Block Edit** toolbar.

20.3.13 Inserting and editing windows and doors

20.3.13.1 Commands

BIMINSERT

20.3.13.2 About

In BricsCAD, the **Insert BIM Component** tool allows you to insert windows, doors and other components, such as furniture, sanitary equipment, plants, etc. in any face of a 3D solid and is typically used to insert windows and doors.



For more information about this command, visit the Command Reference article BIMINSERT.

20.3.13.3 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**: 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 the DYN field in the status bar and **Dynamic UCS** is active by checking the DUCS field in the status bar.

  The **Dynamic UCS** aligns 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**: Click the mouse wheel or press the **Temporary Tracking Points** tool button () on the **Entity Snaps** toolbar 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 the Ignore entity snap elevation tool button () on the **Entity Snaps** toolbar. The state of the tool button (pressed or not) indicates the current value of **OSNAPZ**.

- **Entity snap to negative Z-values**: Press the **Entity snap to negative-z** tool button ( ) on the **Entity Snaps** toolbar to enable entity snaps on points that lie behind the selected face. If **OSNAPZ** is ON, the point is projected onto the selected face.

20.3.13.4 Procedure: inserting windows or doors

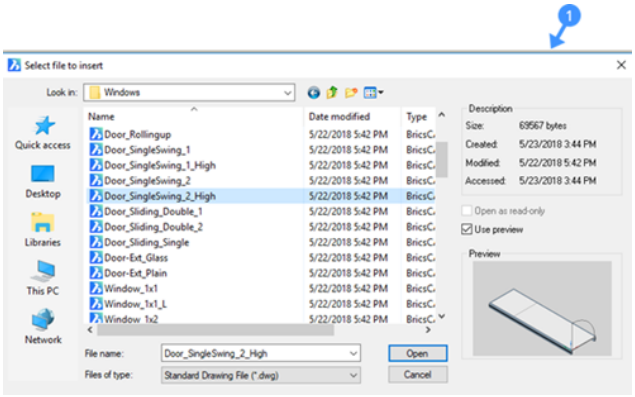
Nota: Make sure **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

Using the BIMINSERT command

1 Select **Insert BIM Component**from the **Model** tab of the No Selection Quad.

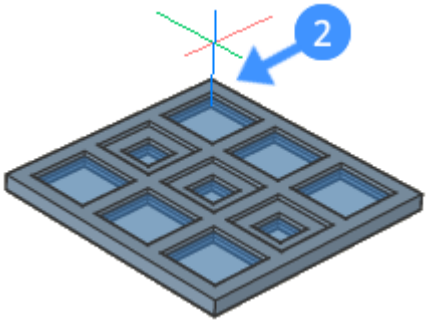
The file explorer opens, displaying the content of the most recently used folder to insert a BIM component. The default folders for BIM components sit in the following subfolder of the BricsCAD installation folder: ...\UserDataCache\Support\en_US\Bim\Components.

2 Select the **Windows** (1) or **Doors** folder.



3 Select a window or door you want to insert in your drawing, then double click or click the **Open** button.

The selected insert is attached to the cursor (2).



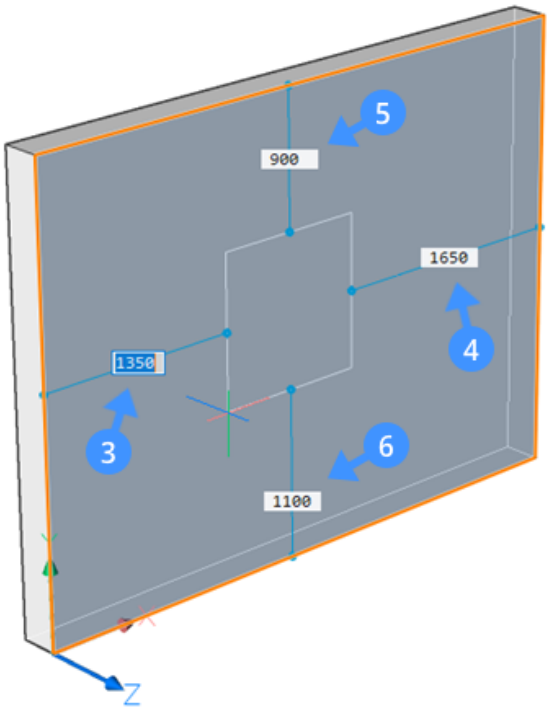
The parameters of the insert display in the **Properties** panel.

Prompts you: Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

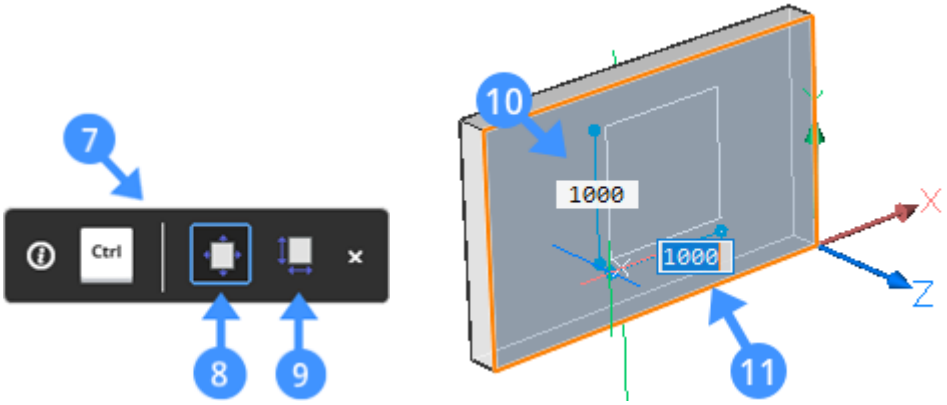
4 Optionally, edit the parameters in the **Properties** panel or choose an option (see Command Options of the BIMINSERT command).

5 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.



If the HKA (Hot Key Assistant)* is ON, the HKA 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 'change parameters' 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.



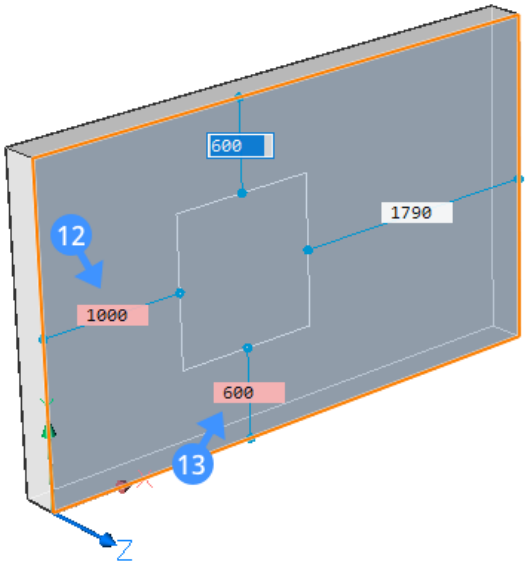
* Click the HKA field in the Status bar to toggle the display of the Hot Key Assistant.

6 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).



Using the components panel

See: Insert a window from the Components Panel into your drawing.

20.3.13.5 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.

Parameters	
Name	Window_3x2:1
File	
W	1000 mm
H	1000 mm
Rebate	0 mm
PlacementDepth	90 mm
FrameThickness	60 mm
FrameDepth	60 mm
PaneDepth	20 mm
PaneOffset	20 mm
MullionThickness	60 mm
TransomThicknes	60 mm

2 Select a parameter you want to change, then type a value in the settings field.

20.3.13.6 Moving an insert

In BricsCAD, there multiple methods to move an insert in the face of a solid:

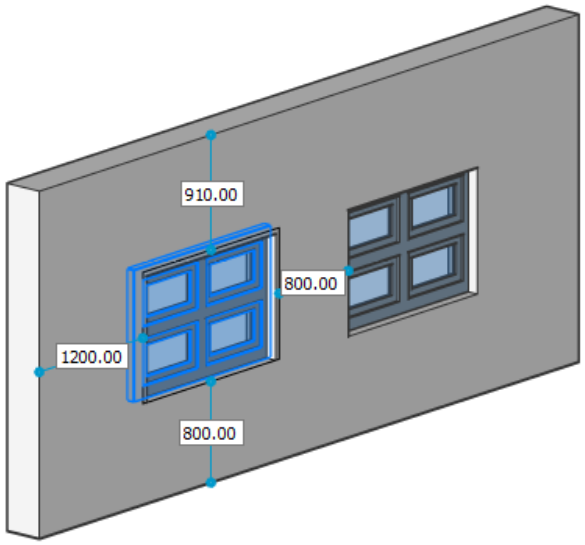
- Using Distance fields.

- Using **Push/Pull**
- Using the Manipulator

Procedure: using distance fields

1 Select the insert in the model.

1 Current distances from the insert to the nearest parallel faces of the parent solid display:

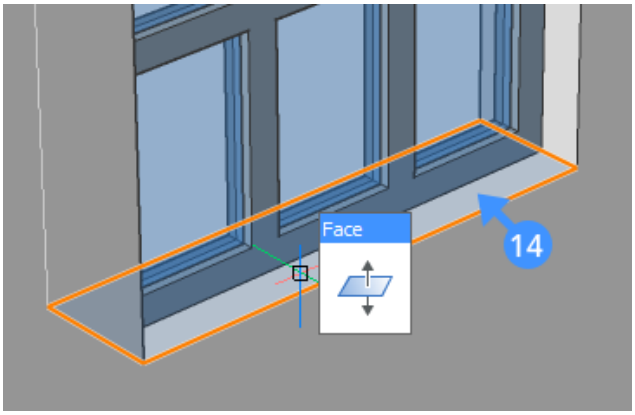


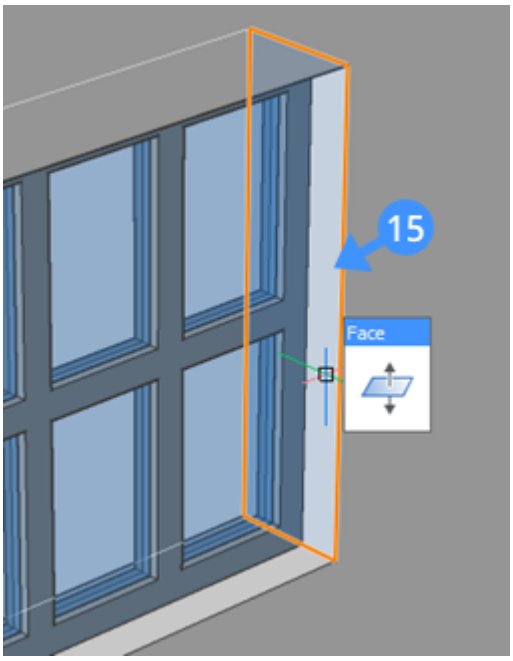
2 Double click a distance field, then type a value and press Enter.

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.





2 When the face highlights, select **DmPushPull** from the **Model** tab of 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.

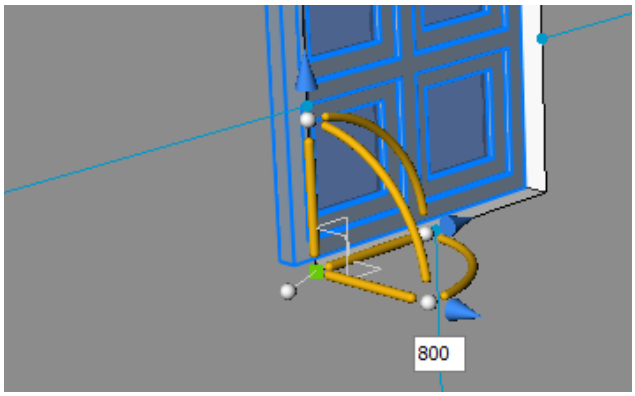
3 Do one of the following to complete the moving insert process.

- Specify a point.
- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

Procedure: using the Manipulator

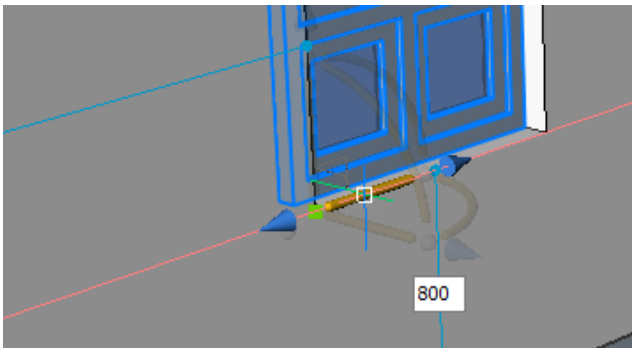
Select the insert by holding the left mouse button down a bit longer. Set the MANIPULATOR system variable to 2.

The Manipulator displays.



1 Hover over the vertical or horizontal axis trigger zone of the Manipulator.

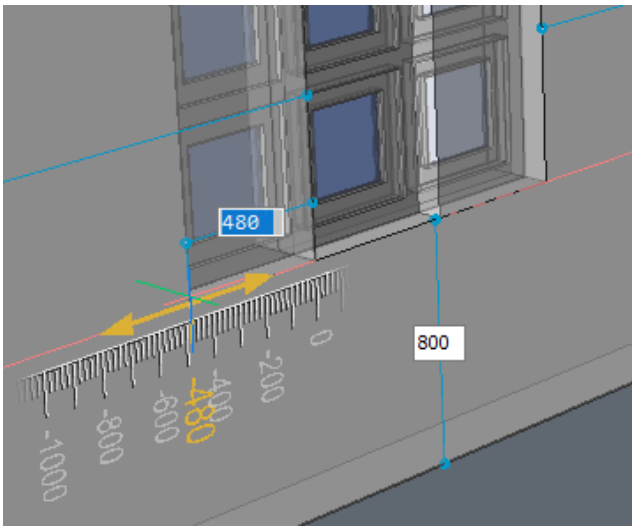
1 The axis highlights and the rest of the Manipulator fades.



2 Click to start the movement operation.

3 The window moves dynamically or if you hold down the Ctrl key creates a copy of the selected insert.

The Ruler displays.



4 Do one of the following:

- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

20.3.13.7 Copying Inserts

Inserting a copy of an insert

1 Hover over the insert. When the insert highlights, choose **Insert BIM Component** in the **Model** command group in the Quad.

2 You are prompted: Select insertion point or [Edit inserted entity/Rotate component/set Base point/ Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

3 Optionally choose the **Multiple** option to insert multiple copies.

4 Proceed with the Inserting Windows and Doors procedure 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.

Nota: 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.

20.3.13.8 Procedure: copying an insert on the wall face

Nota: Make sure that **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

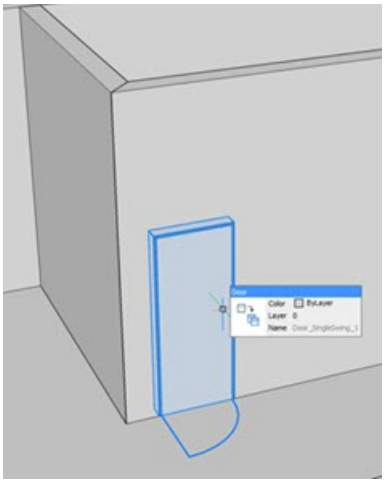
1 Select an insert using the **BIM Insert** tool.

You are prompted: Select insertion point:

2 Place the insert on the wall face.

3 The insert should align with the wall face.

4 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.



You are prompted: Enter base point [Displacement/ mode] <Displacement>:

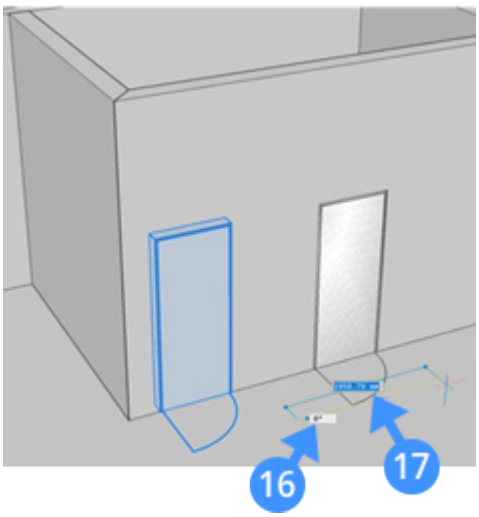
5 Pick a point in the drawing area.

6 You are prompted: Enter second point <Use base point as displacement>.

7 Type a displacement distance from the coordinates of the base point.

Nota: Displacement specifies the distance to position the copy.

You can use the dynamic dimension fields (16,17) to specify the displacement.



You are prompted: Enter second point [Undo/Repeat/Exit].

8 Optionally, use the **Repeat** option to create multiple copies of the insert keeping the displacement distance of each copy same.

9 Press Enter to accept the copying.

20.3.13.9 Replacing an insert

Use the BMREPLACE command to replace an insert.

20.3.14 Mass modeling with quickbuilding

20.3.14.1 Commands

BIMQUICKBUILDING, BIMSPATIALLOCATIONS

20.3.14.2 About

The **QuickBuilding** tool allows you to generate a BIM building just from one or more massing solids. BricsCAD creates a separate drawing and in this drawing walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created

It's useful in early design stages, volume studies and conceptual designs.

The tool will assign BIM properties and spatial information to each part of the new building.

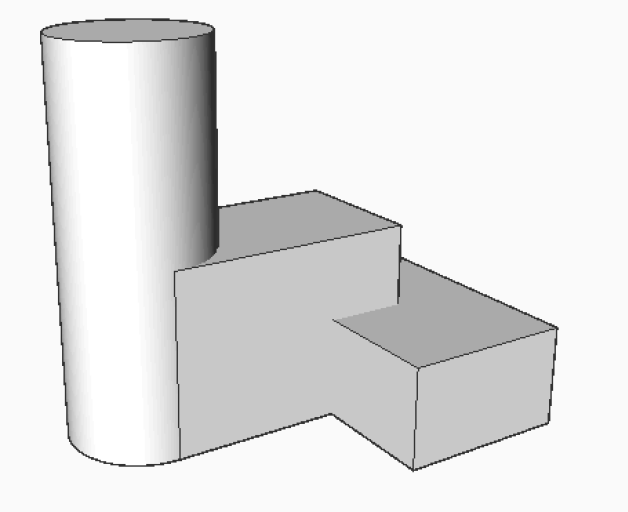


For more information about this command, please visit the Command Reference article BIMQUICKBUILDING.

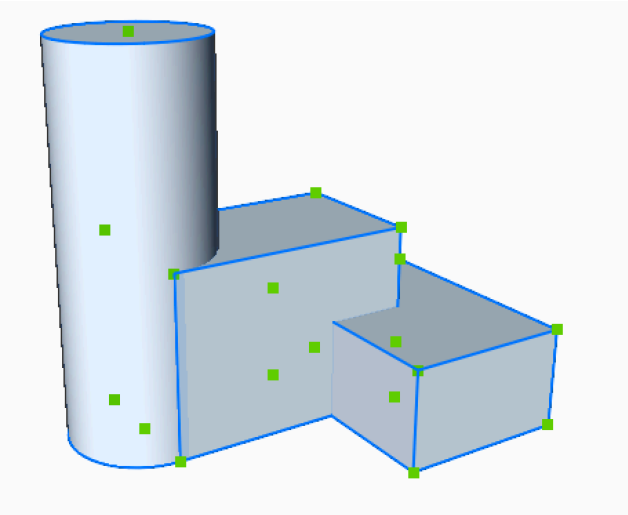
20.3.14.3 Procedure: top-down workflow

1 Draw one or more solids in your model space.

Nota: The solids don't need to be unioned, the command will do it automatically.



2 Type the command BIMQUICKBUILDING in the Command line or select from the Quad, select the solids and press Enter.

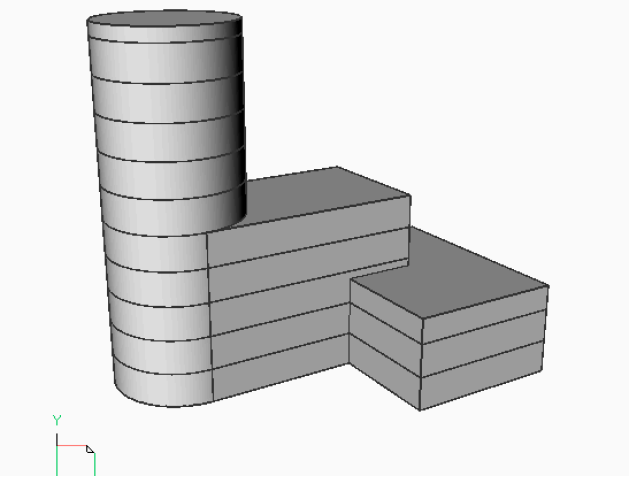


3 You are prompted: Do the selected solids represent [Buildings/Spaces]: Type B to continue with the top-down workflow and press Enter.

4 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatialallocations]:

5 Enter the desired story height.

Press Enter and the building will be generated.



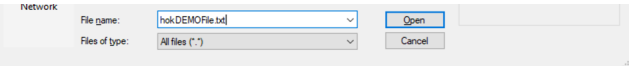
Nota: To generate different floor heights, type (2x4000,*3000,1x5000) which stands for two stories of 4000mm, the rest 3000mm and the top floor has a story height of 5000mm.

Or import the story height values from a .txt file.

Enter BIMQUICKBUILDING in the Command line,

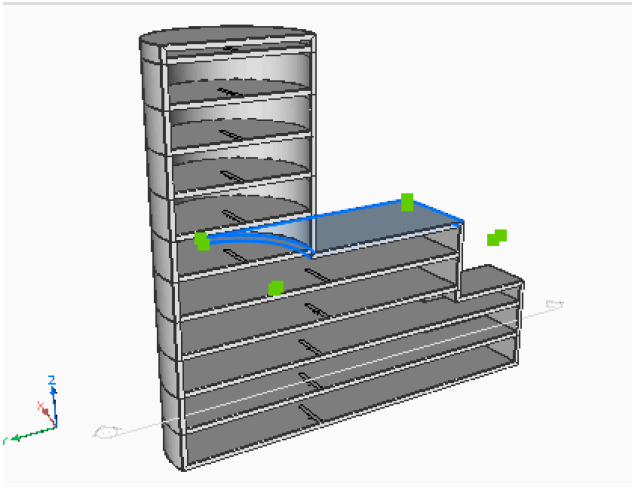
Type BUILDINGREQUIREMENTFILE and press Enter.

Then search for the text file in the file folder and click open.



Now the properties of the text file are applied to the building.
Example of a text file with parameters for your building:
Space:Name='Bricsys Tower',Area=15000
Space:Name='Level 0 - Reception',Area=6%,'Floor-to-Floor Height'=5000
Space:Name='Level 1-10',Area=5%,'Floor-to-Floor Height'=4000,Count=10
Space:Name='Level 11-17',Area=4%,'Floor-to-Floor Height'=3000,Count=7
Space:Name='Level 18-19',Area=4%,'Floor-to-Floor Height'=4000,Count=2
Space:Name='Level 20-21',Area=1.5%,'Floor-to-Floor Height'=4000,Count=2
The area and area ratios that are specified in the file are requirements. If the result of **Quickbuilding** does not satisfy the required areas this will be printed in the Command line, see also the section about Command Line Output later in this article.

6 If you make a section, you can see the newly formed slabs and walls which each have individual BIM properties.

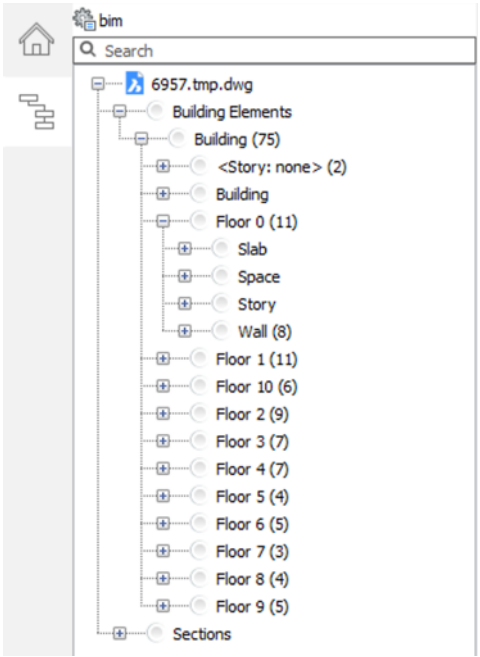


Select the roof and navigate to the **Properties** Panel - **BIM** tab. You will see that this entity has **Roof type** property, as well as some other properties such as spatial locations, quantities or a **GUID**.

BIM	
Type	Roof
Building Element	Roof
Name	
Description	
Building	Building
Story	Floor 5
Composition	
GUID	15iQzJUHr8rxP2hWT$pWj7
Roof type	Not defined
Space bounding	On
Number	
Entity property s	

Nota: All the elements are categorized, in the **Structure Browser** you will see a list of building elements, sorted by floors and building element.

The different buildings elements will be placed in different layers according to their building type (External wall/slab, Flat/pitched roof, Internal wall/slab).

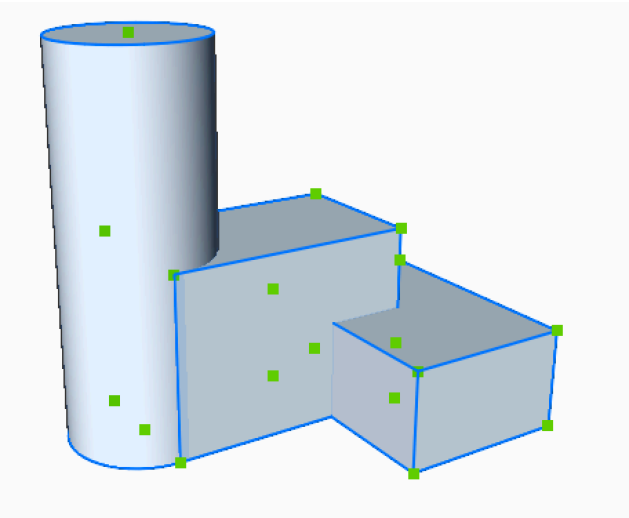


20.3.14.4 Procedure: bottom-up workflow

The bottom-up workflow creates a building out of solids that represent spaces. This approach allows you to first shape the spaces, then use **QuickBuilding** to create the shell and the new spaces.

1 Type QuickBuilding in the Command line.

2 Select the solids and press Enter.



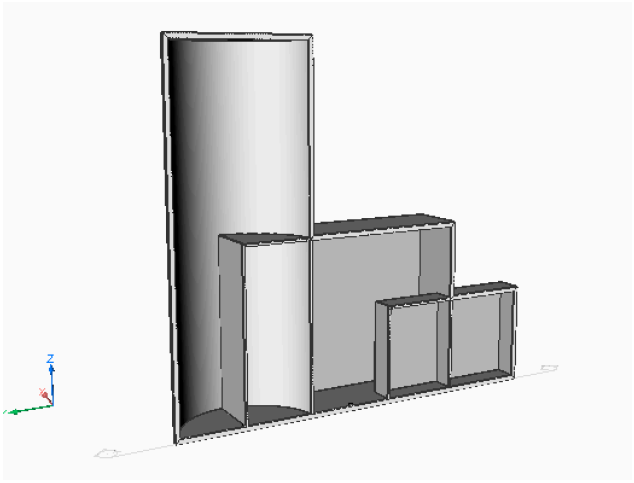
3 You are prompted: Do the selected solids represent [Buildings/Spaces]:

4 Type Spaces to create a building from the selected massing solids.

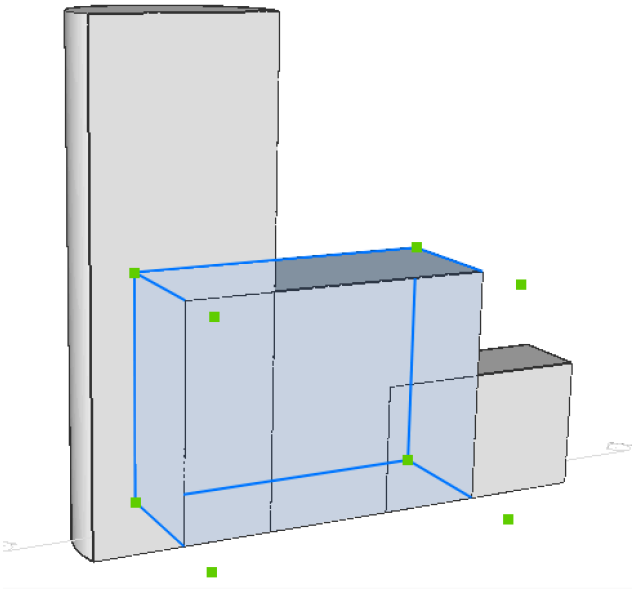
Nota: BricsCAD takes these defined solids and converts them to BIM Spaces with walls and slabs.

5 Press Enter to create the building.

6 (Optional) make a section to see the newly created spaces.



Nota: The original solids are kept, **Quickbuilding** just creates a Building from it with the information that these input solids represent spaces, so it knows that inner walls and slabs should be created based on those input spaces.

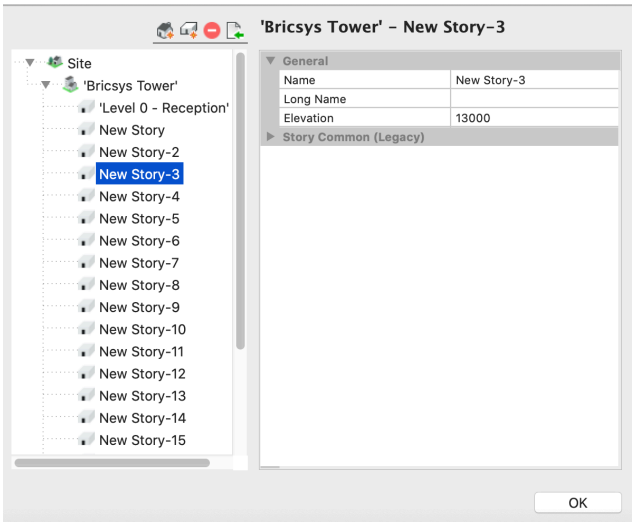


20.3.14.5 Procedure: SPATIALLOCATIONS as input

It is also possible to use a Spatial Structure as input for **Quickbuilding**. So, you can import a Spatial Hierarchy and use that data to subdivide the building.

1 Import or Edit the SPATIALLOCATIONS.

1 For more information about this command, please visit the help center article SPATIALLOCATIONS.



2 Make sure that the solid(s) you wish to use for **Quickbuilding** are:
- classified as a Building (under **Spatial Structure Elements** in **BimClassify**), and
- are assigned the correct 'building' in its BIM properties.

3 Type QUICKBUILDING in the Command line.

4 Select the building and press Enter.

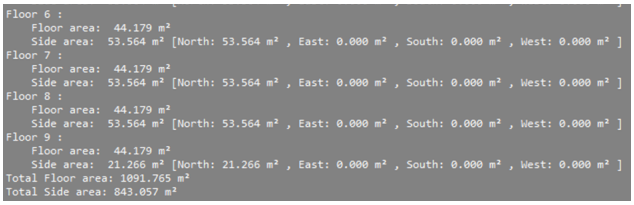
5 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatiallocations]:

6 Type SPATIALLOCATIONS to use the Spatial Structure as input.

Nota: If you add more than one building in the Spatial Locations manager and then assign the building to the solids, **QuickBuilding** will use the different story heights on the different buildings.

20.3.14.6 Command line output

After you run the BIMQUICKBUILDING command, BricsCAD will create an overview with information on some basic building information. For instance, the total floor area of the building or other information for each floor.



Nota: If you used a Building Requirement File, and some requirements are not passed (e.g. not enough total area or floor area), you can see the problems in the output.

```
Total Floor area: 330.990 m² Requirement NOT passed, area should be >= 15000.000 m² m²
Total Side area: 1001.571 m²
Floor to Side ratio: 33.047 %
```

20.3.15 MEP modeling

20.3.15.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMFLOWCONNECT, DRAG, BIMSTRETCH

20.3.15.2 About BIMFLOWCONNECT

It's also possible to connect different segments using the BIMFLOWCONNECT command.

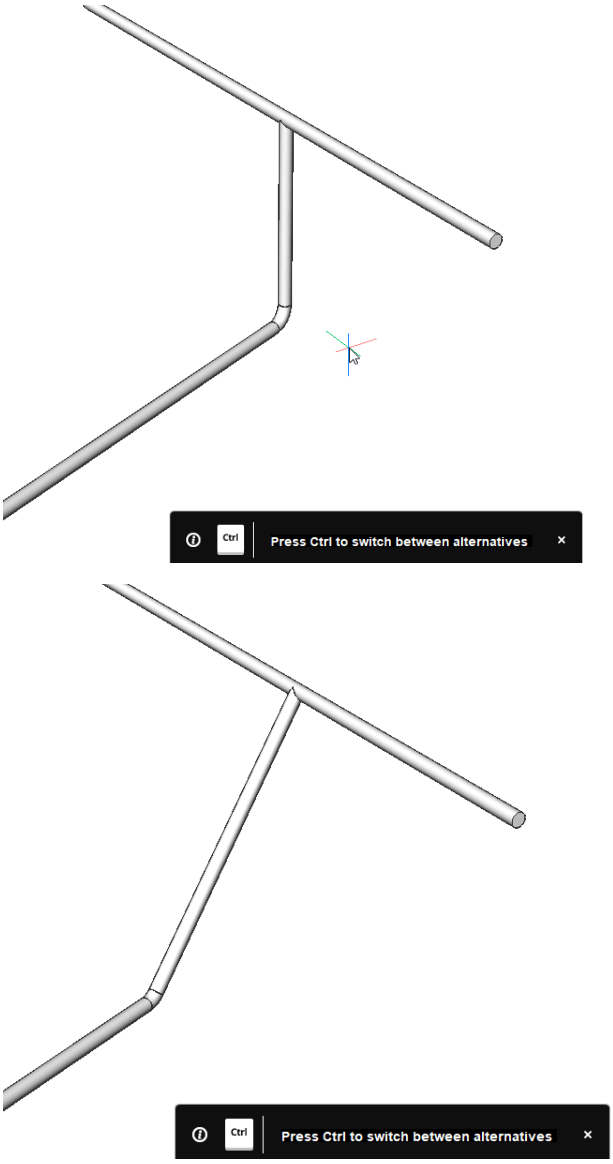


For more information about this command, visit the Command Reference article BIMFLOWCONNECT.

Connecting different segments to each other

Select the segments you want to connect and launch the BIMFLOWCONNECT command.

When the Hot Key Assistant is switched on (**HKA** on the status bar), the following widget appears on the screen press the CTRL-key to cycle through the different options.



End the command by pressing Enter, the segments are now connected.

20.3.15.3 Procedure: altering the position of segments

Before starting with the procedures: enable the selection of sides and ends:



Click the icon on the **Selection Modes** toolbar or set the DISPLAYSIDESANDENDS system variable ON.

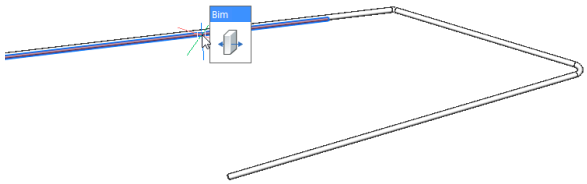
DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing its connection to the other segments.



For more information about this command, visit the Command Reference article DRAG.

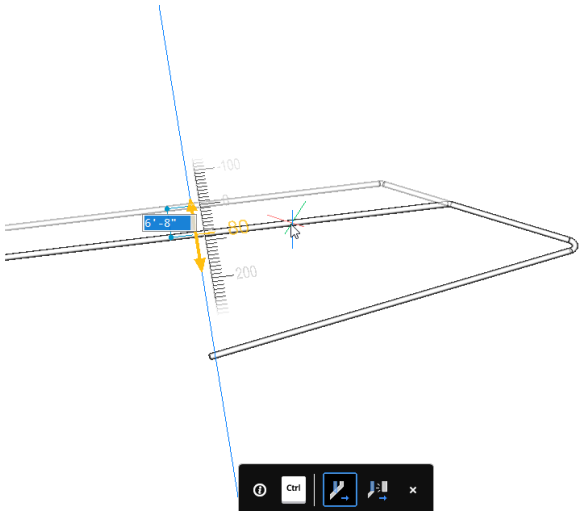
1 Hover over a segment, select a face perpendicular to the section of the profile.



2 Launch the DRAG command.

3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the displacement.

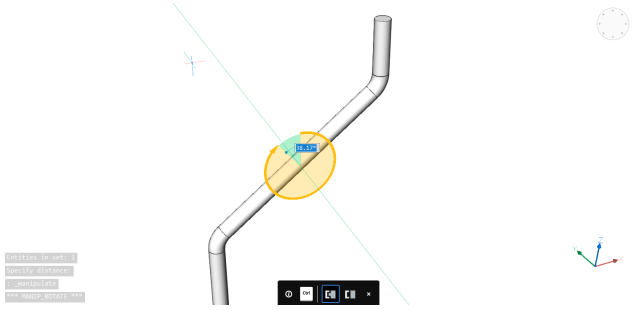
Nota: The software automatically regenerates the connection between the segments.



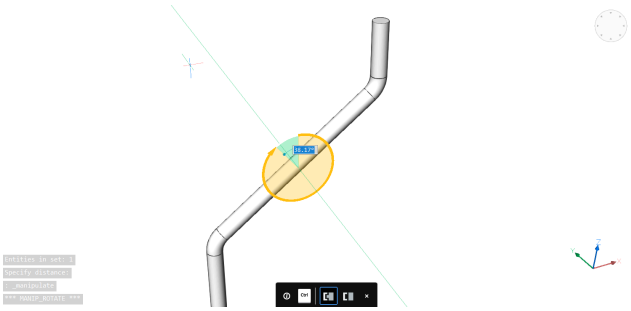
Manipulator

1 Hover over a segment, select a face perpendicular to the section of the profile.

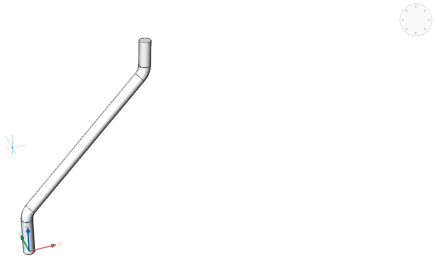
2 Select **Manipulate**.



3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the rotation.

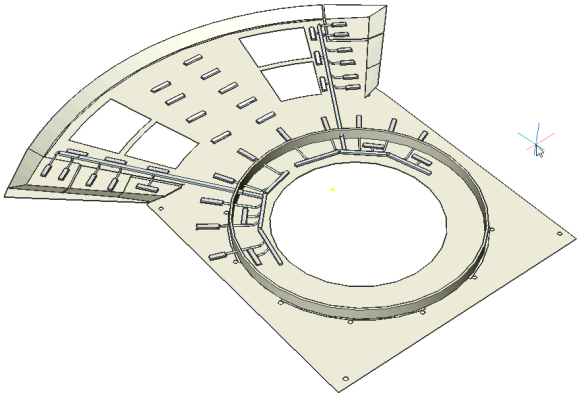


4 Click on the new position or enter the rotation angle and hit enter.



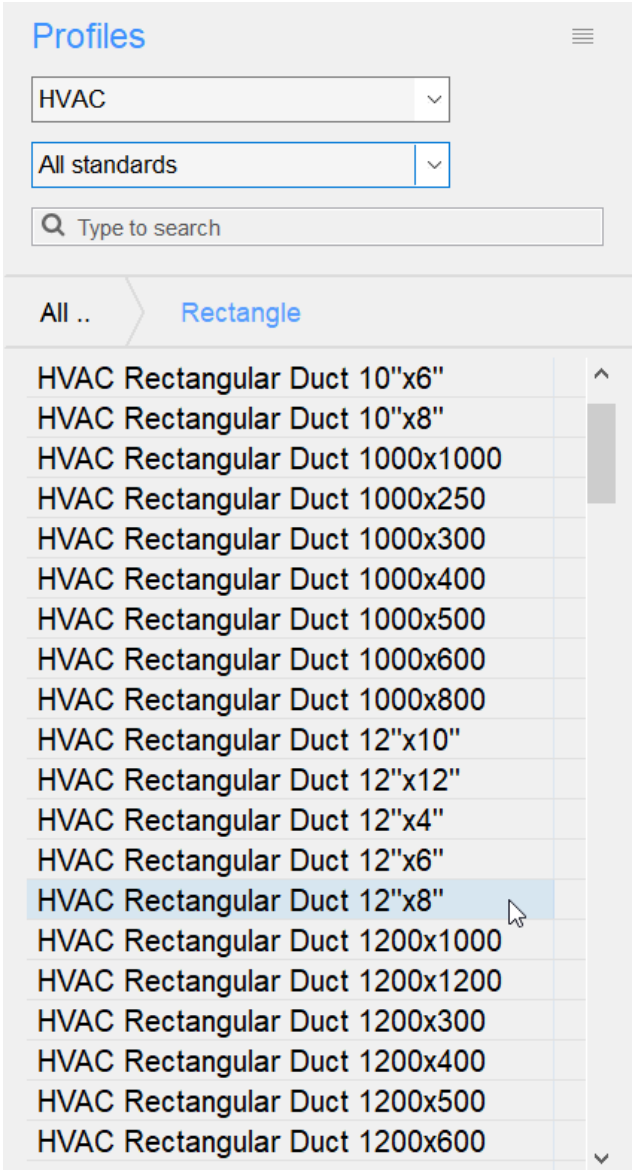
20.3.15.4 Procedure: using the MEP modeling tools to draw and connect profiles

1 Open the drawing that contains the MEP model. You can download this **MEP.zip**-file containing the drawings at the end of this article.



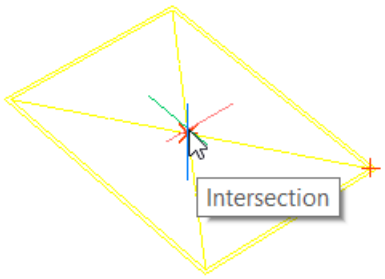
2 Open the BIM **Profiles** panel on the right-hand side of your screen (see Dockable Panels).

Filter on **HVAC** and select the Rectangular 12"x 8" profile, drag it to the model space.



3 Define the starting point of the profile.

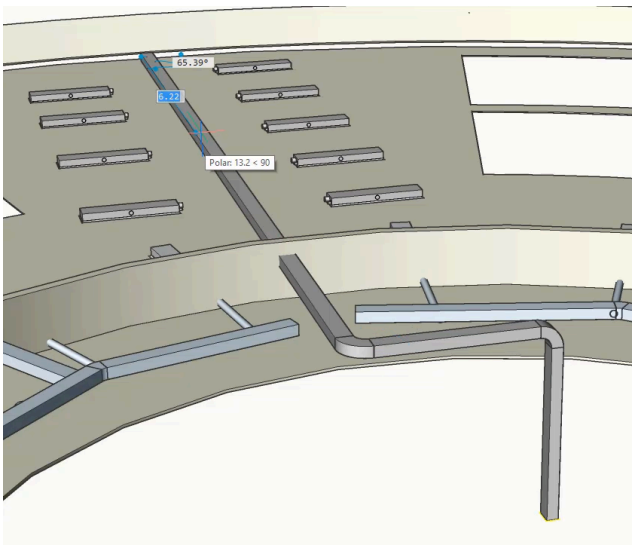
4 Snap to the geometric center of the yellow rectangle.



5 Turn **Ortho** on and define the next points of the profiles by using the dynamic dimension field.

Quarter turns the profile by entering Q in the Command line.

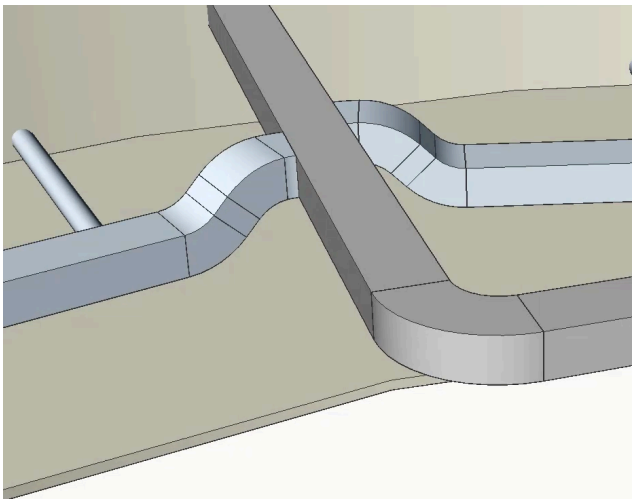
Go up 3.5, go 3.5 to the left and go 21 horizontally.



Nota: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as **Flow Segment**. You can manually classify the profiles through the BIMCLASSIFY command as **Building Service Elements** - **Flow Segments**.

6 Connect the other rectangular ducts on the side to the one we have just created.

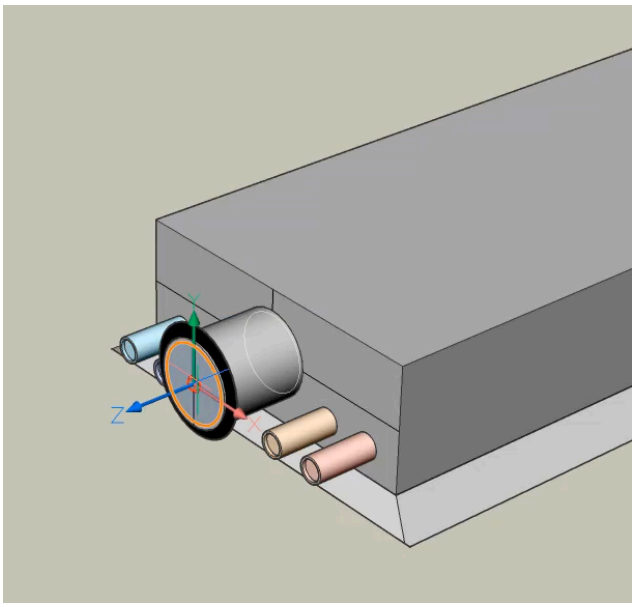
7 Select the ducts together with the main duct and launch the BIMFLOWCONNECT command. Your connection should look like the image below.



8 Create a duct from the flow terminal.

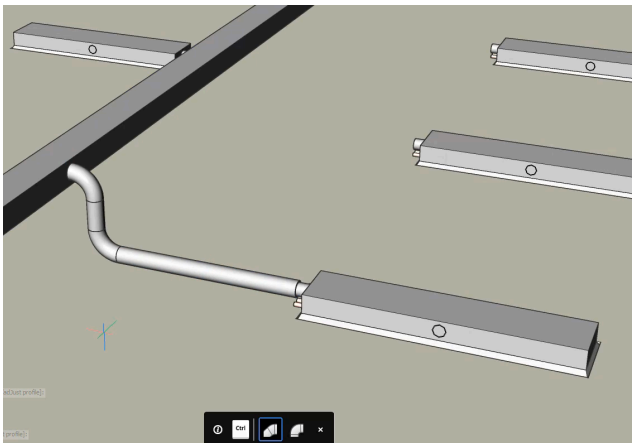
9 Launch the BIMLINEARSOLID command and select the flow connection point.

This connection point contains profile information, so the software automatically creates the predefined circular duct. Give the duct a length of 1.



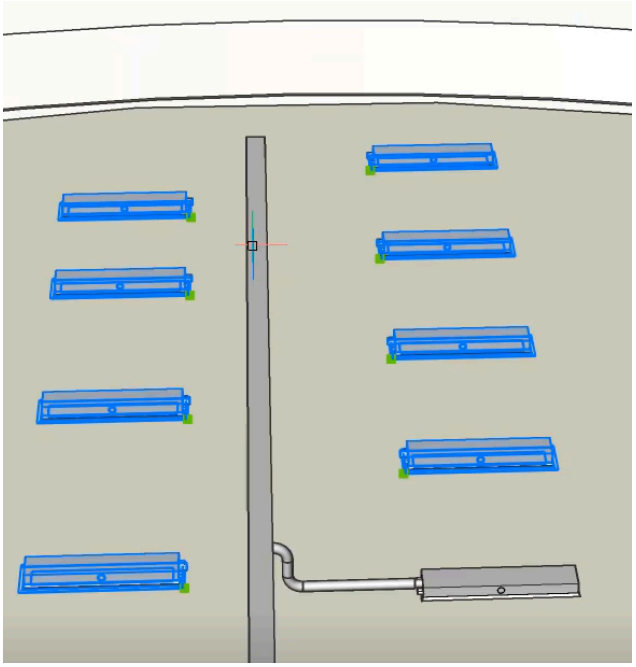
10 Connect the ducts from the flow terminal to the rectangular main duct.

11 Select the main duct and the duct from the flow terminal and launch the BIMFLOWCONNECT command.



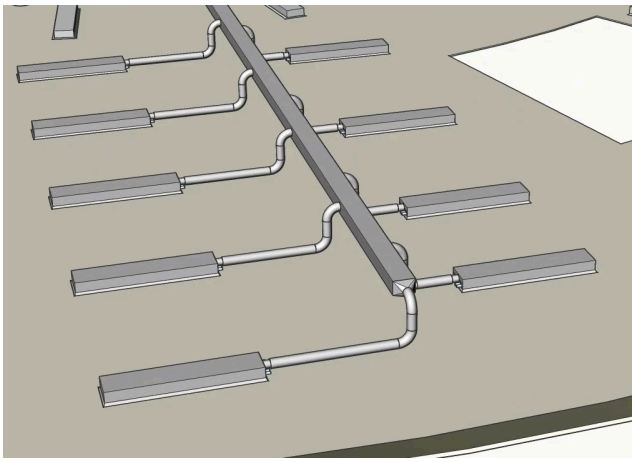
12 Connect all the other flow terminals to the rectangular duct at once.

13 Select the other flow terminals and the rectangular duct and launch the BIMFLOWCONNECT command.



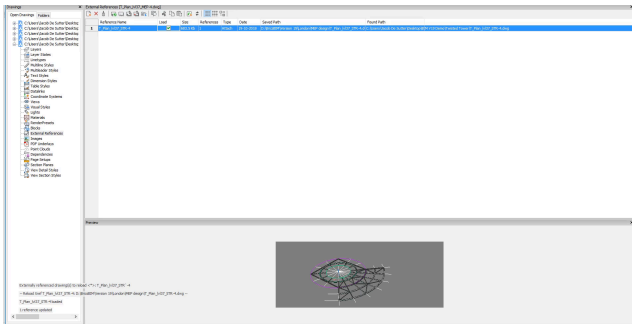
14 BricsCAD uses the information from the flow connection point to determine the type of duct used to connect with the main duct.

Nota: The flow terminal on the back is connected to the back of the rectangular duct.



20.3.15.5 Procedure: Using the MEP Modeling tools to alter profiles

1 Type XRef in the Command line and check the load box next to the structural model to load the structural drawing.



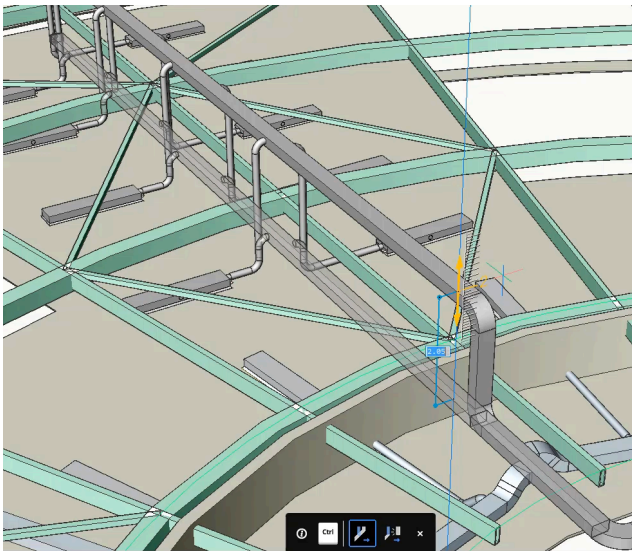
2 Use SLICE command and **Multislice** option to divide the rectangular duct.

Enter 3 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.

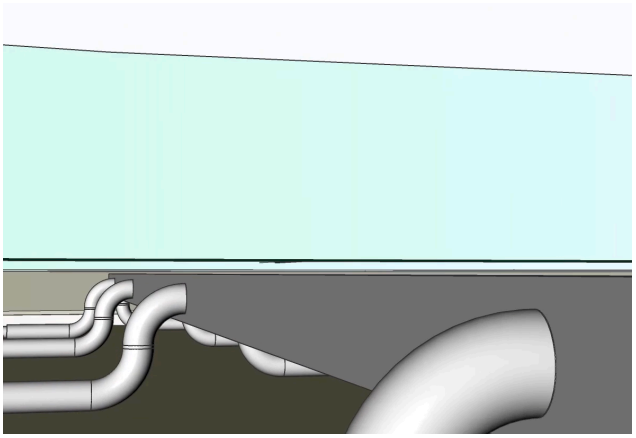
4 Select the upper face of the duct and launch the DRAG command.

Nota: BricsCAD preserves the connections with the circular ducts. You can change this by pressing the CTRL-key if you want to disconnect the ducts.



Move the duct down by 0.23.

5 The structural model no longer interferes with the MEP Model, while the connections between the ducts of the flow terminals and the main rectangular duct were preserved.



20.3.15.6 Procedure: using Bimstretch to modify flow segments

1 Open a New drawing.

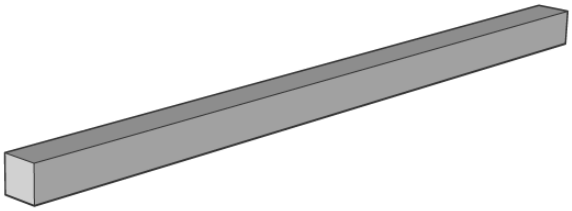
2 Open the BIM **Profiles** panel by clicking on it on the right-hand side of your screen.

Nota: If the BIM Profile panel icon isn't visible yet, right-click on the ribbon and check **Profiles** in the context menu under.

3 Filter on the **HVAC** standard. These profiles will be automatically classified as flow segments once they are dragged into the model space.

4 Search for the profile **HVAC Rectangular Duct 10"x10"** in the **Profiles** panel and drag the profile in the model space.

5 Select a starting point and draw a profile. Press Enter to stop drawing with this profile. Your model should look like this:

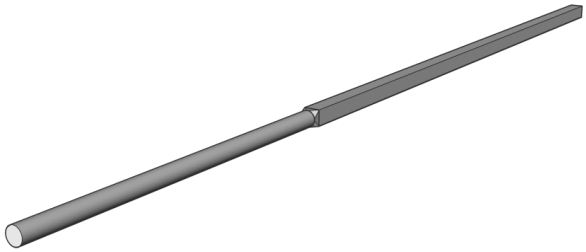


6 Search for the profile **HVAC Round Duct 8"** in the **Profiles** Panel and drag the profile in the model space.

7 Click on the end of the Rectangular profile, after hovering on its profile face.

Nota: To easily snap on the end of the rectangular profile, turn on the **Display Axes** setting or set the DISPLAYAXES system variable to 1.

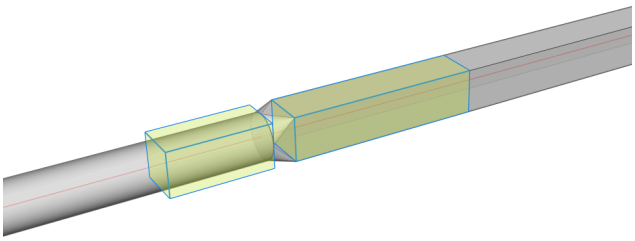
8 Make sure to draw the circular profile along the same axis as the rectangular one. After accepting the last point of the circular profile, it will be connected to the rectangular profile through an automatically generated reducer.



Nota: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as Flow Segment and the connection will not be made immediately. You can still manually classify the profiles through the BIMCLASSIFY command and then use BIMFLOWCONNECT to connect the two through a reducer.

Nota: Make sure the **Display Sides and Ends** and **Display Axes** is enabled.

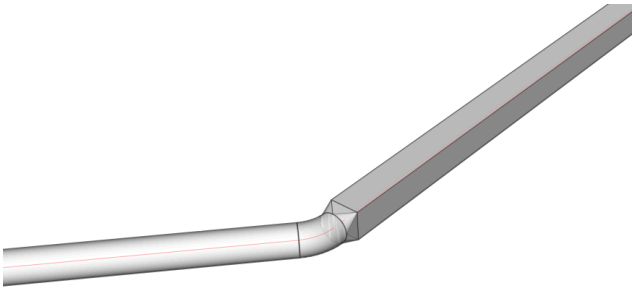
9 Select the connecting ends of both segments.



10 Launch the **BimStretch** tool.

11 Hover your cursor to one of the sides and click on the desired location. BricsCAD automatically stretches the 2 Flow Segments in their new position and adds an extra curved Flow Fitting in the

middle.



Attach **MEP example file.zip**
2 MB Download

20.3.16 MEP flow connection points

20.3.16.1 Commands

BIMLINEARSOLID, BIMFLOWCONNECT

20.3.16.2 About

When creating flow elements like Flow Terminals and Flow Segments, you can add a 2D symbol and classify it as a **Flow Connection Point**. A flow connection point contains BIM data where you can predefine the profile you want to use with this flow component. When you select the flow connection point and use the commands BIMLINEARSOLID or BIMFLOWCONNECT, BricsCAD will automatically use a BIM Profile based on the given information.

See the related links for more information about Linear Solids.

See the related links for more information about MEP modeling.

20.3.16.3 Procedure: creating a flow connection point

In this procedure, you will learn how to create your own HVAC components with flow connection points. As an example, we will create an air heat pump unit.

Creating a flow connection point in the Library panel

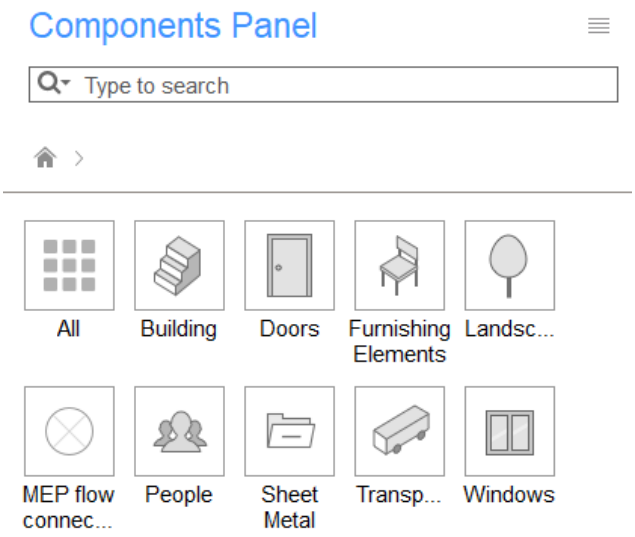
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and launch BIMCLASSIFY with the <Other> option or hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

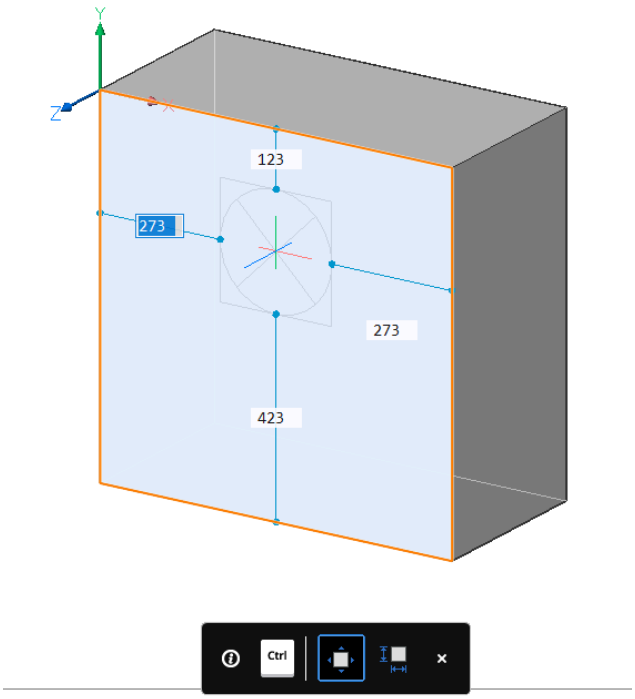
3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Open the **Library** panel.

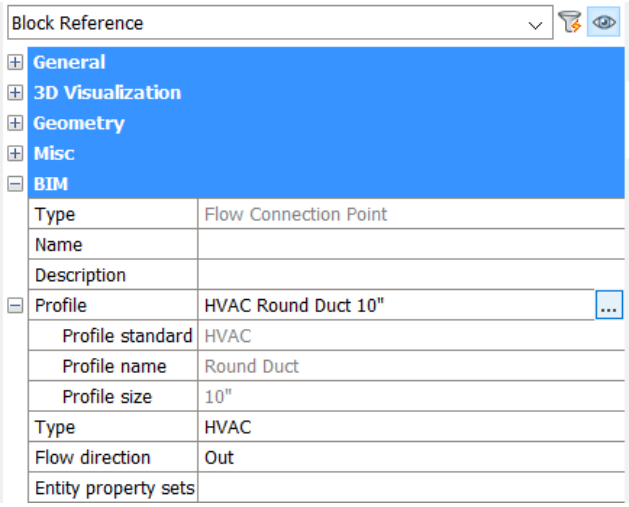
5 Go to the **MEP Flow Connection Points**.



6 Place HVAC Outlets Round on the heat pump box.



7 Select the Outlet and click the **BIM/Profile** property in the **Properties** panel.



8 Click the browse (…) button.

The **Profiles** dialog box displays.

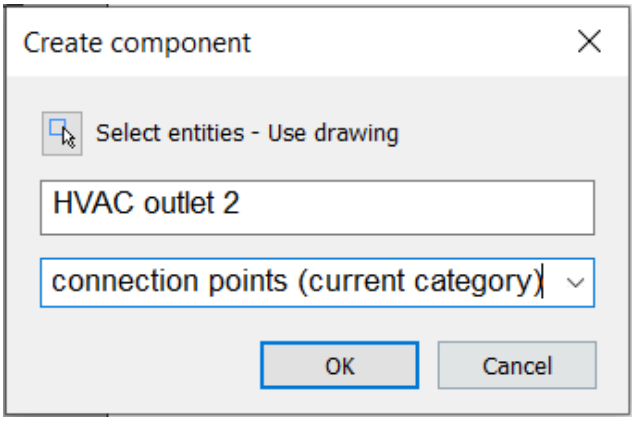
9 Set **Type** and **Standard** = **HVAC**.

10 Choose the desired profile and click the **Select** button.

11 Go back to the **Library** panel and click **Create Component**.

12 Choose a name and category and click the **OK** button.

See the **Components Panel** article for more information about adding a component to the library.



The drawing is now saved as a component and can be used in other drawings.

Creating a flow connection point from scratch

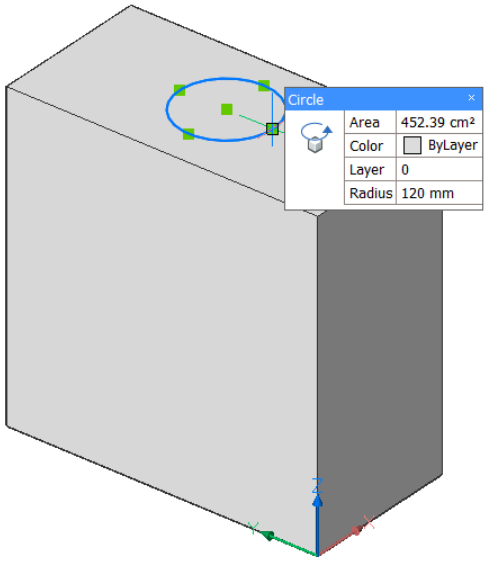
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and do one of the following:

- Launch BIMCLASSIFY with the **Other** option.
- Hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Draw a circle on the top face of the box.

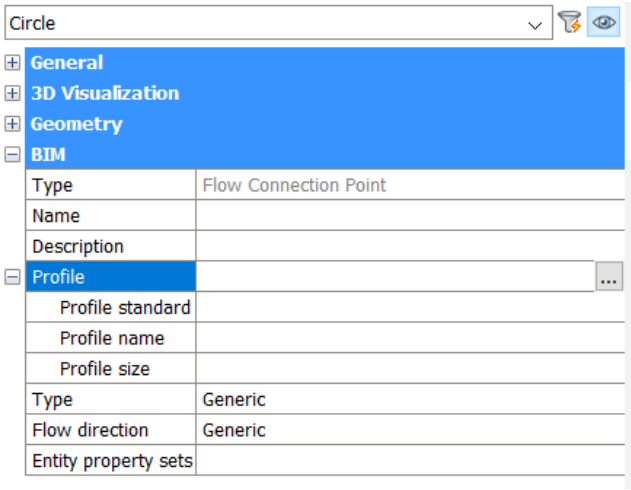


5 Select the circle and launch BIMCLASSIFY with the **Other** option.

6 Classify the circle as **Building Service Elements** > **Flow Terminal** and click **OK**.

7 Select the circle.

8 In the **Properties** panel, select the **BIM**/**Profile** property and click the Browse (…) button.



The **Profiles** dialog displays.

9 Select the desired profile and click **Select**.

10 On the **Library** panel and click **Create Component**.

11 Choose a name and category and click **OK**.

The drawing is now saved as a component and can be used in other drawings.

20.3.16.4 Procedure: Creating an HVAC System

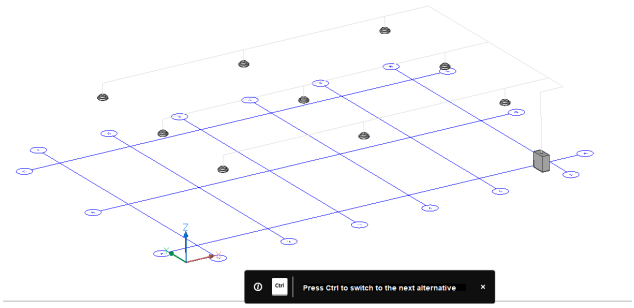
In this procedure, we will use flow connection points to create a ventilation system.

1 Let's start from a file with a series of flow terminals and a ventilation unit.

2 Select the flow terminals and the ventilation unit.

3 Select **FlowConnect** in the Quad or in the **Structural**/**MEP** ribbon tab.

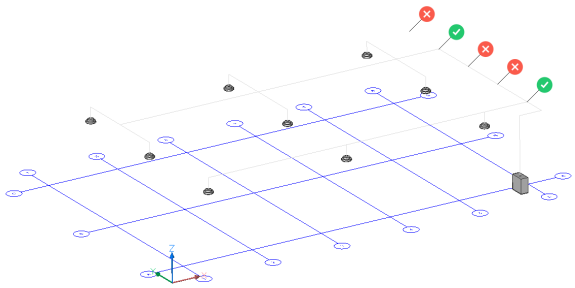
4 Press the CTRL-key to switch between the alternative setups and press Enter.



5 Click the **Tune** option or enter T in the Command line to modify the chosen setup.

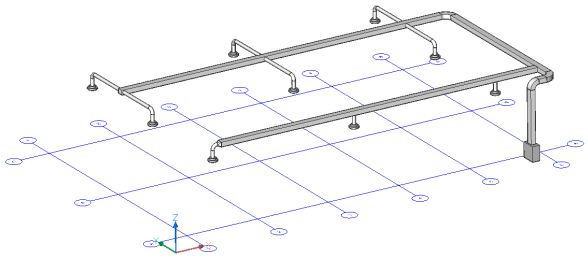
6 Select **Topology** in the toolbar or enter T and press Enter.

7 Select the check icons to modify the layout of the system.



8 Press Enter 3 times to exit the command.

9 ***Nota:*** BricsCAD automatically uses the profiles that were assigned to the ventilation units. This information is stored in the **Flow Connection Points**.



20.3.17 Offset

20.3.17.1 Commands

OFFSET

20.3.17.2 About

The **Offset** tool allows you to create copies of linear entities and align them parallel to the original entities at a specified distance.



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 the **Select Edges** option ( ) is enabled or on or set the SELECTIONMODES system variable to 1.
- The edges of a 3D solid face if the **Select Faces** option ( ) is enabled on or set the SELECTIONMODES system variable to 2.
- Detected boundaries if the **Select Detected Boundaries** option ( ) is enabled on or set the SELECTIONMODES system variable to 4.

20.3.17.3 Procedure: make a parallel copy of a 3D solid edge

Before starting the process, make sure that Quad is active and the **Select Edges** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor over the edge of the solid to highlight the edge.

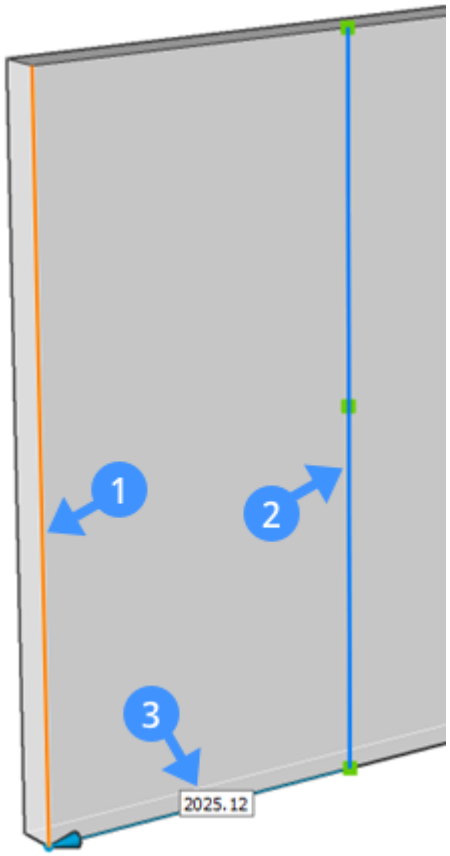
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by dragging the edge of the solid face.

Nota: 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.

5 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.



20.3.17.4 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.

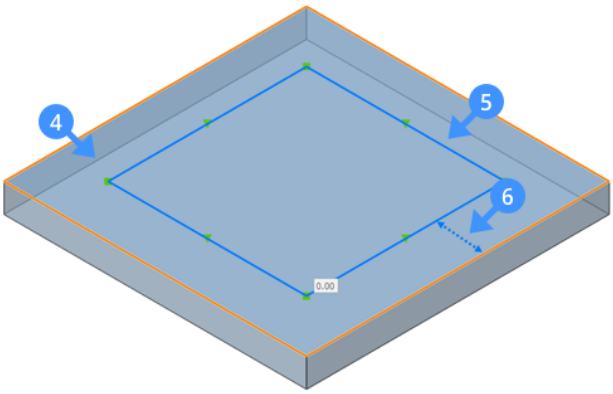
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by moving the cursor outside or inside of the solid.

4 The parallel copy of the face is created dynamically.

5 Enter a distance in the dynamic entry field and press Enter, or left-click to finish the offset.

6 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.



20.3.17.5 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.

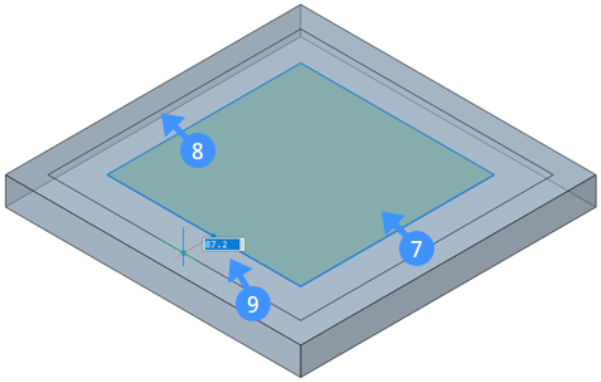
2 Choose the **Offset** tool from the Quad.

3 Move the cursor to define the offset distance and left-click.

4 The parallel copy of a detected boundary is created dynamically.

5 Enter a distance in the dynamic entry field to set a precise offset distance and press Enter or left-click to accept the offset.

6 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.



20.3.18 Splitting solids

20.3.18.1 Commands

BIMSPLIT

20.3.18.2 About

The BIMSPLIT command either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

20.3.18.3 To split a solid automatically

Make sure that the value of the SELECTIONPREVIEW system variable is 2 or 3.

1 a Do one of the following:

- Click the BimSplit tool button ( ) on the **Model** tab of the ribbon.
- Choose **Split** in the BIM menu.
- Type BIMSPLIT in the Command line, 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.

3 The face highlights.

4 Without moving the mouse, hit the TAB key until the complete solid highlights, then click.

5 You are prompted:

Entities in set: 1

Select solid for auto-splitting or cutting faces for manual splitting:

6 Right click.

7 The selected solid is split.

The number of parts is reported in the command window.

Using the Quad

Make sure the **Select Faces** option ( ) of the SELECTIONMODES system variable is switched off.

Place the cursor over a face of the solid.

When the solid highlights, choose Split ( ) in the BIM command group of the Quad.

The solid is split.

The number of parts is reported in the command window.

20.3.18.4 To split a solid using cutting faces

Make sure that the value of SELECTIONPREVIEW system variable is 2 or 3.

1 Do one of the following:

- Click the **BimSplit** tool button on **Model** tab of the ribbon.
- Choose **Split** in the **BIM** menu.
- Type BIMSPLIT in the Command line, 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.

3 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:

4 (Optional) Repeat step 4 to select more faces.

5 Right-click to stop selecting faces.

6 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 the **Select Faces** option of the SELECTIONMODES system variable is set.

1 Place the cursor over a face you want to use as a cutting plane.

1 When the face highlights, do one of the following:

2 Go to step 3.

3 Go to step 2 to select more faces.

4 (Optional) Place the cursor over a face you want to use as a cutting plane.

5 When the face highlights, do one of the following:

6 Go to step 3.

7 Repeat this step to select more faces.

8 Choose **Split** in the **BIM** command 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.

20.3.19 Stair tool

20.3.19.1 Commands

BIMSTAIR

20.3.19.2 About

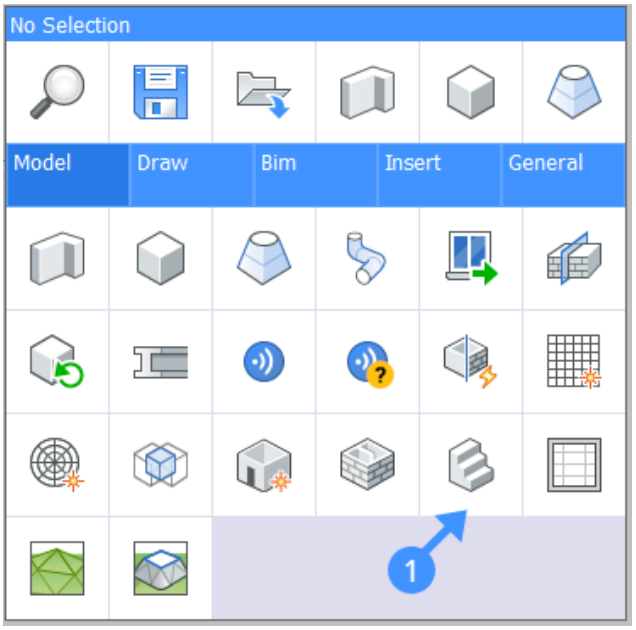
The **Stair** tool is a powerful tool that allows you to add a custom flight of stairs for your project in just two clicks. You can change the default stair settings as required. The **Stair** tool will automatically try to fit these requirements. It is possible to create single-flight straight, double-flight L-shaped, and double or triple-flight U-shaped stairs.



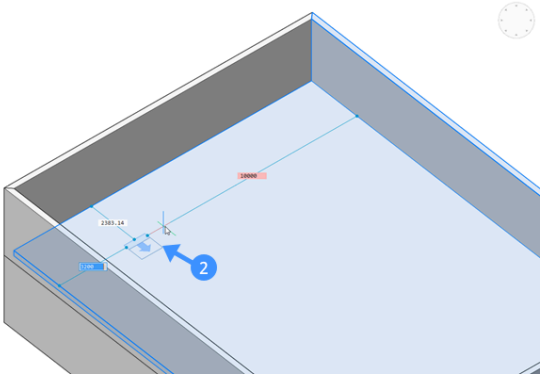
For more information about this command, visit the Command Reference article BIMSTAIR.

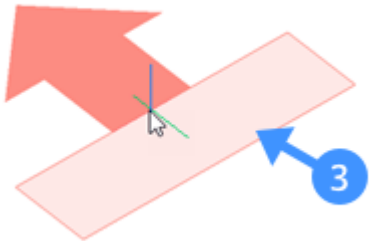
20.3.19.3 Create a straight flight of stairs

1 Select the **Stair** tool from the Quad (1).



2 Hover over a slab. The Stair Cursor will appear (2) and the slab on which the stair cursor hovers, is highlighted. This indicates the position of the first step. If the position of the cursor is invalid, the cursor will turn red (3).



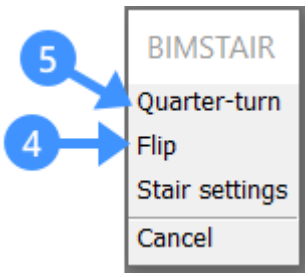


3 To flip the direction of the first step, click the **Flip** button (4) or type F.

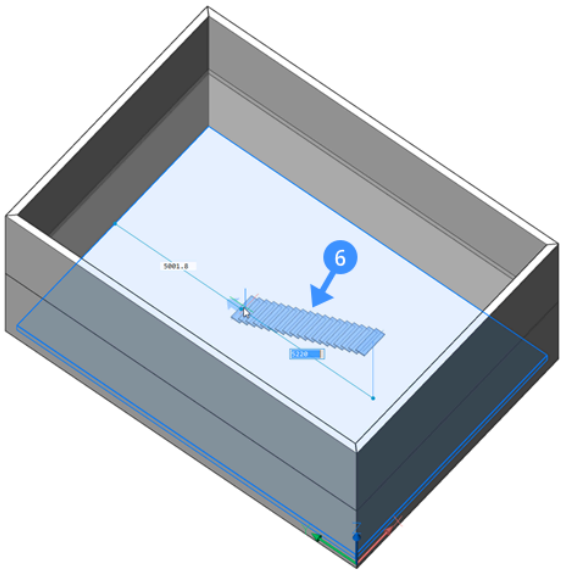
To turn the direction of the first step 90°, click the **Quarter-turn** button (5) or type Q.

To cycle through the different stair variants, the CTRL-key can be used when the Hotkey Assistant is active.

Nota: The cursor will automatically flip or quarter-turn when it is positioned close to a wall or the edge of a slab. This auto-quarter-turning/flipping behavior is only active when the relevant Stair Setting is turned on.

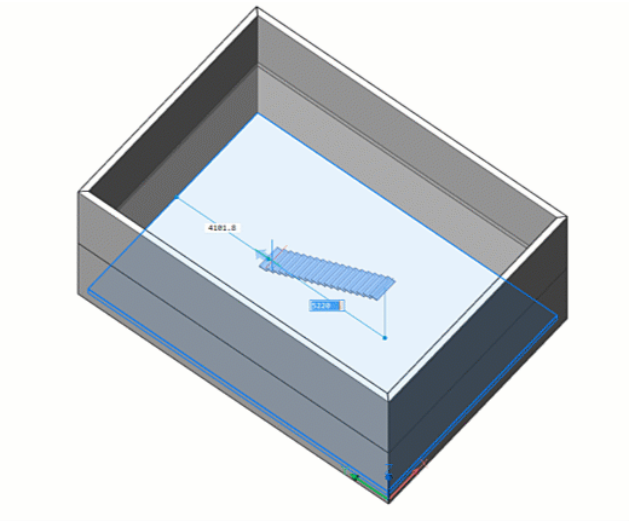


4 Once the cursor is in position, press Left-click. If the upper slab is blocking the view, the slab will become transparent.

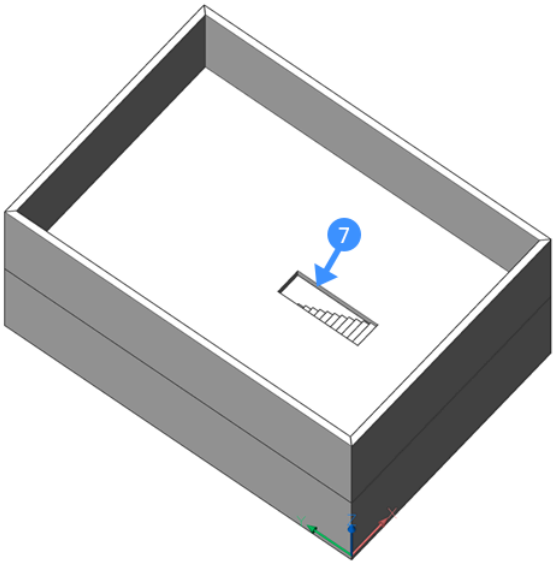


5 To create a straight flight of stairs, align the mouse perpendicular to the first step.

Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (6). The **Stair** tool will automatically snap to the configurations that correspond to "ideal step length" setting, as can be seen in the animation below. When the optimal configuration is satisfied, this preview will display in a darker shade of blue. The other settings are taken into account as well of course, but not to this specific snapping feature (where the stair preview turns darker blue).



6 Once you have the cursor in the correct position, left-click. A flight of stairs will be created. A hole is automatically generated in the upper slab, corresponding to the headroom setting (7).

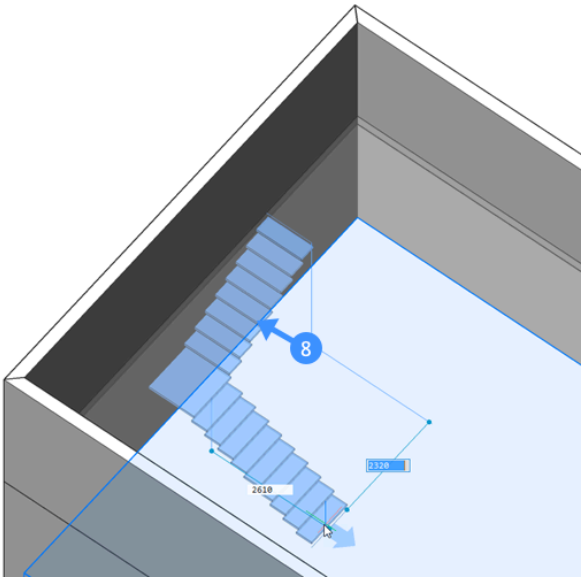


20.3.19.4 Create an L-shaped staircase

1 Select the **Stair** tool from the Quad.

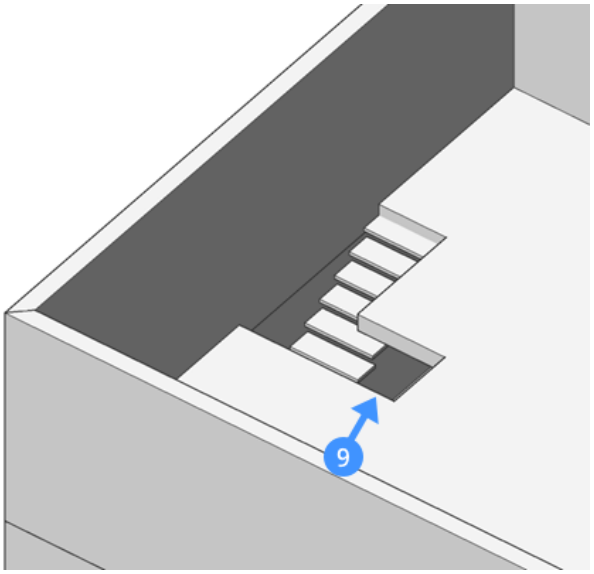
2 Hover over a slab. The Stair Cursor will appear. Left-click once the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to create an L-shape flight of stairs (8).



Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (8). The **Stair** tool will automatically try to find optimal configurations of the stair that correspond with the dimensions given in the settings dialog. It will snap to the optimal configuration and display this preview in a slightly darker color.

4 When you are satisfied with the preview, left-click. A flight of stairs will be created and a hole will be automatically generated (9). This time, the hole may be L-shaped, depending on the length of the upper flight and the "Headroom" stair setting. If the headroom is so large it cannot be covered with only the topmost stair flight, it will span also a part of the second stair flight and thus be L-shaped.



20.3.19.5 Create a U-shaped staircase

1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct

position.

3 Move the cursor away. You can type Q and Enter so that the direction of the second cursor is rotated 180° with respect to the first cursor.

Or use the CTRL-key when the Hot Key Assistant (**HKA** field in the **Status** bar) is ON.

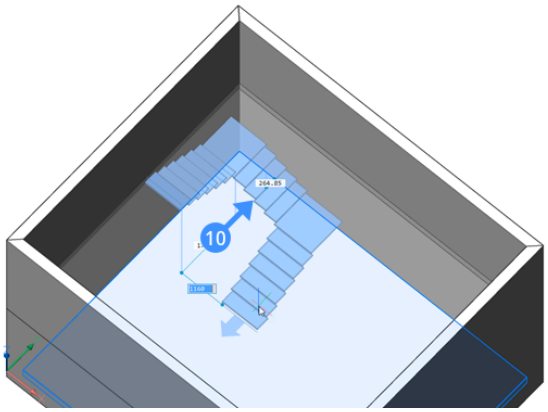
1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to get an L-shaped stair.

4 Press the CTRL-key for switching between an L-shaped stair and a U-shaped stair.

5 **Or**align the last step with the first fixed step. When entering a strip parallel to the first step, the last step will automatically quarter turn.



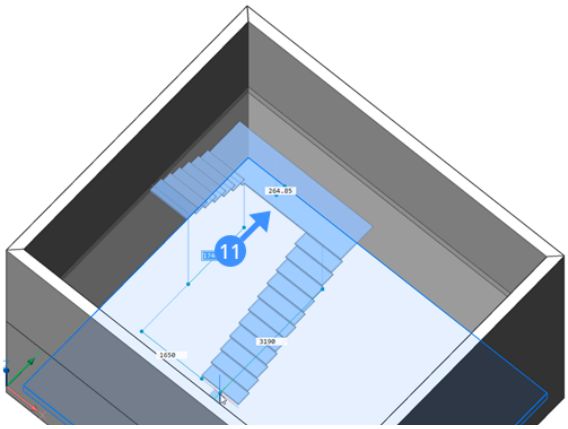
Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (10). The **Stair** tool will automatically snap to the configurations that correspond to the current settings. When the optimal configuration is satisfied, this preview will display in a darker shade of blue.

6 There are two possible types of U-shaped stairs you can switch between:

- The middle section includes steps (10).
- The middle section is a landing in between the two opposite stair flights (11).

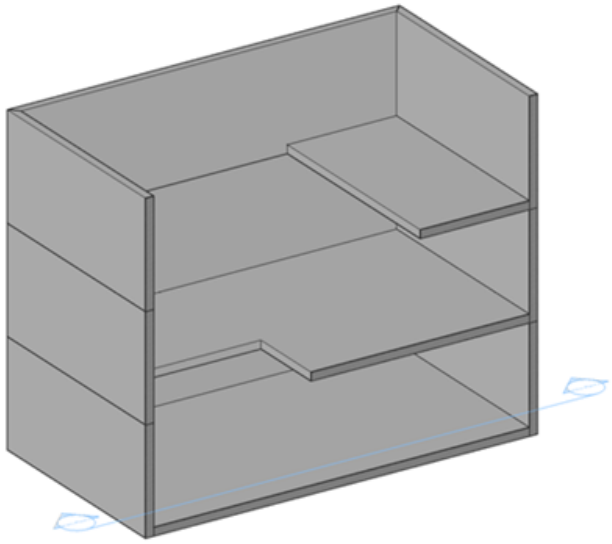
It is possible to switch between these two modes by typing U and hitting Enter.

Orpress CTRL to cycle between different stair variants.



20.3.19.6 Create a flight of stairs in an existing opening

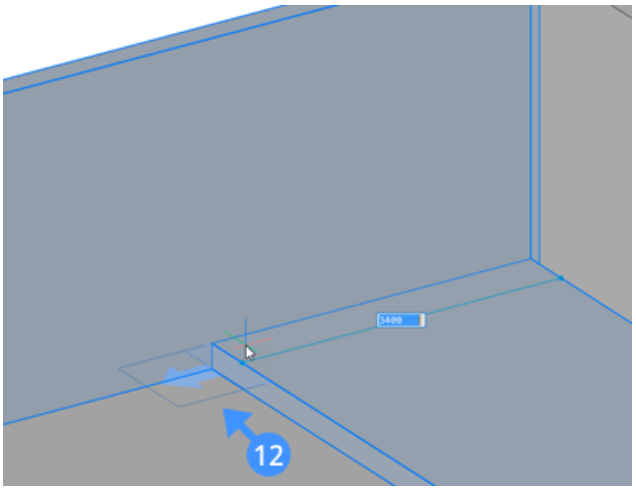
Sometimes it is necessary to create a flight of stairs starting from the edge of a slab, where an opening already exists.



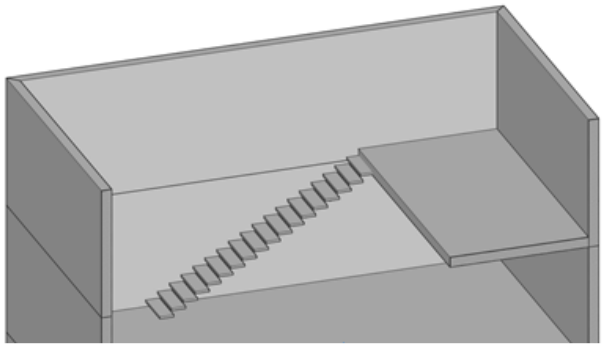
In the example above, different floors need to be connected with flights of stairs. However, an opening does not need to be created.

1 Select the **Stair** tool from the Quad.

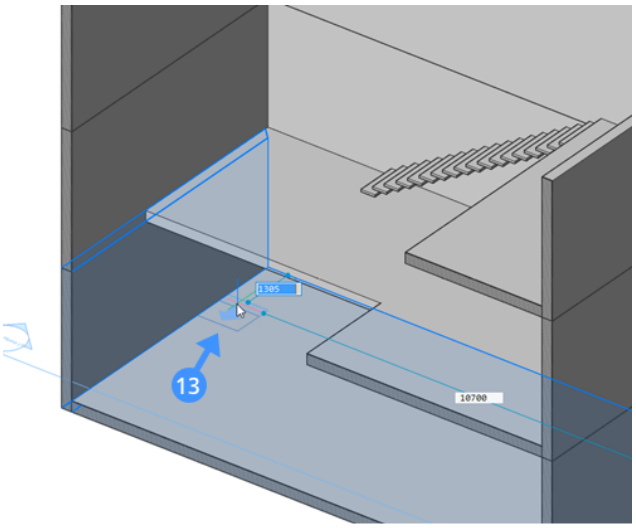
2 Hover the Stair Cursor near the edge of the slab (12). The cursor automatically snaps to the outer side of the edge.



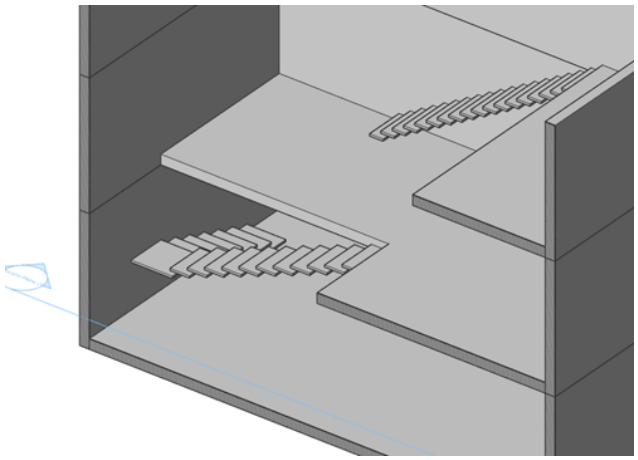
3 You can now create a flight of stairs as described above.



4 You can also make a flight of stairs by starting at the lowest point. This time select a starting point on the first floor (13).



5 Now hover over the edge of the slab on the second floor. The Stair Cursor automatically snaps to the edge of this slab. Left-click to create the flight of stairs.



20.3.19.7 Change the Stair Tool Settings

You can change the stair settings by clicking the **Stair settings** button in the **Stair** tool menu after the **Stair** tool command is launched or simply open the **Settings** dialog and search 'Stair'.

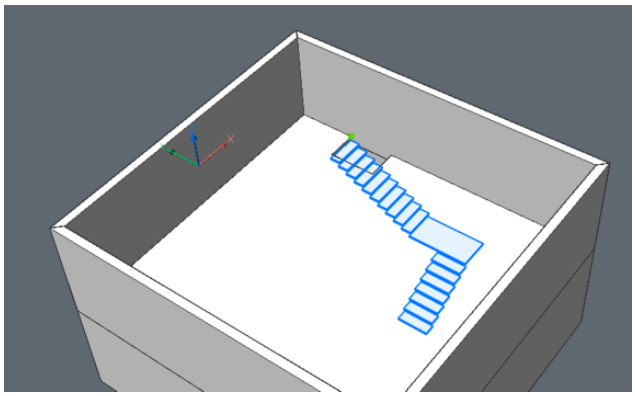
Edit a flight of stairs

You can adjust multiple parameters of a flight of stairs created with the **Stair** tool in the **Properties** panel. These parameters include:

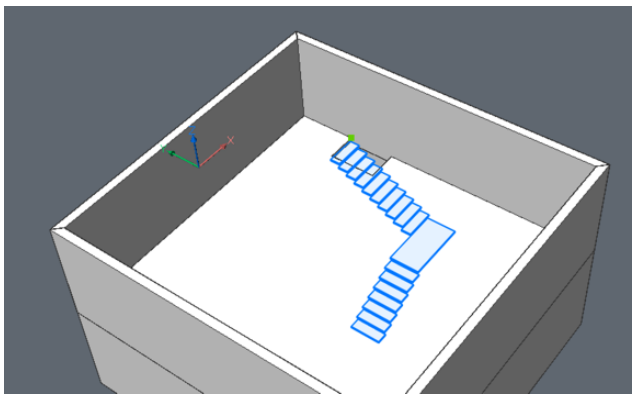
Parameters	
Height_Staircase	325 cm
Stair_Width	1000 mm
Ideal_Riser_Height	170 mm
Step_Thickness	50 mm
Nosing	50 mm
Length_StairFlight_0	145 cm
Length_StairFlight_1	174 cm
Length_StairFlight_2	145 cm

Height_Staircase	The overall height of the flight of stairs. The height is fixed at the top and will adjust from the base.
Stair_Width	The width of the flight of stairs. When a flight of stairs is selected a green dot appears on the upper slab. This dot indicates the insertion point of the flight of stairs. When this point is in the middle of the flight of stairs, the flight of stairs will widen from the center. When it is located at the side, for example when a stair is attached to a wall, the stair will only widen the side away from the dot.

Ideal_Riser_Height	The ideal riser height. When creating a flight of stairs, the real riser height will be calculated based on the ideal riser height and the height of the steps, so that the risers are equally divided as close to the ideal riser height as possible.
Thickness_Step	The step thickness. The upper face of the steps is fixed so that the thickness increases downwards.
Nosing	The horizontal distance of overlap between subsequent treads.
Length_StairFlight_0	The length of the first flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs.
Length_StairFlight_1	The length of an optional, second flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for L-stairs and U-stairs.
Length_StairFlight_2	The length of an optional third flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for U-stairs.
Landing_Extension_Up	The length of the landing going down the first flight of stairs. The first flight is fixed so the second flight adjusts to the given parameter value.
Landing_Extension_Down	The length of the landing going down the second flight of stairs. The first flight is fixed so the second flight adjusts to the given value.



Setting: **Landing_Extension _Up**.



Setting: **Landing_Extension_Down**.

You can re-position a flight of stairs created with the **Stair** tool using the MOVE command.

For complex edits to a flight of stairs created with the **Stair** tool, use the BEDIT command to edit the block definition. When in the BEdit-mode navigate to the **Mechanical Browser** and here you can see and edit all the constraints that apply to a flight of stairs.

20.3.20 Structural steel

20.3.20.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMSTRUCTURALCONNECT, BIMADDECCENTRICITY, BIMRECALCULATEAXIS, DRAG, BIMSTRETCH

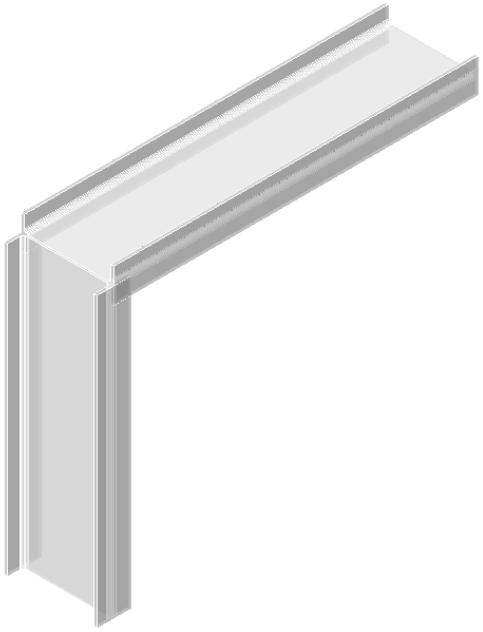
20.3.20.2 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the BIM **Profiles** panel to the model space.

Nota: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



Nota: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

20.3.20.3 About BIMSTRUCTURALCONNECT

The BIMSTRUCTURALCONNECT command automatically generates connections for profiles.

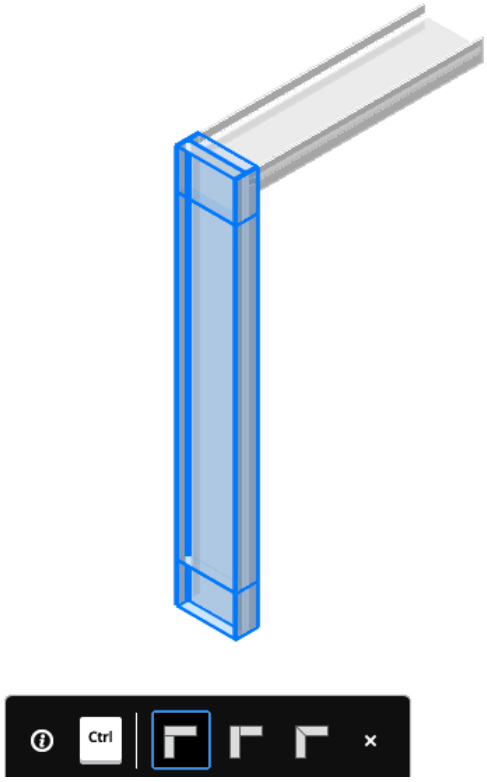
For more information about this command, visit the Command Reference article BIMSTRUCTURALCONNECT.

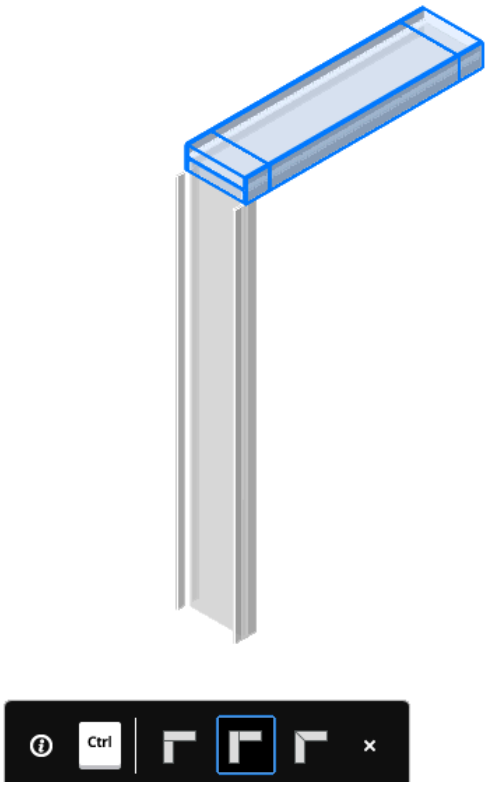
Making a structural connection to linear solids

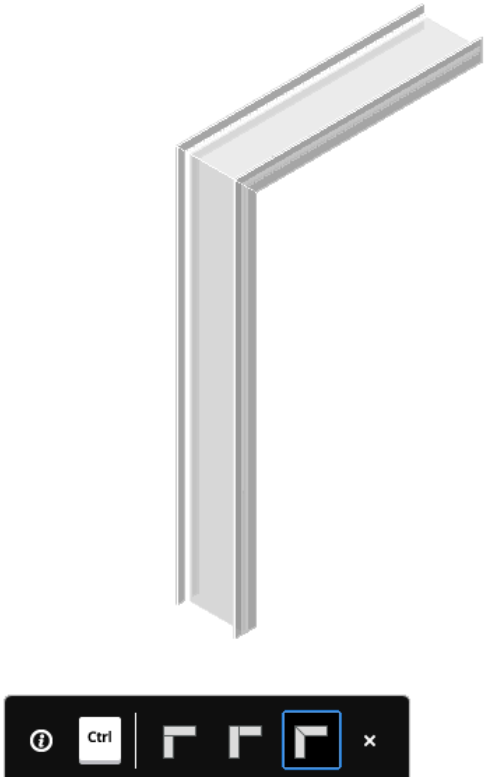
1 Select the profiles you want to connect with.

2 Launch the BIMSTRUCTURALCONNECT command. by typing it in the Command line or select **StructuralConnect** in the **Model** tab in the Quad.

3 When the Hot Key Assistant is on (**HKA** on the **status** bar), a widget (as displayed below) appears on the screen. Tap the CTRL-key to cycle through the different options.


Ctrl


Ctrl



4 Press Enter to accept the selected option and end the command.

20.3.20.4 About DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing the connection to other segments.



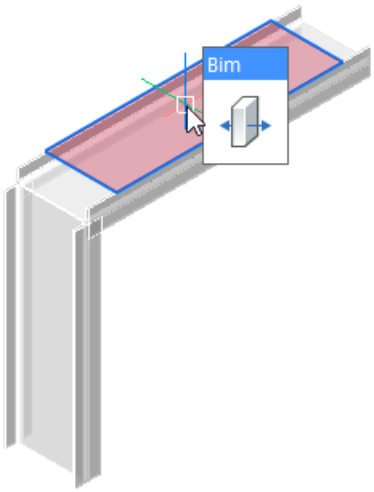
For more information about this command, visit the Command Reference article DRAG.

Altering the positioning of segments

Nota: This only works if the connections were created with the BIMSTRUCTURALCONNECT command.

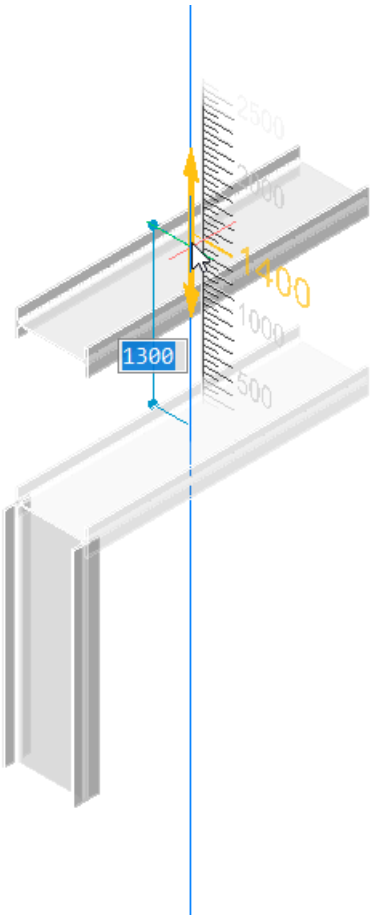
1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Select a face perpendicular to the section of the profile.



3 Launch the DRAG command.

4 Click the mouse to set a new position for the segment. Or use the dynamic dimension field to enter a value for the displacement.



20.3.20.5 Procedure: use modeling tools to create a structural steel model

1 Draw a BimGrid as a base.

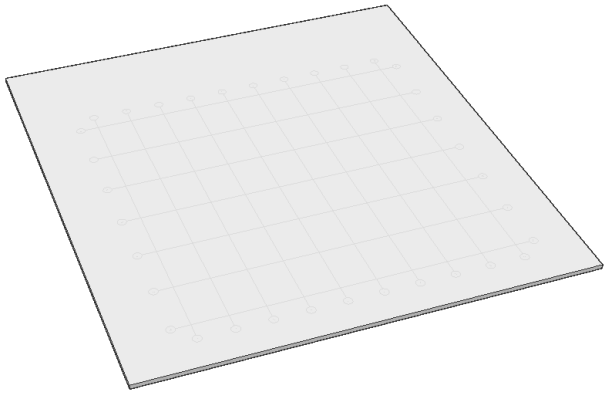
2 Launch the BIMGRID command.

3 Click a starting point and enter values into the dynamic dimension fields.

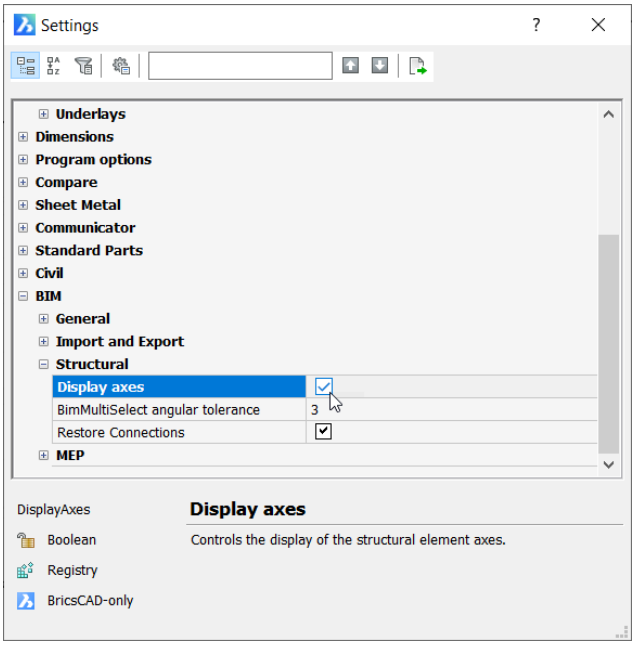
4 Draw a base slab underneath the grid.

5 Create a slab using the BOX command.

6 Ensure that the grid is entirely on top of the slab and that the axes of your profiles are displayed.

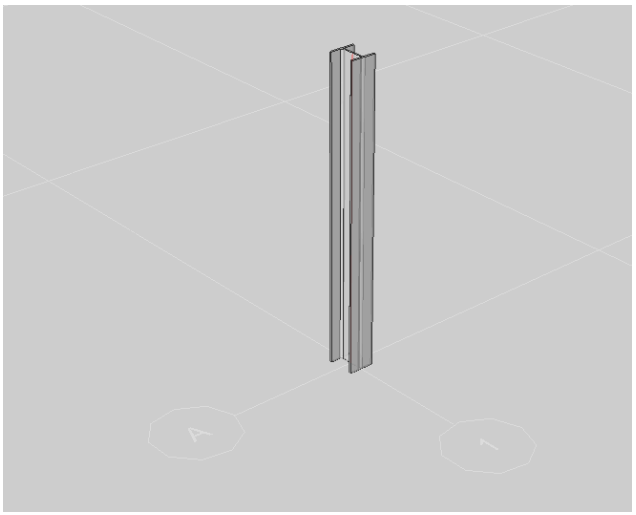


7 Go to the **Settings** dialog and under the **BIM** > **Structural** section. Ensure that **Display Axes** is checked.



8 Open the **Profiles** panel and drag a structural steel profile into your model. Start drawing at the intersection point of two grid axes and draw vertically upward.

Nota: To snap specific points, make sure you have toggled on the different types of points you want to snap in the context menu when right-clicking on ESNAP.



9 Launch the BIMPROPAGATE command.

10 Select the slab as base solid and press Enter.

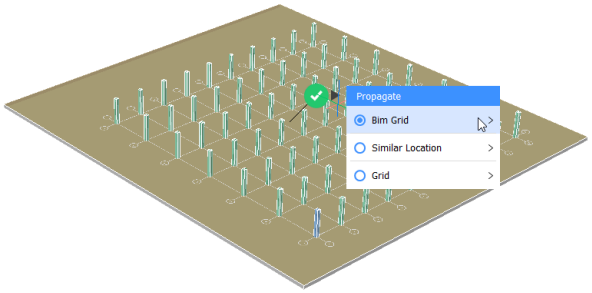
11 Select the column as your detail element and press Enter. BricsCAD will zoom into the 3D detail. Press C to copy the column as a solid, not as a block.

Nota: The default setting for BIMPROPAGATE is to copy the object as a block.

12 Hit Enter.

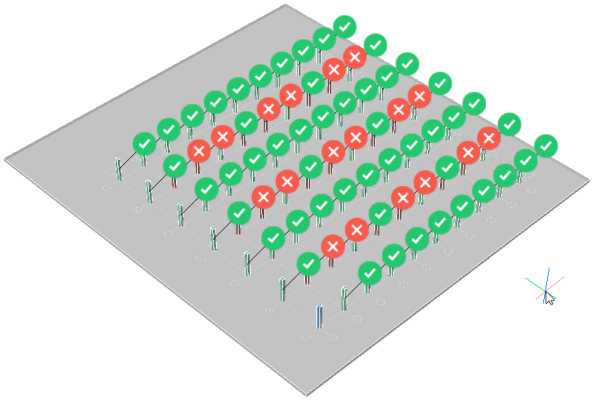
Nota: BricsCAD automatically copies the beam to every grid intersection. This is because you chose the floor slab with a BimGrid as your base solid.

13 Hover over the green checkmark and click **Bim Grid** in the dialog box.



14 Click **Explode**. You can now manually toggle individual suggestions to create the pattern of your choice.

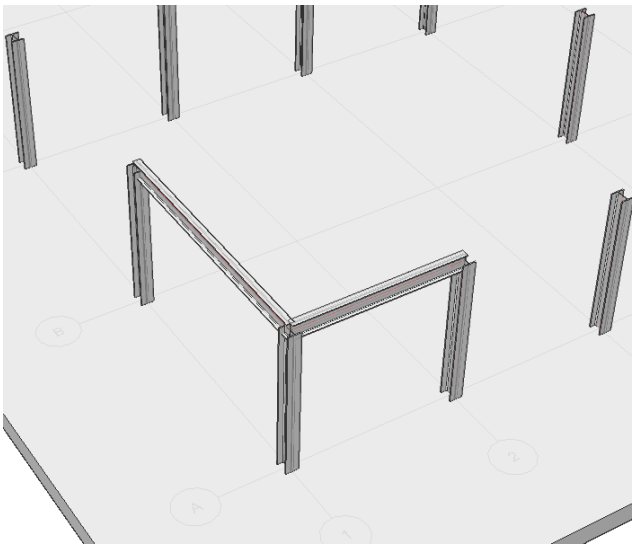
15 Press Enter to accept and end the command.



Create beams between the columns

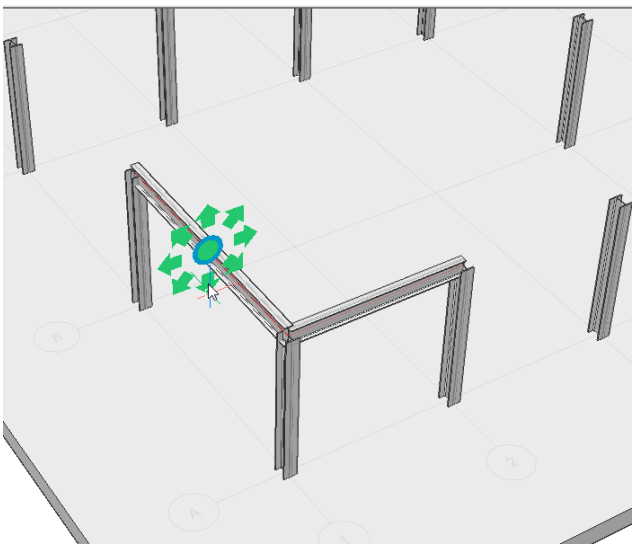
1 Open the **Profiles** panel and drag a structural steel profile into your model. To set the start and endpoints, click the top of the column and then the next. Do this in both directions.

Nota: If you want to turn your profiles to make the flanges are facing upwards, press Q (from Quarter turn). The profile will turn 90 degrees around its axis.



2 Drag and drop a beam profile from the **Profiles** panel onto this beam. Give the second beam a large profile, as it has a longer span.

3 Select a beam and launch the BIMADDECCENTRICITY command to alter the relative position of the axes of the beams.

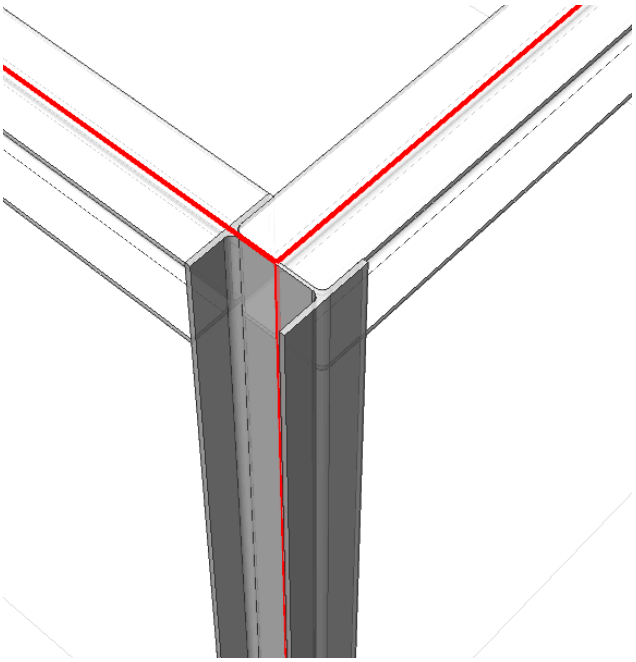


4 Click the down arrow. The beam moves down in relation to its axis.

5 Press Enter to end the command.

6 Repeat steps 3-5 for the other beam.

7 Select the column and the two beams then launch the BIMSTRUCTURALCONNECT command to make a simple connection. Press Enter to accept the default option.



8 Repeat step 7 for the other 2 junctions.

9 Select the two columns connected to the beam in the Y-direction and launch the BIMPROPAGATE command.

10 Select the beam in the Y-direction as the detail solid.

11 Press C to copy it as a solid, not as a block.

12 Press Enter.

13 Choose which suggestions to accept. Press Enter to accept and finish the command.

Nota: Suggestions are made at the bottom of every column because the situation is symmetrical. You can toggle the suggestions off.

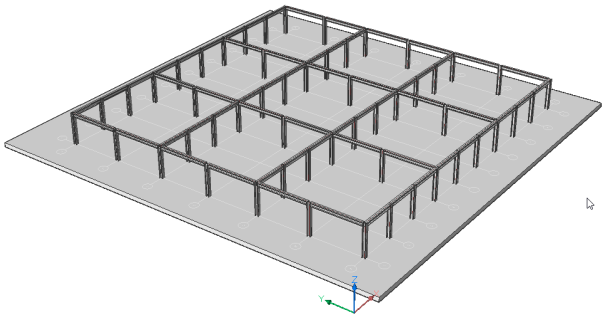
14 Repeat steps 9 - 13 to create beams on the X-direction.

Nota: You can use box selections to select all the lower beams at once.

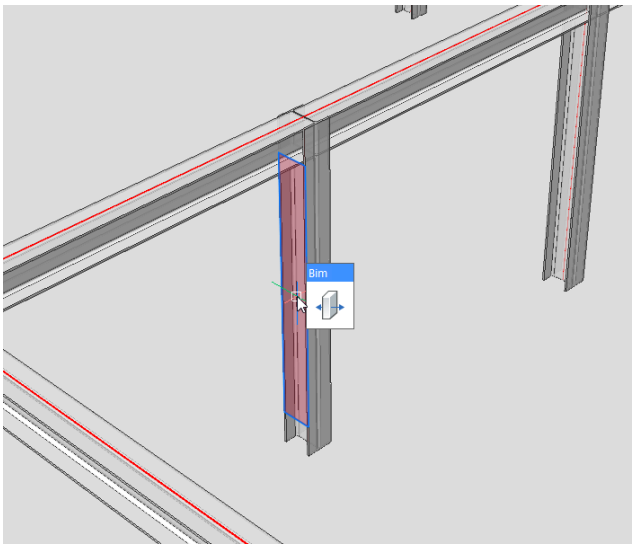
Move a Beam

1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Highlight the side face of the desired column and launch the DRAG command.



3 Move the column to the desired position.



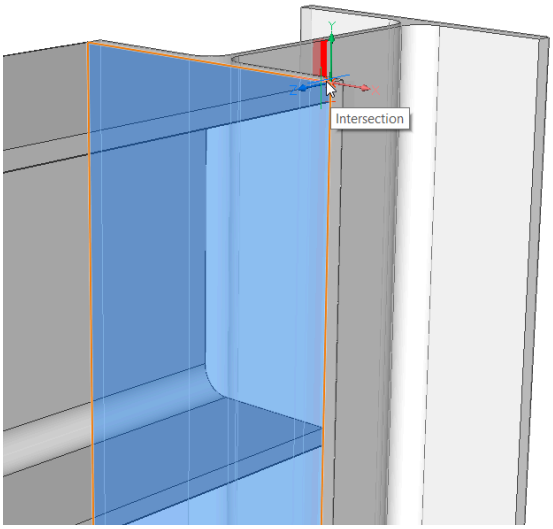
Nota: The connections with the adjoining beams are maintained.

20.3.20.6 Procedure: use the modeling tools to create a more detailed connection

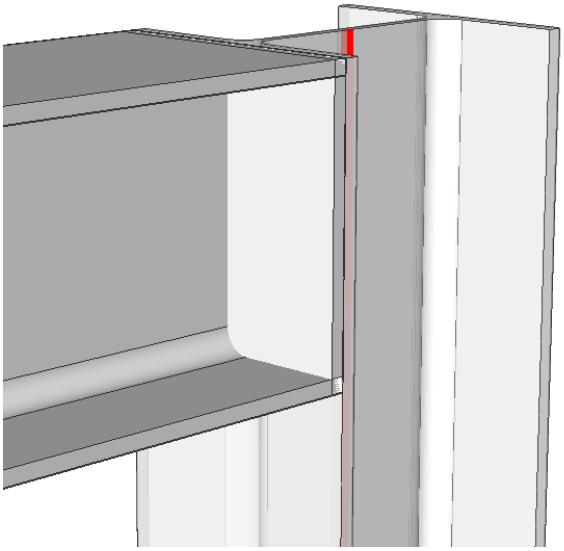
1 Set the value of DISPLAYSIDESANDENDS system variable to 0.

2 Launch the BOX command and hover over the side face of the column.

3 Press Shift once so that the face is highlighted in blue.

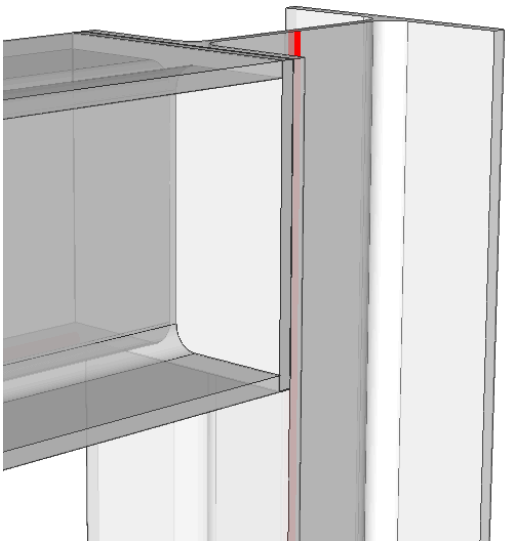


4 Now draw a plate on the flange of the column.



5 Select the beam and launch the SUBTRACT command.

6 Select the endplate and press Enter.

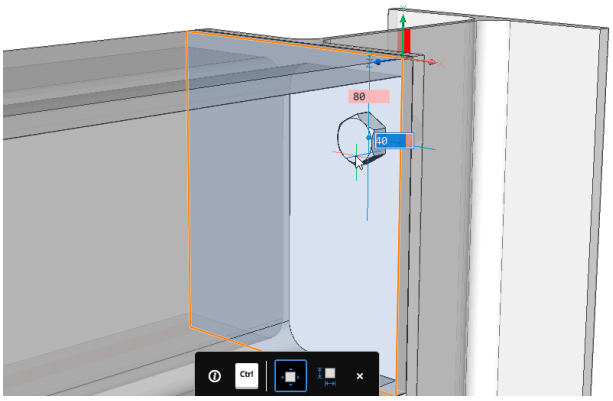


7 Select the face and launch the BIMIFY command from the Quad.

Nota: You can also Classify the plate manually by Quad selecting **Classify Manually** in the **BIM** tab.

8 Open the **Library** panel and drag a bolt onto the plate.

Nota: You can position it precisely by entering values in the dynamic dimension fields.

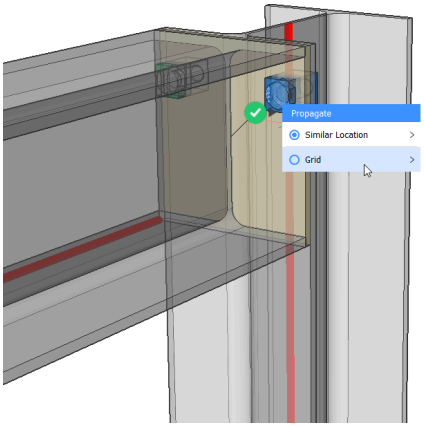


9 Select the plate and launch the BIMPROPAGATE command.

10 Select the bolt as the detail and press Enter.

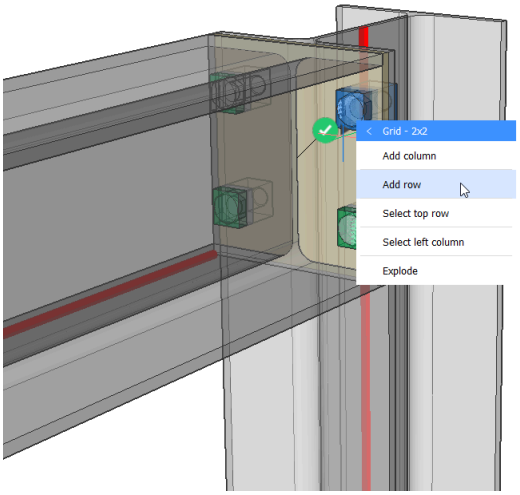
11 Press Enter again and click on the blue question mark. It will turn into a green checkmark.

12 Hover over the checkmark and select **Grid**.

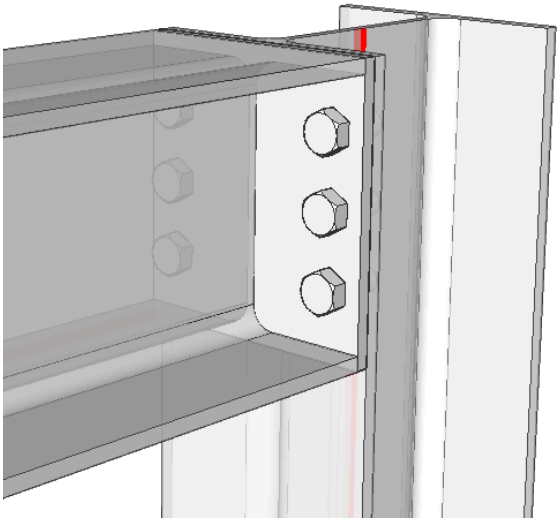


13 Add one row.

14 Press Enter to end the command.



The detailed connection has now been created.



20.3.20.7 Procedure: using BIMSTRETCH to modify structural steel

1 Open a New drawing.

2 Open the **Profiles** panel by clicking on it on the right-hand side of your screen.

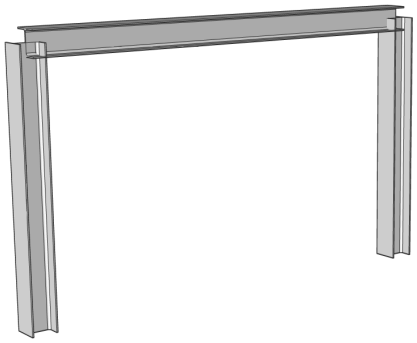
Nota: If the **Profiles** panel icon isn't visible yet, right-click on the ribbon and check **BIM Profiles** in the context menu under **Panels**.

3 Drag the profile you want to draw a structure with within the model space.

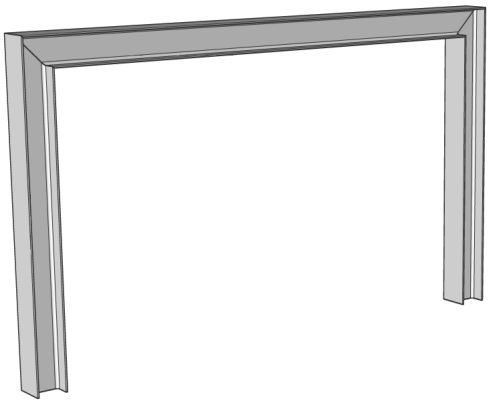
4 Draw a column with a height of 3000 mm. Don't exit the command and draw a connecting beam with

a length of 5000 mm horizontally. Now, add another column at the end of the beam. The height of this column is also 3000 mm.

Nota: Make sure you turned the profiles in the right way.



5 Select the 3 beams and select **Connect Structural** in the ribbon or select the command from the Quad. You will be able to choose which connection you prefer. Choose the L-connection option and press enter. Your entities are now connected.



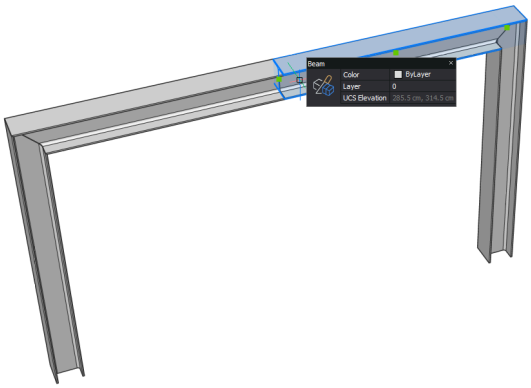
6 In the ribbon, make sure you enable the **Display sides and ends** and **Prioritize selection of Faces**. You can find them in the **Structural**/**MEP** – **View** tab or set the value of DISPLAYSIDESANDENDS system variable to 1 and SELECTIONMODES system variable to 2.

7 Check if the profiles are correctly classified as **Beam** and **Columns**.

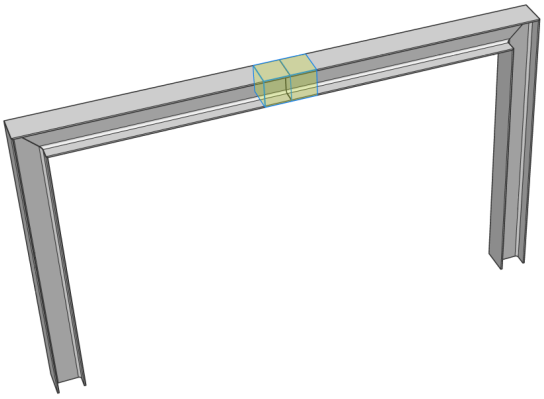
8 Launch the SLICE command and slice the beam in the middle.

9 You will be prompted: Slice along axis? Press Enter to accept the default option (Yes).

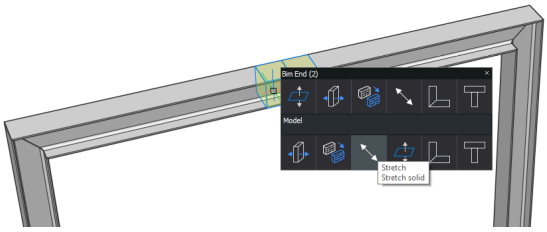
10 Select the midpoint of the beam and press Enter. Your beam is now sliced into two beams.



11 Select the 2 ends of the beams in the middle of the structure. They will be highlighted in yellow.

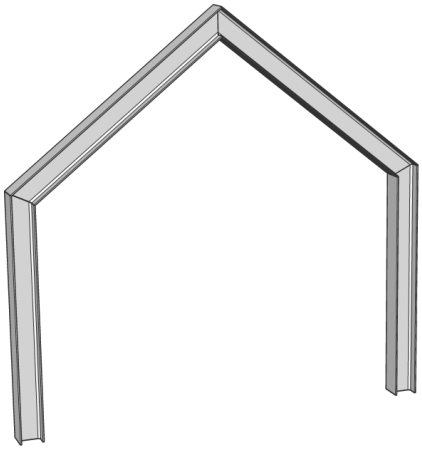


12 Quad select the BIMSTRETCH command in the **Model** tab.



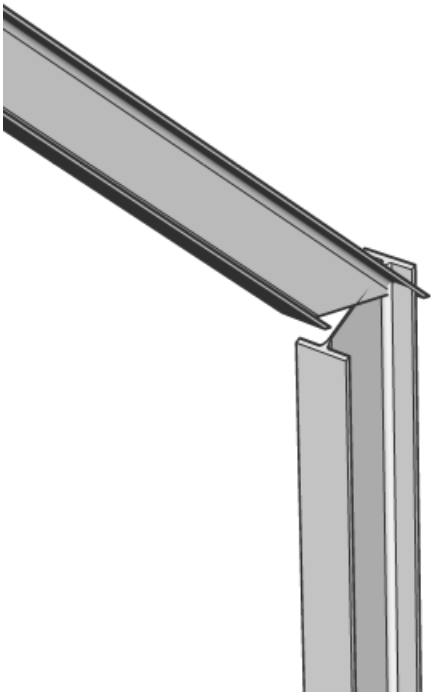
13 Hover orthogonally upwards, enter a height of 2000 mm and press Enter to accept.

14 The result should look like this.



As you can see, the connections between the profiles are automatically adapted to the new angle between the entities.

Nota:If you would only have selected one of the two side-ends in the middle of the beam, the connections on the side that was not selected won't be included in the stretch and will look like this:



20.3.21 Using BIM quickdraw

20.3.21.1 Commands

BIMQUICKDRAW

20.3.21.2 About

The **Quickdraw** tool allows you to get started with conceptual modeling easily. It also automatically adds BIM data. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, **Quickdraw** will remain active until you explicitly exit the command or until another command is executed.

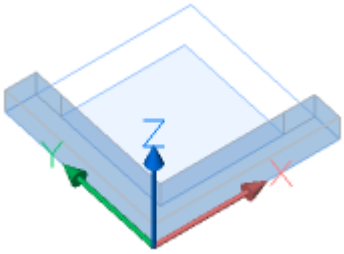


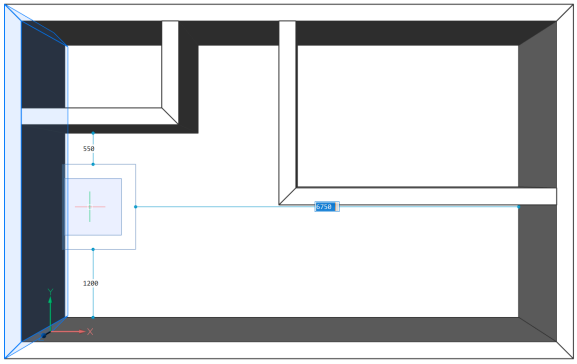
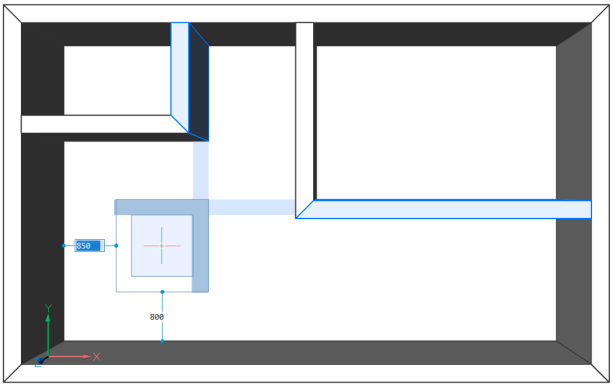
For more information about this command, visit the Command Reference article BIMQUICKDRAW.

20.3.21.3 The magnetic snapping feature

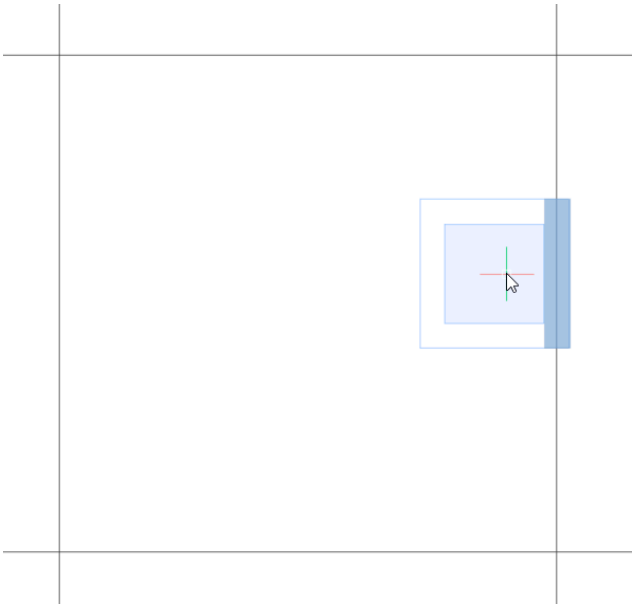
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object.

You have the following options:

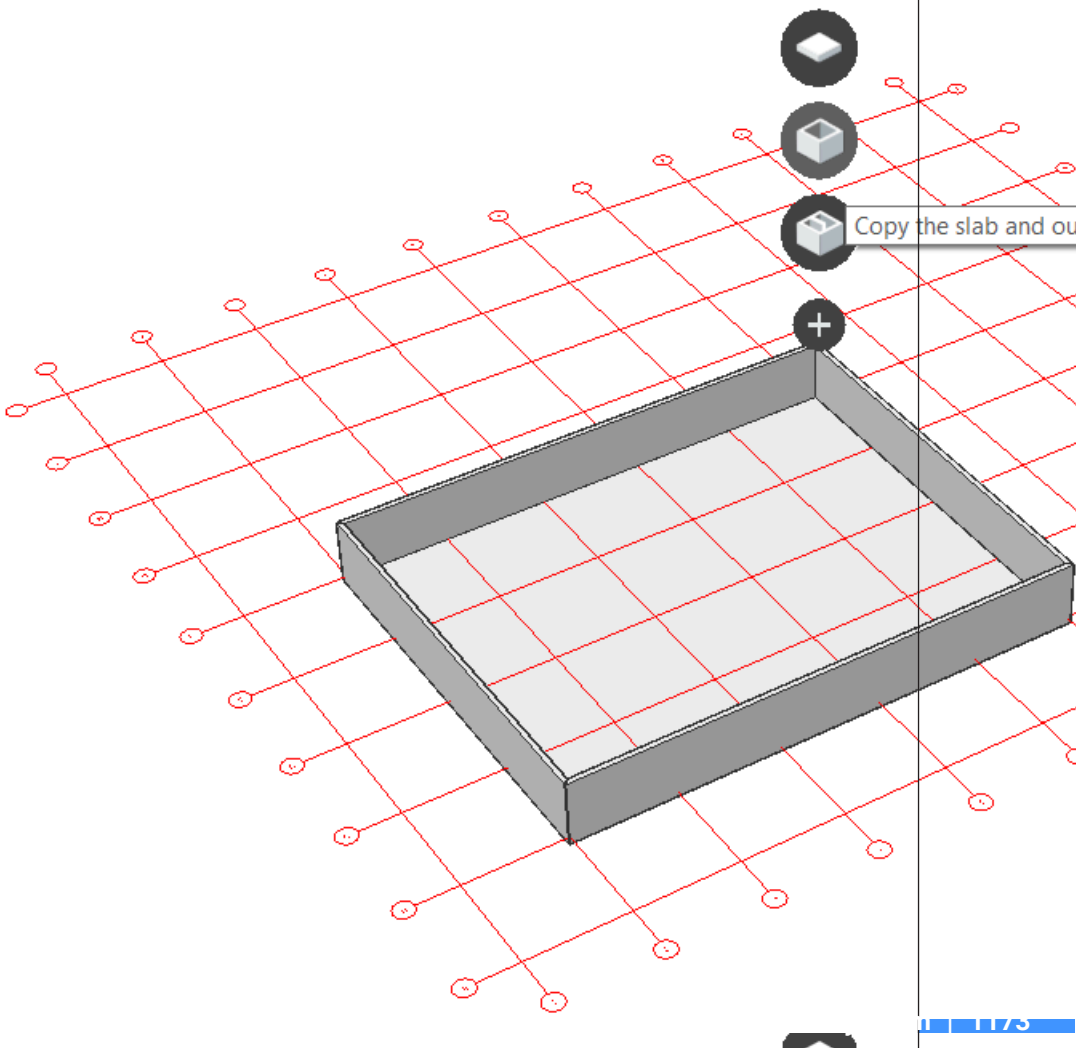
1. Align to the origin	

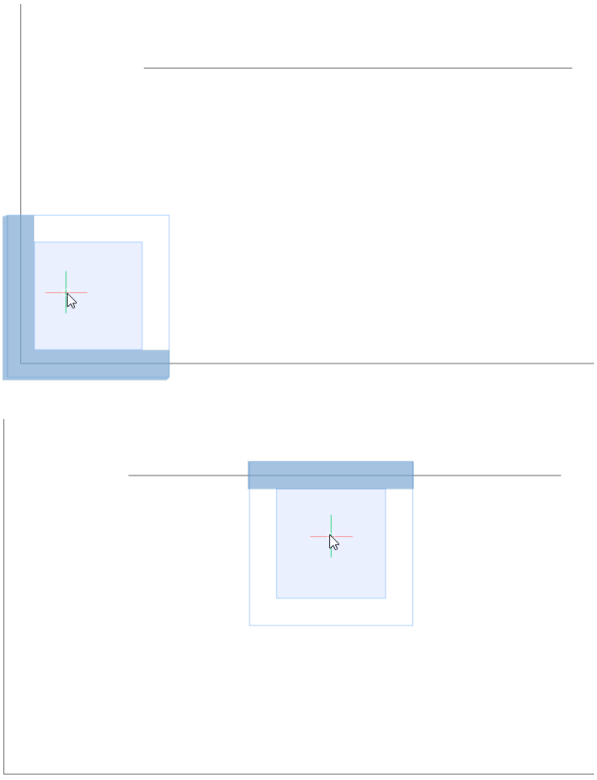
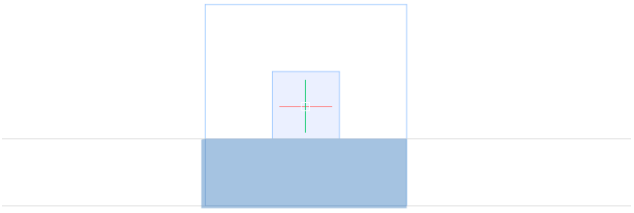
2. Align to existing walls	To the wall itself  To the extents of the wall 

3. Align to lines	To grid lines



Nota: If you use **Quickdraw** to copy a story, the gridlines will only be copied if the grid is assigned to the same spatial location of the ground floor slab and walls. If this is not the case, the gridlines that lie in the boundary of the ground slab will be projected on the slab of the top floor. This is only a visual help. The gridlines on the slab of the top floor will disappear when you exit the QUICKDRAW command.



	To X- and Y- aligned lines and line segments 
	To parallel lines 

20.3.21.4 Setting the Quickdraw system variables

You can use the **Quickdraw Settings** to set the Quickdraw system variables:

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the

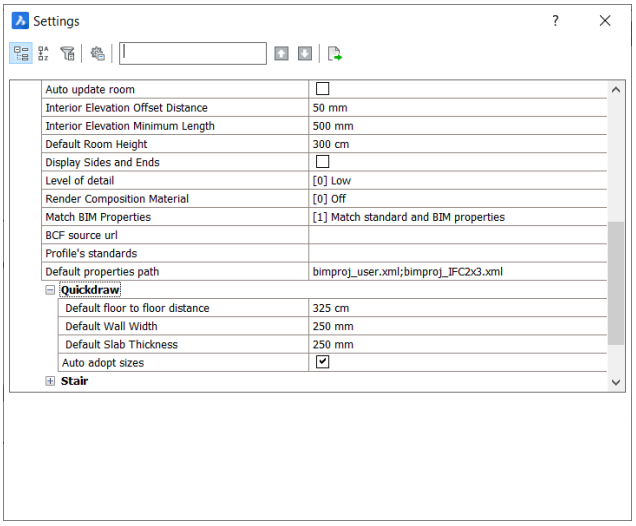
Command line.

2 You are prompted: Select first point or [quickdraw Settings]:

3 Enter S for the **Quickdraw Settings**.

4 Specify a new value to the system variables of **Quickdraw** to change their default values (e.g. **Default Wall Width**, **Default Slab Thickness**, and Default floor to floor distance).

When the **auto adapt sizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to.



20.3.21.5 Changing system variables while running Quickdraw

It is possible to edit the dimensions of the walls you will draw while the QUICKDRAW command is launched. Change these settings in the **Quickdraw Settings** dialog box, select the settings in the prompts menu or enter the specific letters in the Command line and press Enter to change the dimensions.

- H: Change the dimension of the floor height.
- W: Change the dimension of the wall width.
- T: Change the dimension of the slab thickness.
- A: Adapt the dimensions of the room you are drawing in/against automatically.

Nota: Make sure that you enable the hidden options in the prompt menu. Set the value for PROMPTMENUFLAGS system variable to 1.

20.3.21.6 Procedure: creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

You are prompted: Select the first point or [quickdraw Settings]:

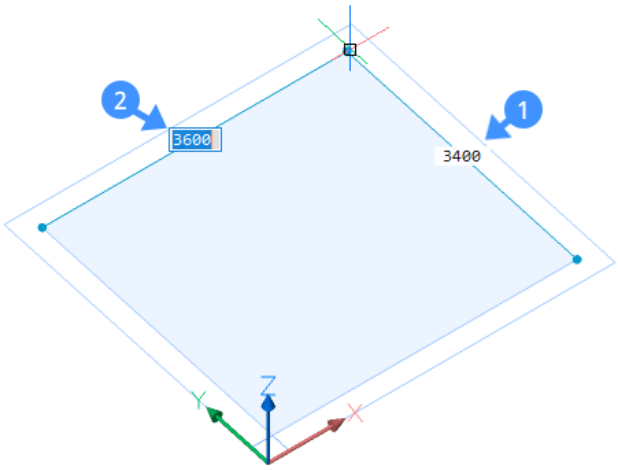
Nota: Use the **Quickdraw Settings** to change the wall width, wall height and slab thickness of the room before you click and then specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

Nota: To start from the origin, use the magnetic snapping feature of **Quickdraw**.

3 Drag the mouse cursor to define the footprint of the building.

Nota: If the **Dynamic dimension** (DYN) option is on, the current values of the room width (1) and length (2) are displayed in a dynamic dimension field.

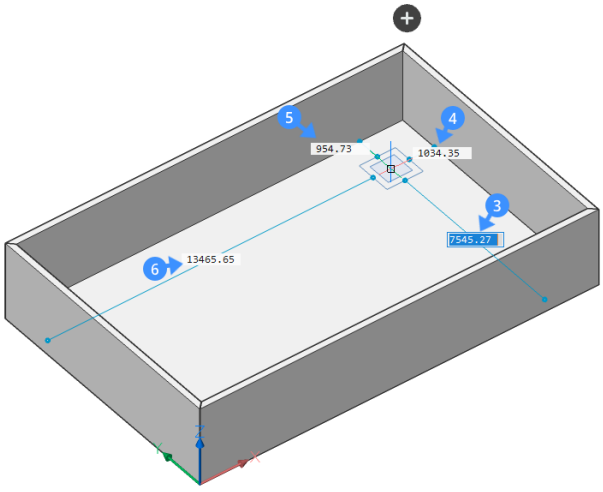


Specify the values of the room width and length in the Dynamic dimension field. Press the Tab key to switch between width and length and press Enter or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

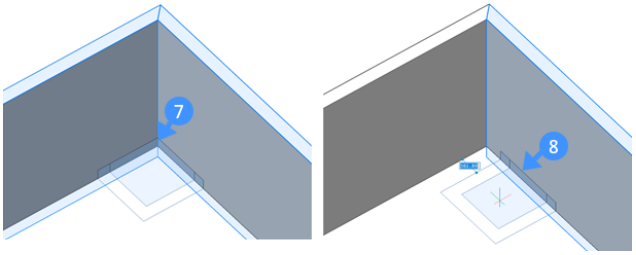
4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).

When the "auto adapt sizes" option is turned on, the dimensions of the walls of the new room will be the same as the wall you snap to.



If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



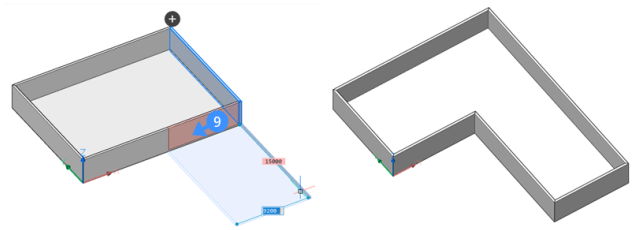
To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

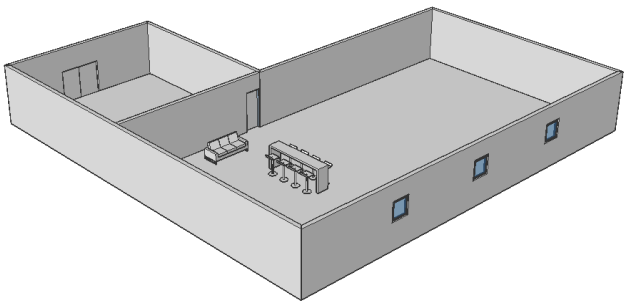
Nota: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.



20.3.21.7 Procedure: creating a story and a roof

Start with a similar configuration:



Go through the following steps to create two extra stories, a floating part and a flat roof.

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab or type BIMQUICKDRAW in the Command line.

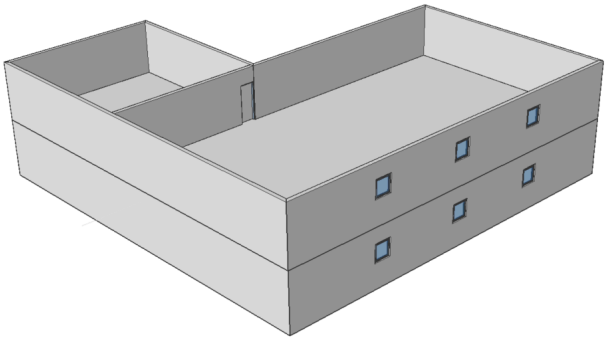
2 Click on the icon (⊕) to see the options:

- Fully copy the top floor. All features will be copied (windows, doors, interior walls, and

furniture, excluding doors on the outer walls of the ground floor).

- Copy the slab and the outer walls of the top floor.
- Finish the building with a flat roof.

3 For the first story, choose the first option (Full copy).

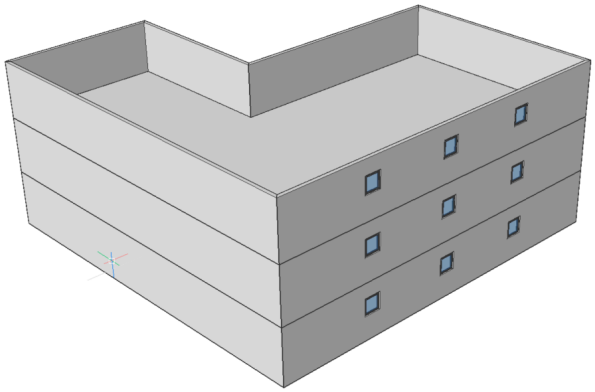


Nota: The interior wall and interior door and the windows in the exterior wall are copied. The door in the exterior wall on the ground floor is not copied.

Nota: The furniture is not copied in this case, because it is not assigned to the ground floor (see the **Properties** panel). If you want to copy the furniture, use BIMIFY to assign the furniture to a spatial location. The new furniture will be automatically assigned to the new spatial location.

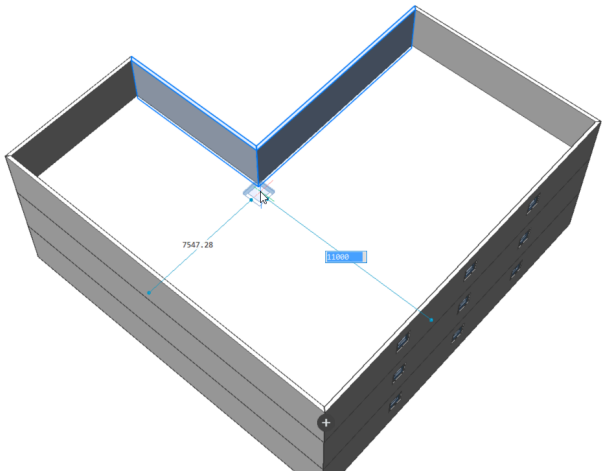
Nota: Walls inherit the spatial location of the slab on which they are created.

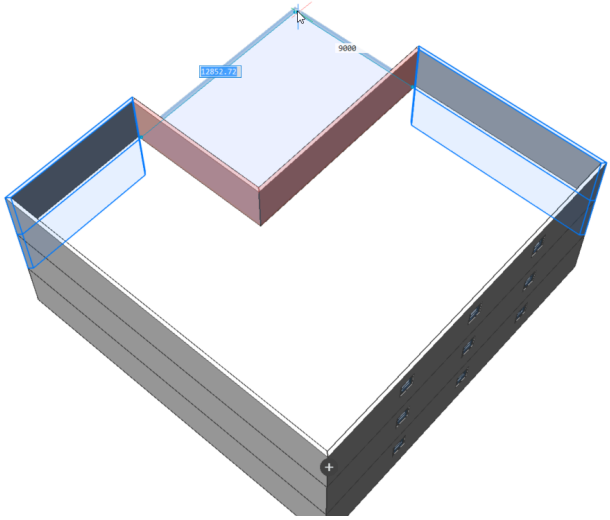
4 Repeat step 2. Choose the second option (Copy the slab and the outer walls) to create the top story.



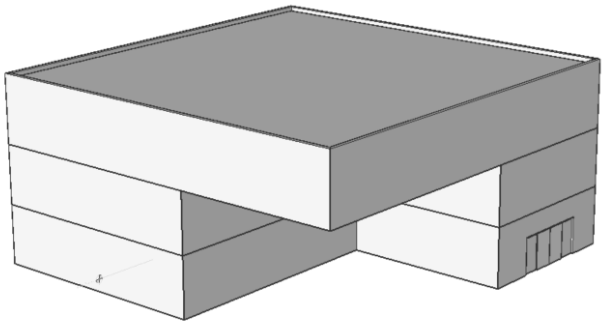
5 To make the overhang you should remove parts of the outer walls of the second floor. First, snap the Quickdraw cursor in the corner of the two outer walls, inside the building. Then click and drag the cursor outside.

6 The outer walls will turn red to indicate that the walls will be removed.



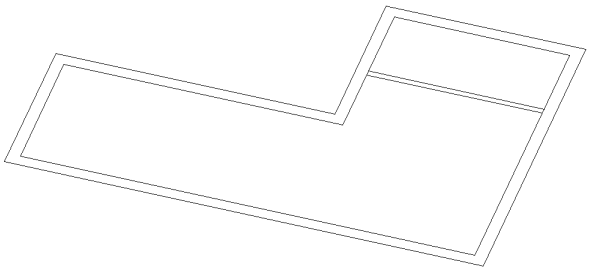


7 Use the same method as in step 2, selecting option 3 (Finish the building with a flat roof).



20.3.21.8 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

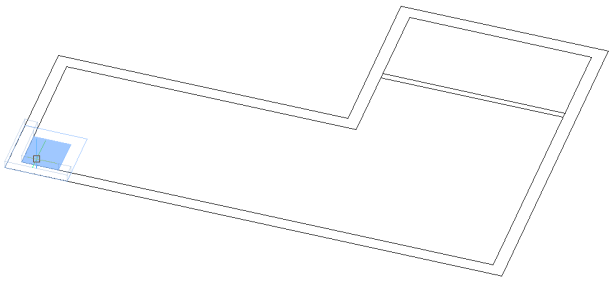


Nota: It is also possible to draw a simple 2D floor plan with the (POLY)LINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

Nota: The Quickdraw cursor now supports snapping to (double) lines inside XREFs.

2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

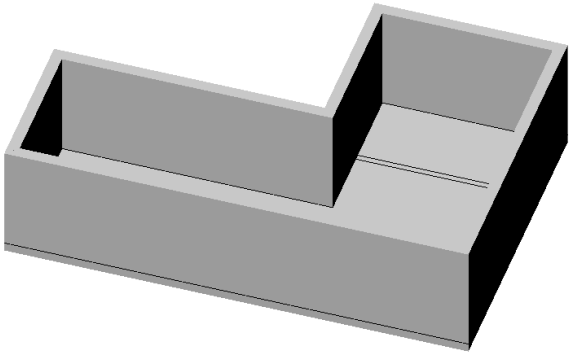
3 Hover over one of the outer walls with the cursor.



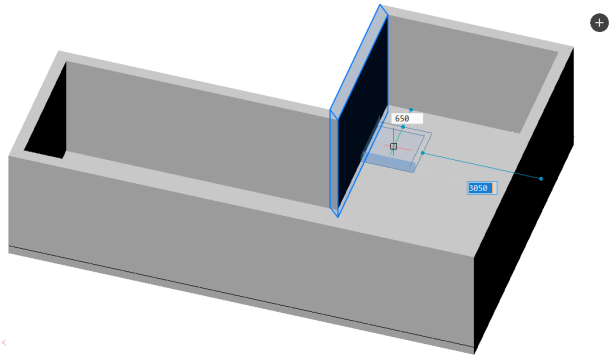
Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

Nota: If the lines in the (imported) 2D floorplan are not optimal (e.g. not exactly X- or Y-aligned), the QuickDraw-cursor will not snap to it (as it is perfectly X-Y-aligned). In that case, use the Optimize command to make sure all 2D lines exactly X- or Y-aligned.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

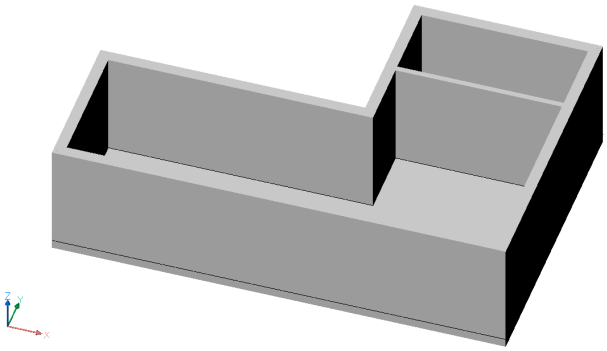


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



Nota: you can use the same method as in step 2 and 6 of the previous procedure to add floors or finish the building with a flat roof.

20.3.22 Using drag

20.3.22.1 Commands

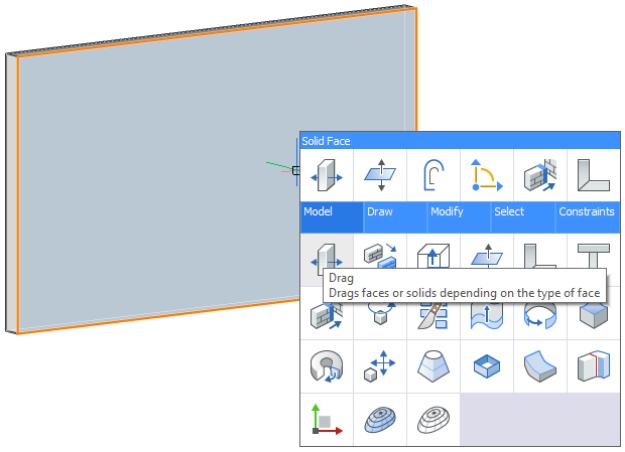
DRAG

20.3.22.2 About

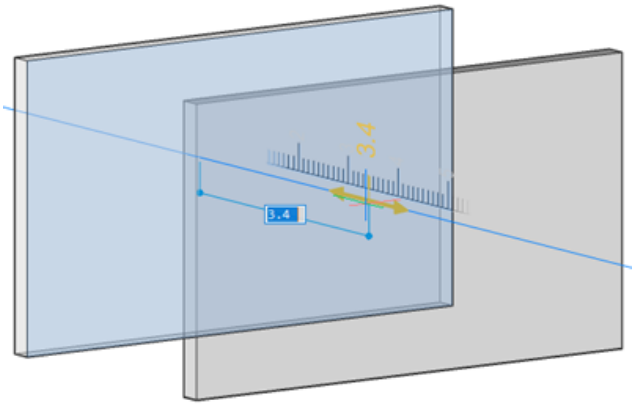
The **Drag** tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, the **Drag** tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids 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).

20.3.22.3 Procedure: using drag on a 3D solid

1 Highlight/select any faces of the solid and select **Drag** from the **Quad** > **Model** tab.

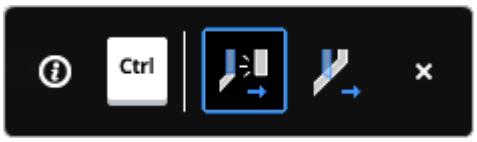


2 Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter.



Nota: 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.

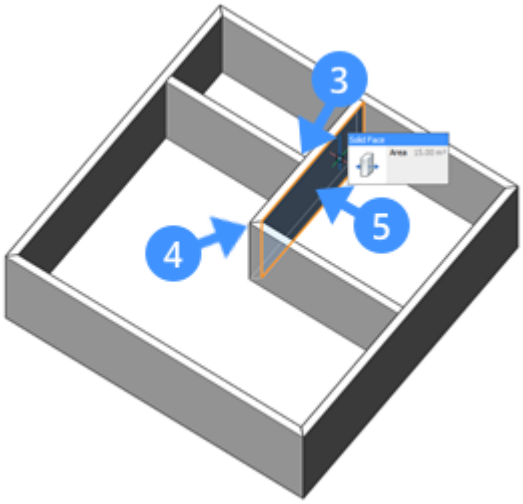


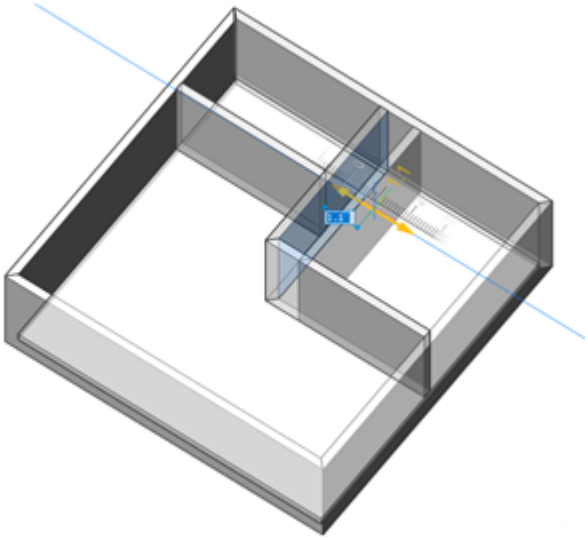
Nota: The current mode is indicated by a blue frame.

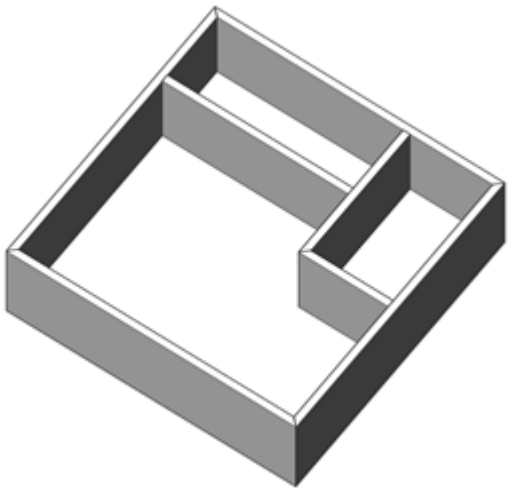
Hit the CTRL-key to toggle the connectivity mode. Alternatively, type D+Enter to disable connectivity or type E+Enter to enable connectivity.

- Select the **Enable connectivity mode** option, to move the solid while preserving the connections between solids.

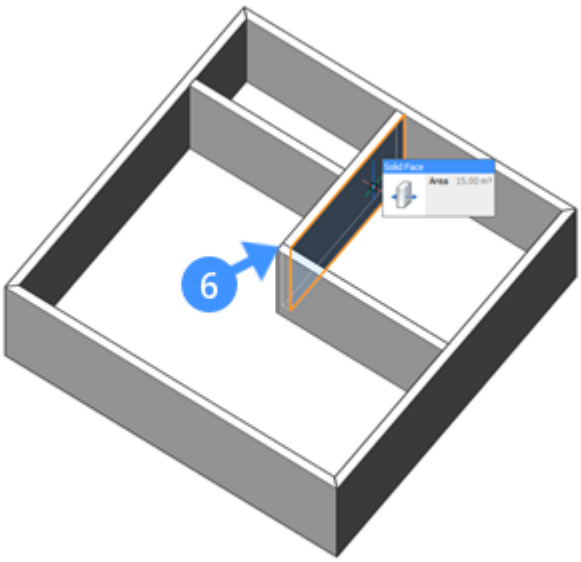
In the following illustration, the solid behavior is shown before and after applying the DRAG in **Enable connectivity mode**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5).

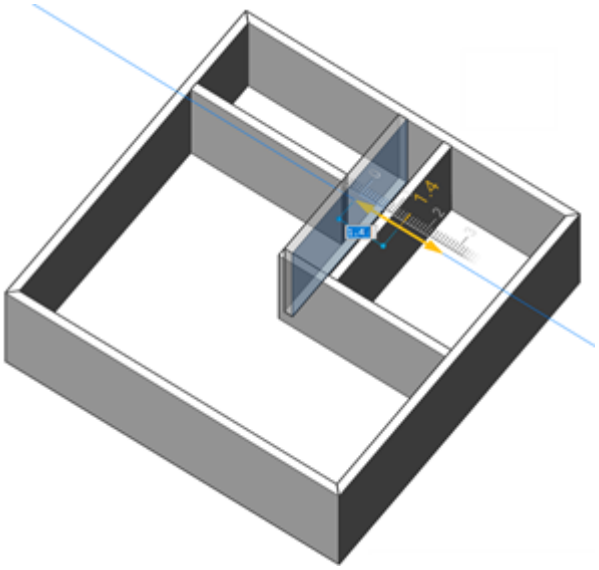


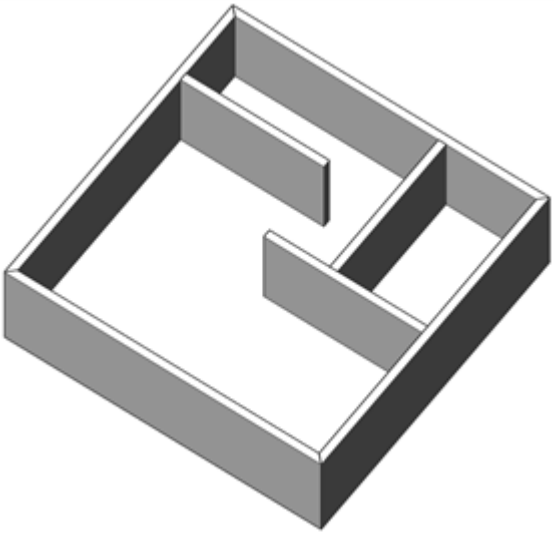




- Select the Disable connectivity mode (default) option to move the solid while breaking the connections between solids.
- In the following illustration, the solid behavior is shown before and after applying the Drag in "Disable connectivity mode". The connectivity between walls, i.e. T- (3) and L- (6) connections are disjointed after dragging an inner wall surface (5).







20.3.23 Using parametrize

20.3.23.1 Commands

PARAMETRIZE, PARAMETERSPANELOPEN

20.3.23.2 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.

20.3.23.3 Procedure: parametrizing a component automatically

1 Open the geometric model that represents your component.

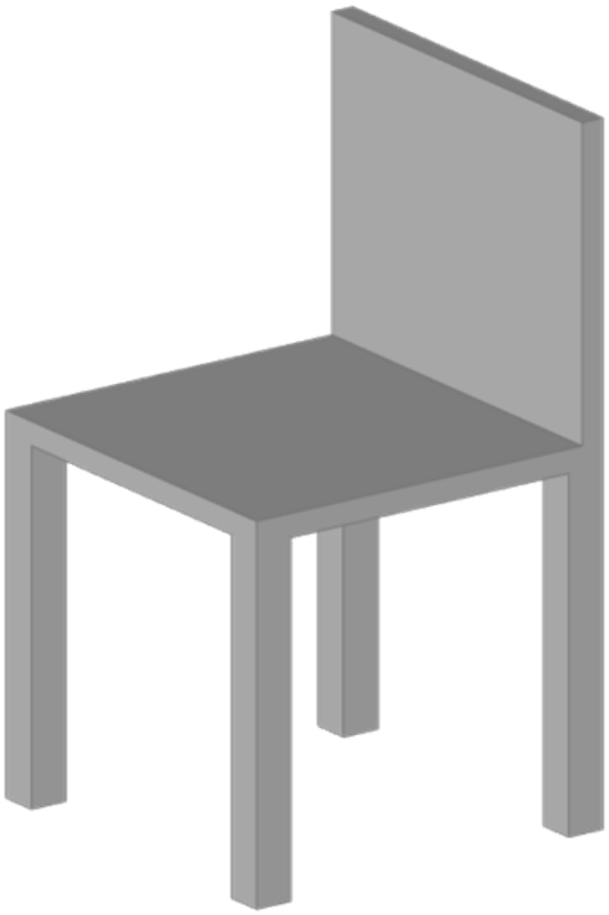
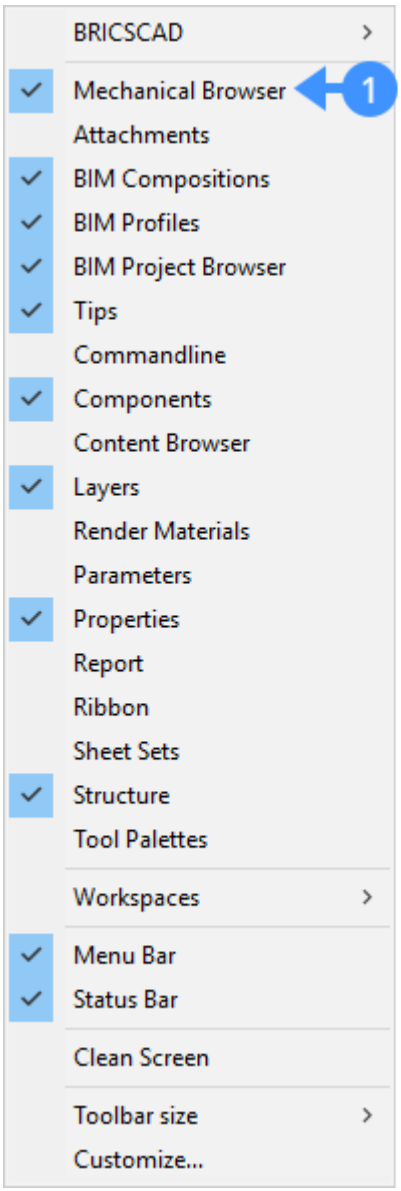


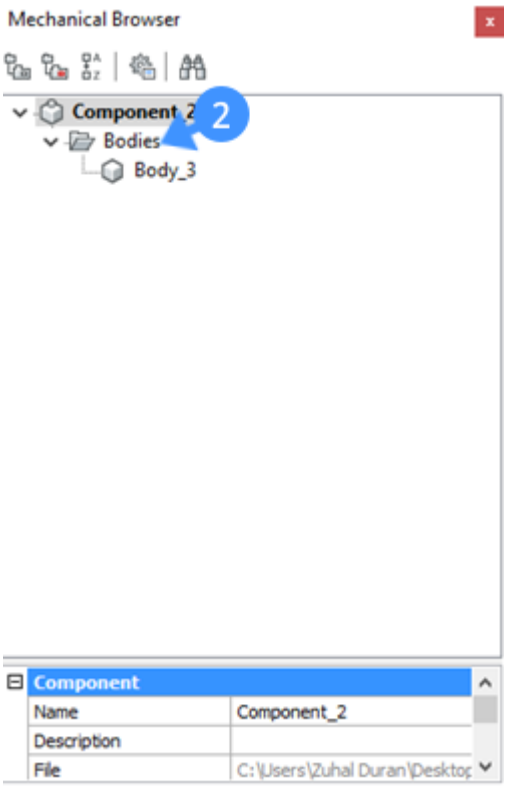
Figure 1, a geometric model of the sample chair.

2 Open the **Mechanical Browser** by clicking its icon in the **Toolbar** panel or type the MECHANICALBROWSEROPEN in the Command line.

Nota: If the Browser or Parameters icons are not displayed in the **Toolbar** panel, place the cursor on a toolbar, then right-click to open **Panels** and choose **Mechanical Browser** (1) in the context menu.



If the model does not contain any parameters and constraint it will display only the 3D solids under the **Bodies** (2) section. If the model contains any constraint or parameter, the parameterized tool asks whether you want to reuse the existing ones.



3 Select the component geometry you want to parametrize.

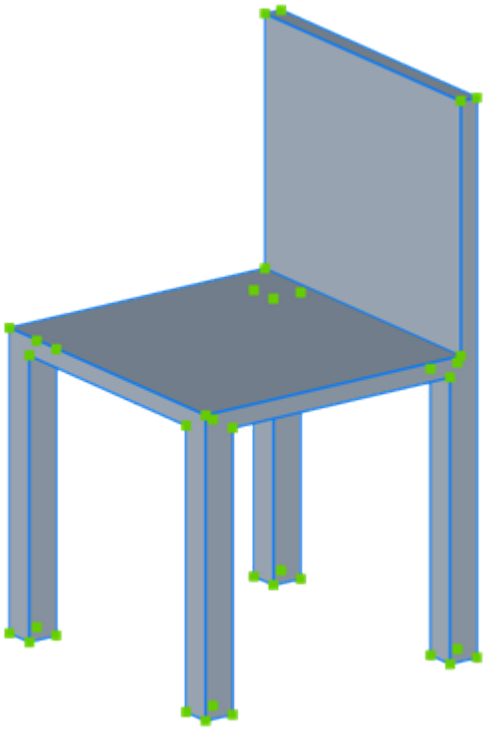


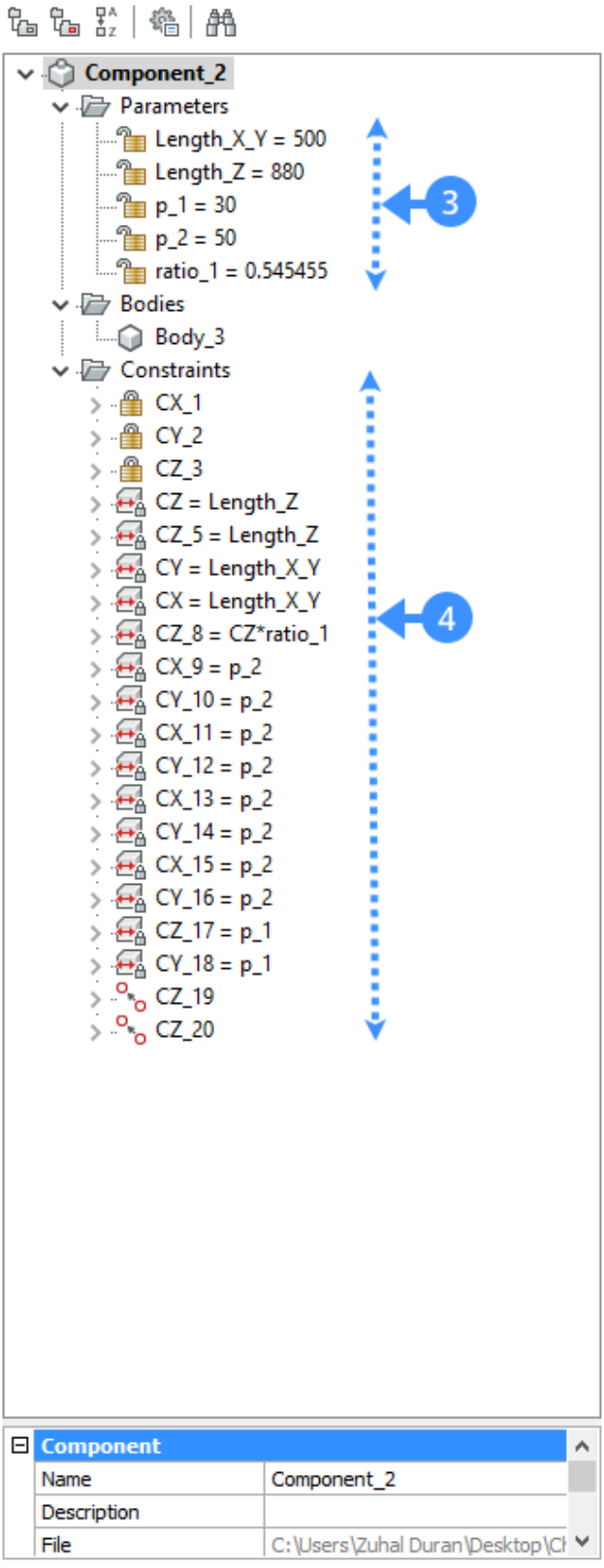
Figure 2, selected object.

4 Launch the **Parametrize** tool in the **Constraints** tab from the Quad.

Or type PARAMETRIZE in the Command line.

A set of parameters and constraints are defined automatically.

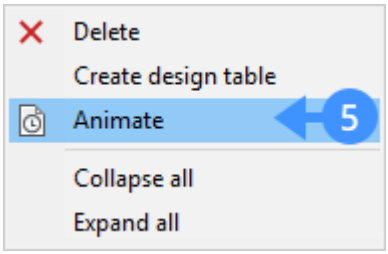
These parameters and constraints are shown in the **Mechanical Browser** (3 and 4).


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
4
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
Component
Name
Component_2
Description
File
C:\Users\Zuhal Duran\Desktop\Cl

Nota: **Parametrize** creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints depending on the geometric model.

Nota: Perpendicularity and tangency can be set (if they are applicable) by changing the DMRECOGNIZE system variable in the **Settings** dialog box.

5 To check easily which parameter corresponds to what, right-click one of the parameters, and choose **Animate** (5) in the context menu.



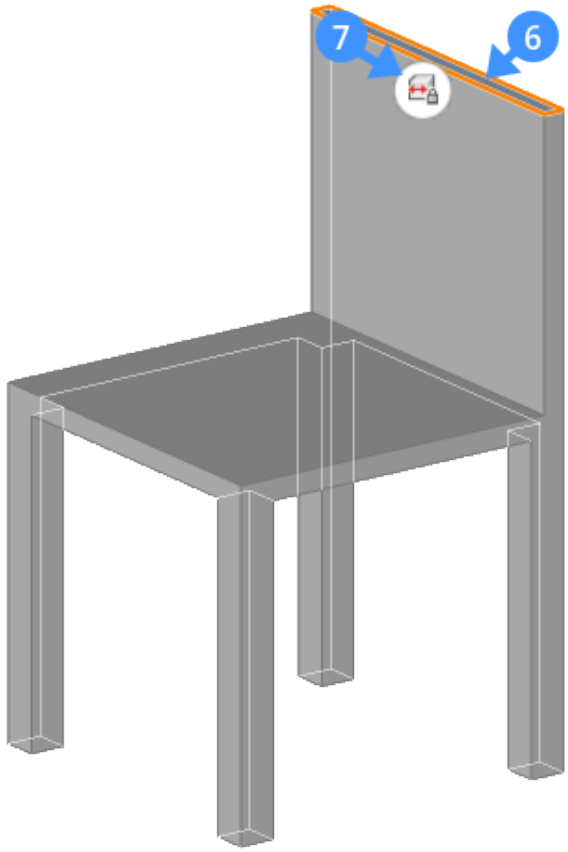
Nota: In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the **Mechanical Browser**. Each parameter corresponds to one measurement in the chair geometry. For instance, **Length_X_Y** controls the width and depth, **Length_Z** controls 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.

20.3.23.4 Procedure: changing the parameters and constraints in the parameters manager

Nota: After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

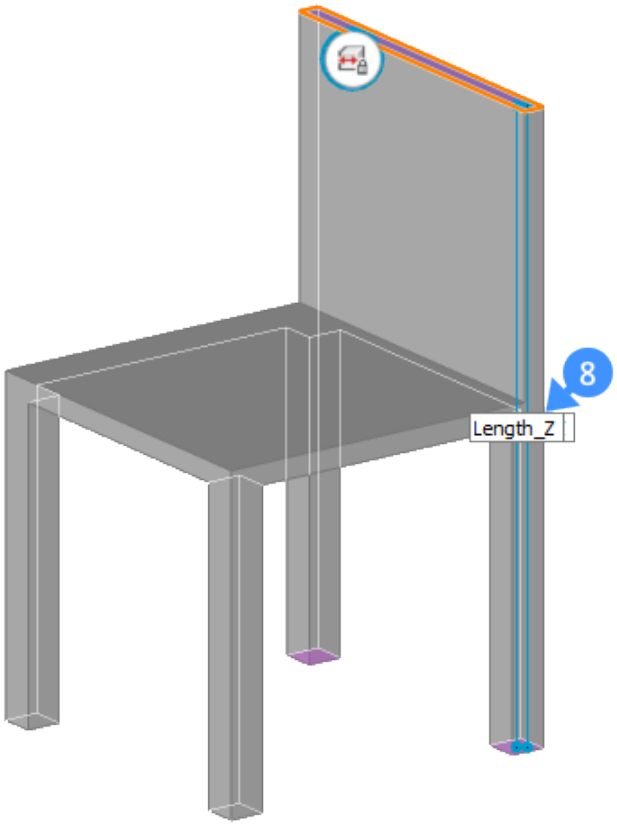
1 Select one of the faces (6) of your component. The constraint type (e.g. distance, fix) regarding the

face is displayed.

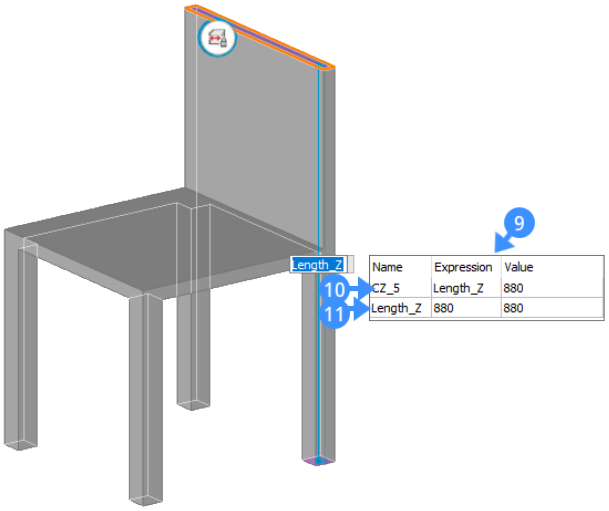


2 Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

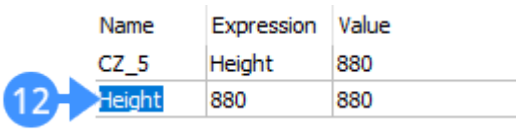
3 Click the editable dimension box (8).



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.



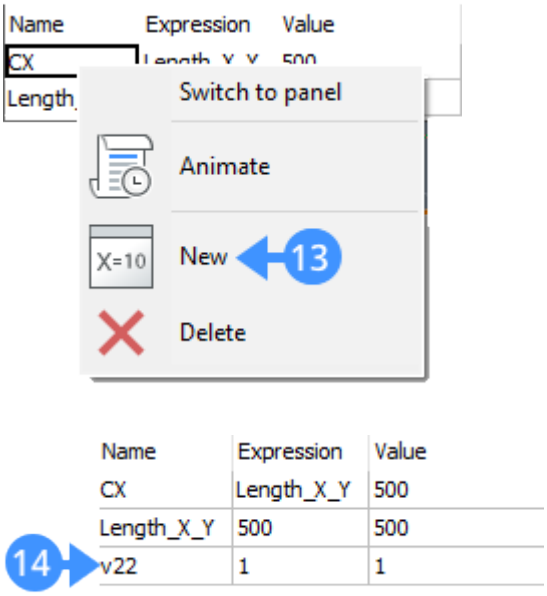
4 Change the parameter name by clicking its field (12) in the parameters manager.



5 (**Optional**) A new parameter can be defined using the **New** option in the parameter manager's context menu.

6 To do so, right-click the parameters manager and select **New** (13).

7 A new parameter is added to the list (14).



Name	Expression	Value
CX	Length_X_Y	500
Length_X_Y	500	500
v22	1	1

14

8 Specify a **Name** (15) for the newly created parameter and enter an expression using the **Expression** field (16).

Name	Expression	Value
CX	Length_X_Y	500
Length_X_Y	500	500
Width	500	500

15 16

9 To link the newly created parameter with an existing constraint, change the expression field of the constraint with the new parameter name (17).

Name	Expressi	Value
CX	Width	500
Length_X_Y	500	500
Width	500	500

17

Nota: The above procedure can also be done using the **Mechanical Browser**.

20.3.24 Working with parameters and constraints

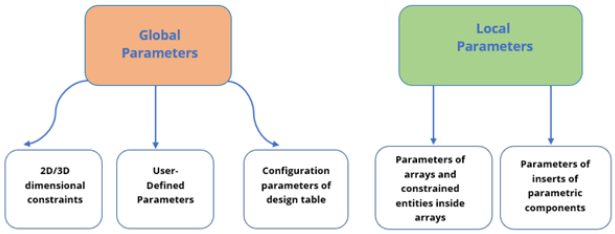
20.3.24.1 Parametric modeling

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.

Nota: 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.



20.3.24.2 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.

20.3.24.3 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.

20.3.24.4 Dimensional constraints

Dimensional constraints allow you to control the sizes of 3D entities in drawings and the distances between them.



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.

20.3.24.5 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)
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).

20.3.24.6 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.

20.3.24.7 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.

Nota: The **Mechanical Browser** allows to navigate through all the constraints and parameters 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

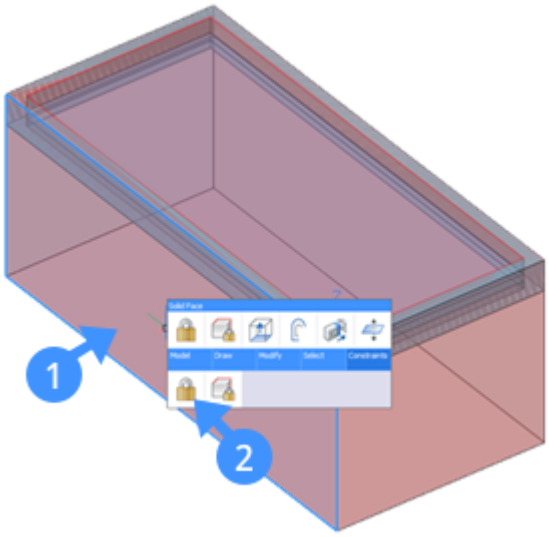
Command: DMFIX3D

1 Turn on the **BIM_Subtract** layer.

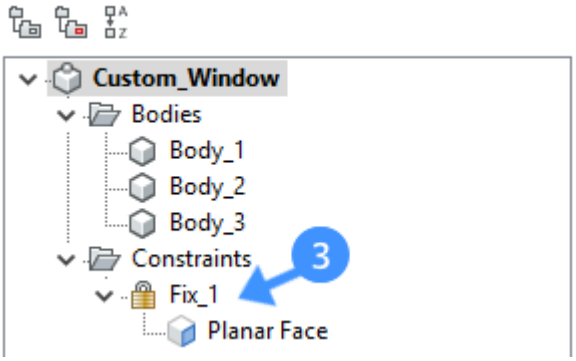
2 Select the entities (1) you want to add a fixation to.

3 (**Optional**) Press the TAB key to select obscured geometry.

4 Choose **Add Fixation** (2) in the 3D constraints command group in the Quad.



The fixation appears in **Mechanical Browser** as **Fix_1**.



Step 3: Applying rigid constraints

Command: DMRIGIDSET3D

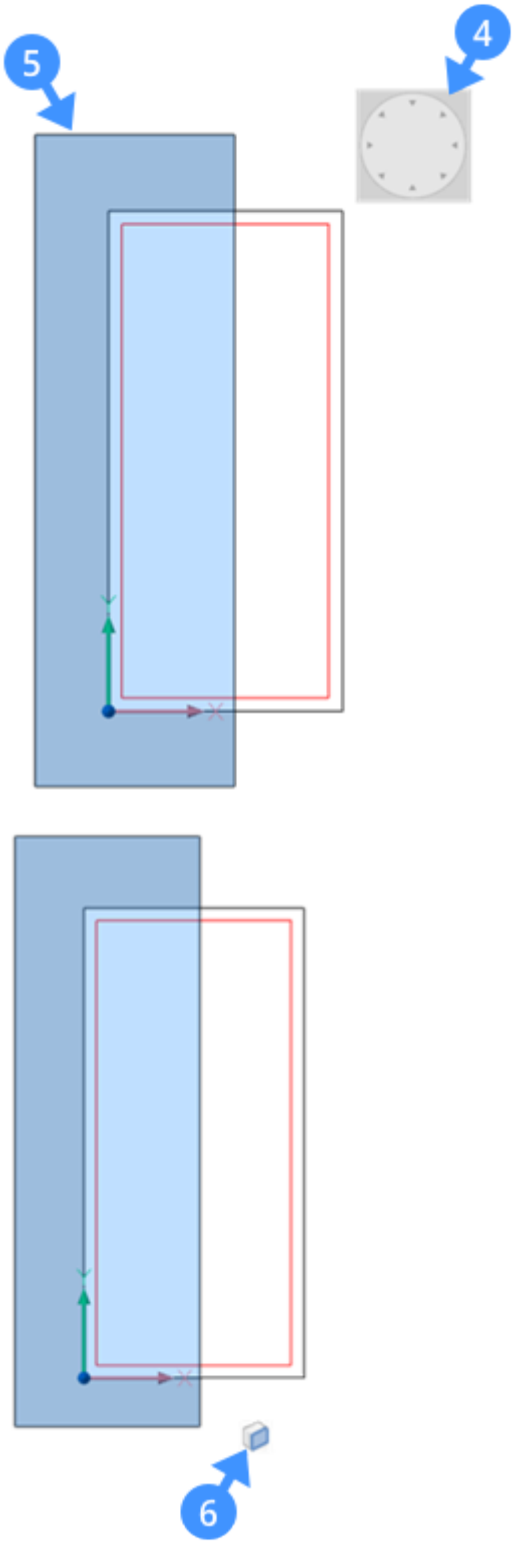
1 Select the entities to apply the **Rigid Set** constraints.

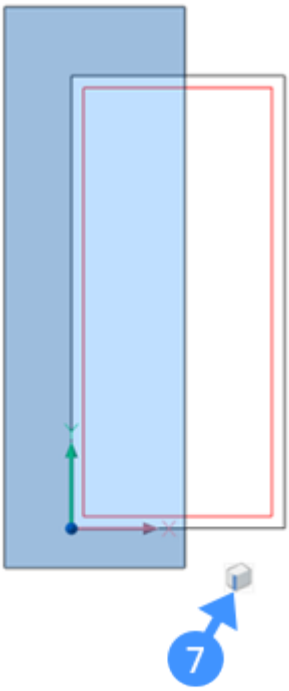
The entities remain in the same position with respect to each other when they are in a Rigid Set.

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).

Nota: By default, the selection boxes see and select solids.


5
4
6



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.

Step 4: Applying distance constraints

Command: DMDISTANCE3D

Dimensions of the window and the thickness of the glass pane.

1 Turn off the **BIM_Subtract** layer.

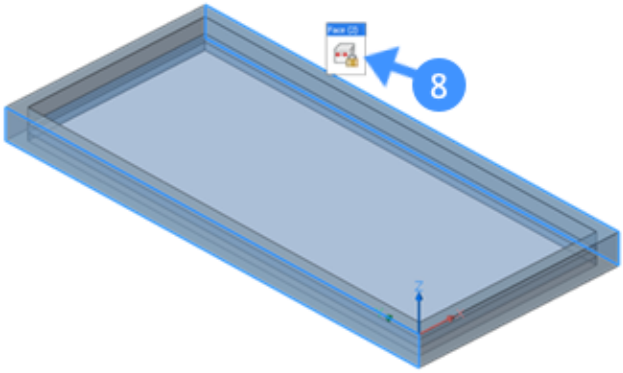
2 Select the first face.

3 The face highlights.

4 Hover over the second face.

5 Hit the TAB key to highlight an obscured face.

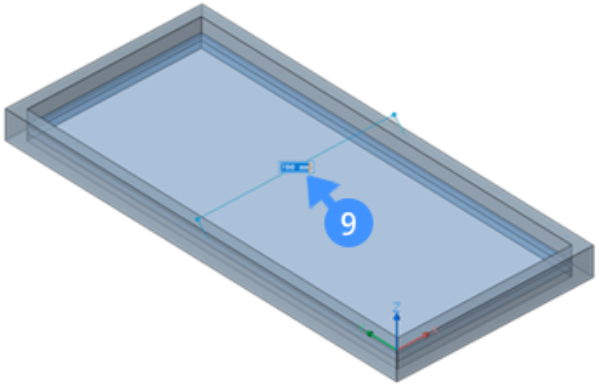
6 When the second face highlights, choose **Add Distance** (8) in the 3D Constraints command group in the Quad.



You are prompted: Specify distance value or [Geometry-driven] <xxx.xx>:

7 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.



- Type a value in the Command line.

8 The distance value between the two surfaces appears in the **Mechanical Browser**.

Nota: 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.

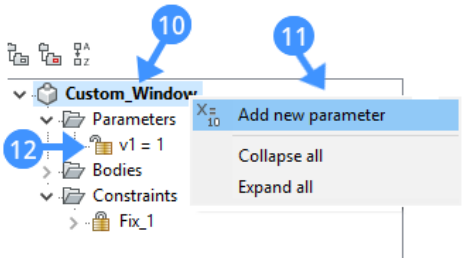
Step 5: Applying parameters

We will create a parameter that controls the width of the window.

1 Right-click the file name (10) in the **Mechanical Browser**.

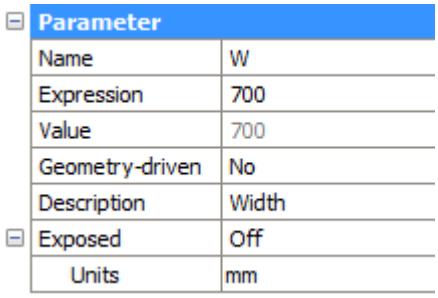
2 Select **Add New Parameter** (11) in the context menu.

The parameter settings grid is created and the parameter is added in **Mechanical Browser** as **v1 = 1** (12).



3 Select the parameter, then edit its properties in the settings grid.

4 The following illustration shows the properties of a **Width** parameter of the custom window. The table below demonstrates each property.



Name	Type a parameter name.
Expression	Type a value or formula.
Value	Shows the current value of the parameter.
Geometry-driven	If yes, makes the parameter geometry-driven.
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.

5 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** field in the settings grid of the constraint.

Constraint	
Type	Distance
Name	Distance_1
Expression	W
Value	700 mm
Enabled	Yes
Exposed	Off
Directions	Same

6 **(Optional)** 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.

20.4 Construindo dados

20.4.1 Analytical model

20.4.1.1 Commands

BIMANALYTICALMODEL

20.4.1.2 About the BIM analytical model

An Analytical Model is in essence a simplified wireframe model that only contains the load-carrying building elements. The linear building elements like columns, beams and members are represented by lines. The planar building elements like walls and slabs are not yet supported in the analytical model.

In BricsCAD, the BIMANALYTICALMODEL command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal but allows you to further adapt the model to your desire. The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. The number of such changes can be controlled by the deviation settings.

You can modify the generated proposal afterward by using the options when you select an element (nodes or axes). To propagate the effect of a local change further you can use recalculate. This will launch a global solution that again tries to minimize rigid links but will respect the changes you made.

After editing the model, you can export it to an IFC file and a CIS/2 file. This analytical model can then be used for further structural analysis after import in a dedicated analysis software package.

It's important to emphasize that the input model has to be fully classified since the BIMANALYTICALMODEL command assumes that all the load-carrying elements are indicated in the building element properties.

In the procedures, the analytical model is explained with an example model. You can find this model in the appendix.

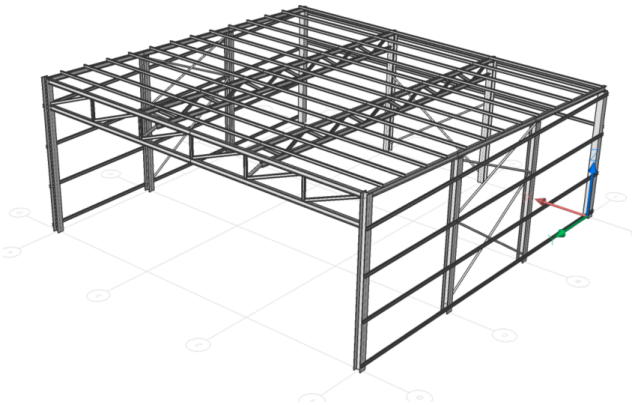
20.4.1.3 Procedure: creating an analytical model with strict deviation settings

1 First, you have to classify your BIM model. This is necessary for using the BIMANALYTICALMODEL command.

1 Use the BIMIFY command on your model or classify elements manually.

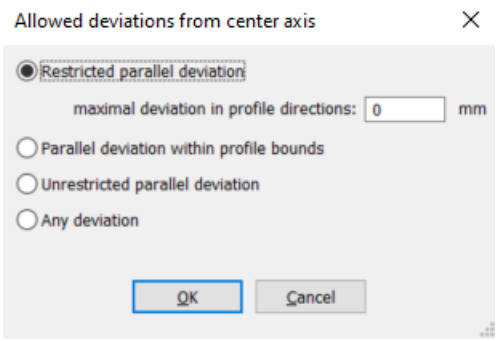
You are prompted: *run Bimify on entire model or [Advance] <model>*

2 Press Enter if you want to classify everything in your model or type **A** if you want a selection of elements.



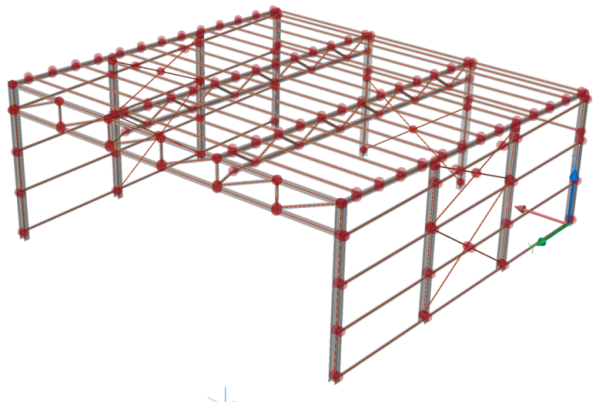
3 Launch the BIMANALYTICALMODEL command in the Command line.

4 A pop-up window opens.

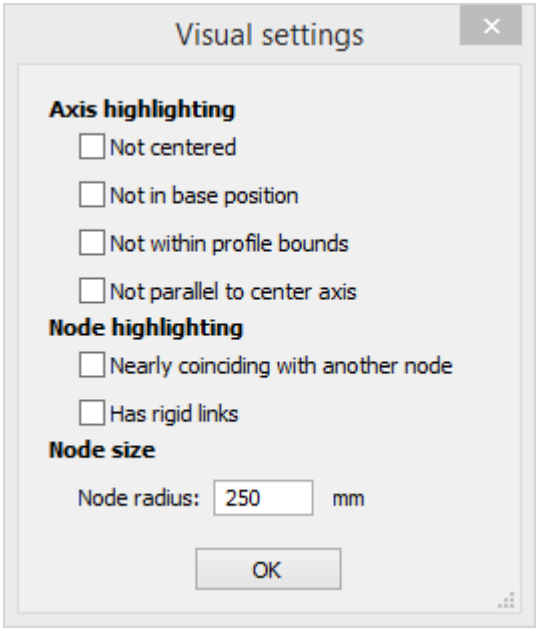


This window allows you to set the level of freedom used when generating an analytical model. When hovering over one of the options, a tooltip shows you what type of restrictions are imposed for that option. In this case, choose the **Restricted parallel deviation** with a maximal deviation in profile directions of 0 mm. No axis is allowed to deviate from its central position in the profile.

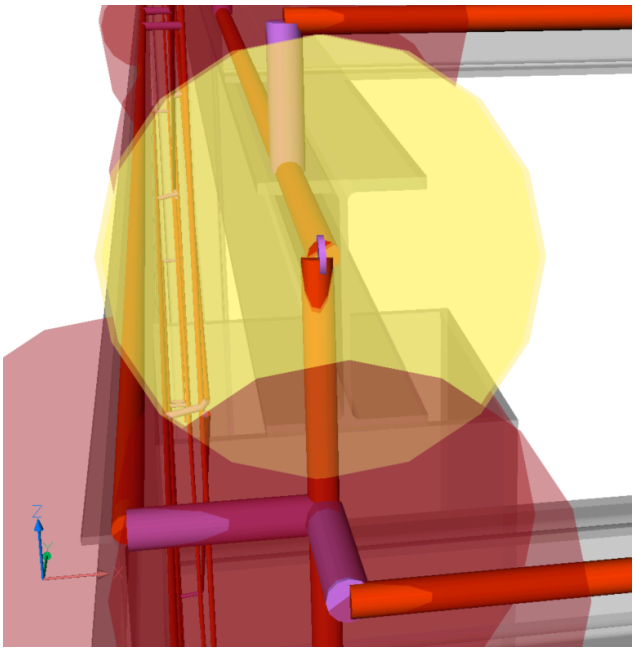
5 An interactive preview of the analytical model is constructed based on the chosen global deviation setting.



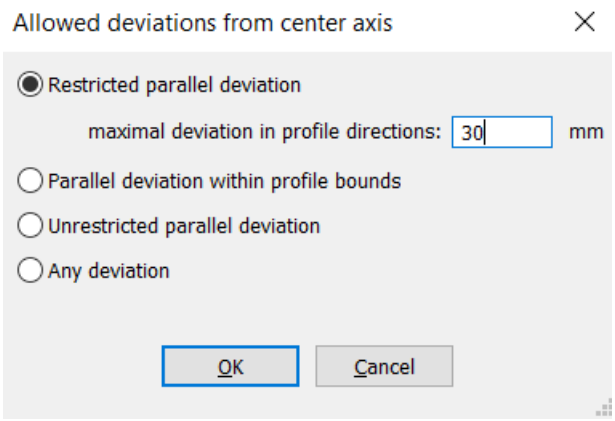
Also, the visual settings window appears. In this window, you can check and uncheck different features of axes and nodes to highlight (in yellow) the axes and nodes which correspond to those features. This allows you to explore the current properties of the axis and nodes of the analytical model. Based on what you find you can determine if any adjustments need to be made. You can also change the size of the disks representing the nodes. This is useful in combination with the node highlighting, which can be used for example to find nearly coinciding nodes.



6 Select the checkbox **Nearly coinciding with another node** in the **visual settings** window. You will see that the center axes at these nodes do not align perfectly. As a result, small rigid links (eccentricities) had to be introduced. This is because of the previous setting in step 4. These small rigid links are displayed in purple.

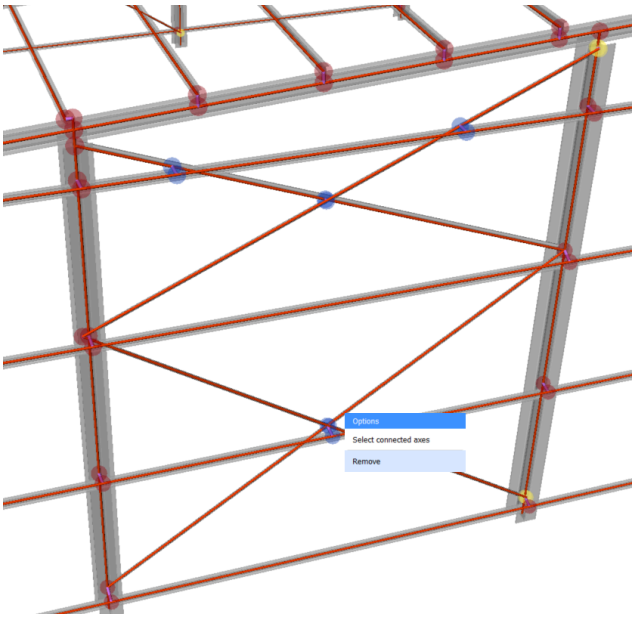


To avoid this, restart the command with a slightly more flexible setting. In the **Allowed deviations from center** window, choose again for **restricted parallel deviation** but this time with a maximal deviation in profile directions of 30 mm.

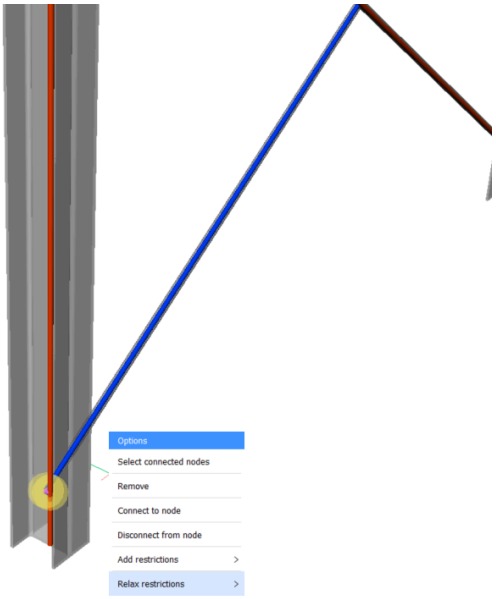


7 For the detection of nodes in the preview of the analytical model, the proximity of structural elements is used. Because of this, it is possible that there arise nodes in the model where tie rods cross. You can easily remove these nodes by clicking on the node and select **Remove**.

Nota: You can select several nodes at once using box selection.

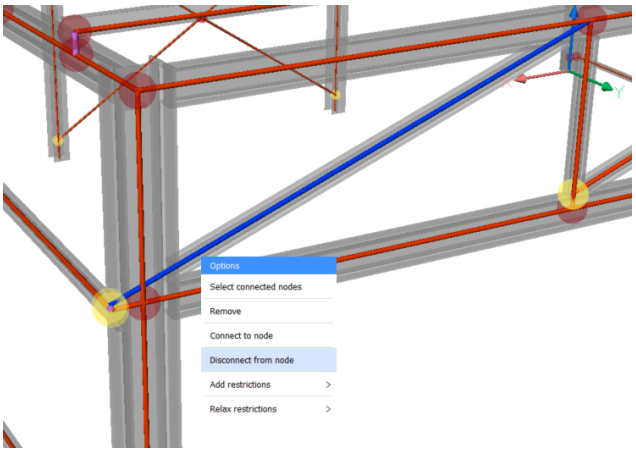


8 If you take a look at the remaining coinciding nodes, you see that the tie rods are not aligned perfectly with the center lines of the columns and small rigid links are formed. You can remove these rigid links by relaxing the position of the axes of the tie rods to a non-parallel position. You can do this by clicking on the tie rod, click **Relax restriction** and click **Non-parallel**. The extra freedom will be used to remove the rigid links by moving the axes of the tie rods a bit.



9 The only coinciding nodes that are left are at the front truss of the hangar structure. If you take a look at the side of the front truss, you see that a tilted member is connected to the horizontal beam rather than the column.

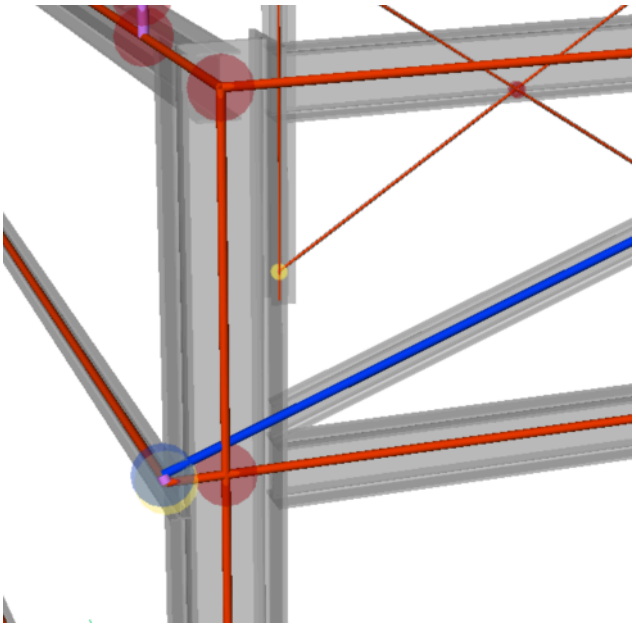
10 You can disconnect the tilted member by clicking on it and choose **Disconnect from node**.



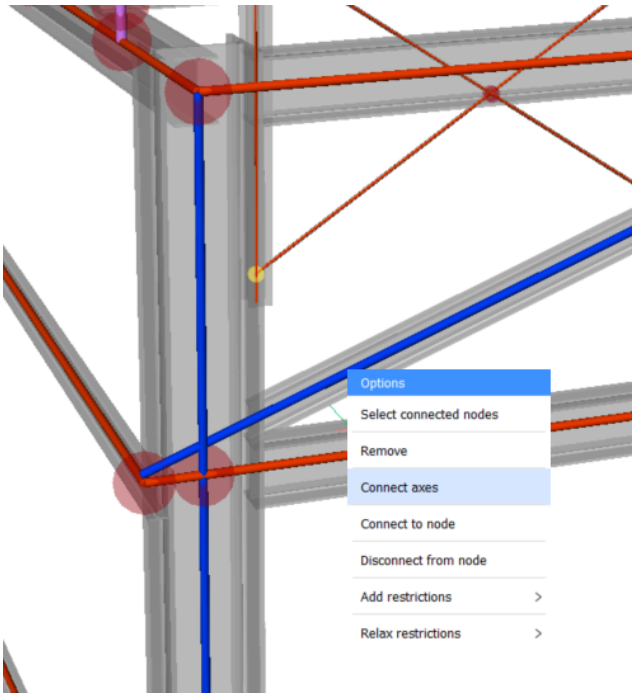
You are prompted:*Select node disconnect from axes [Ok]*

Select the node you want to disconnect from.

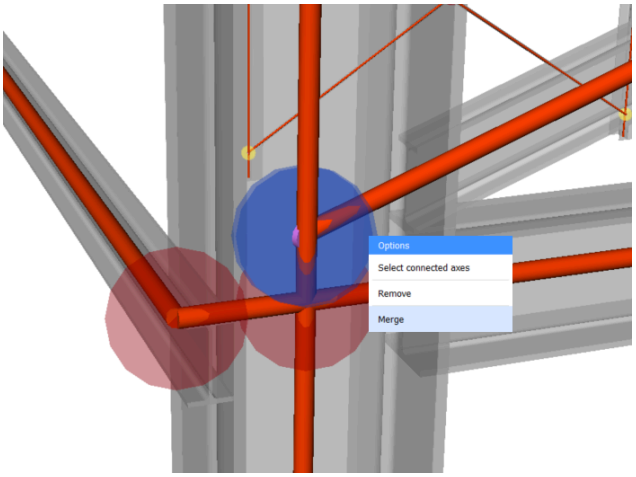
Type **O** to choose Ok.



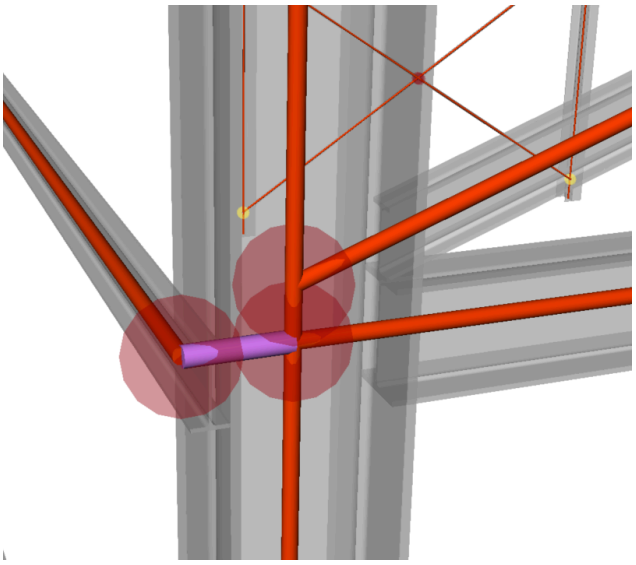
Now select the two axes that you want to connect and click **Connect axes**.



With the global deviation setting, you chose this can only be done by introducing a small rigid link again. You can remove this by merging the two nodes. Select both nodes and click **Merge**.

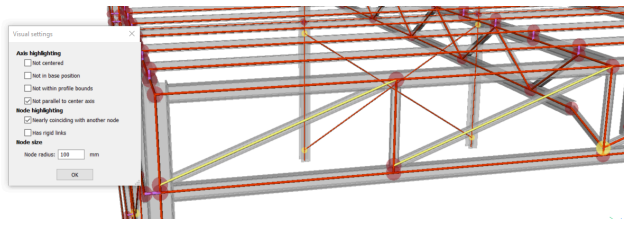


11 You see that the side member also is attached to the front beam instead of the column. Repeat step 9 for the side member.



12 You can remove the last coinciding nodes in the front truss by merging the nodes. This will shift the axes of the tilted members and introduces rigid links on the other end.

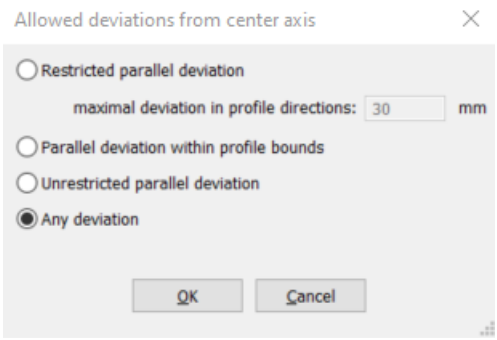
Nota: If you take a look at the axis **Not parallel to center axis** by using the **Visual settings** window, you will see that the tilted member in the front truss has been allowed to be non-parallel to the center axis. So, you could also remove the small rigid links by relaxing the tilted member to a non-parallel position like as we did for the tie rods.



20.4.1.4 Procedure: creating an analytical model with the least restrictive deviation settings

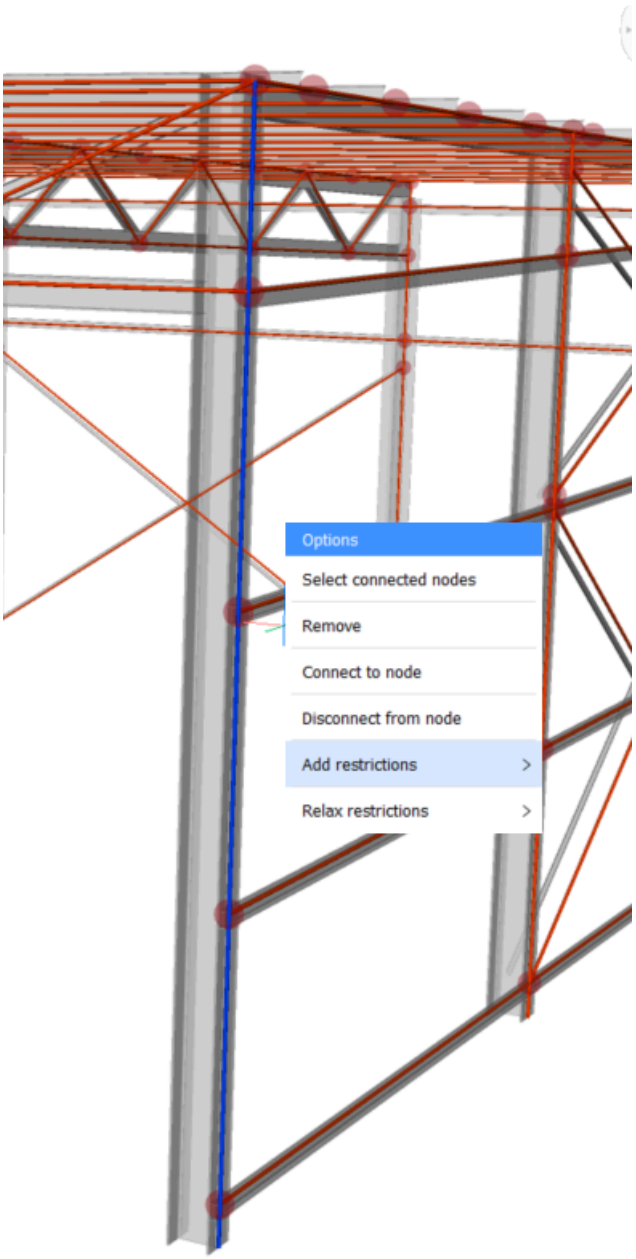
1 Repeat the steps from the previous procedure until step 4.

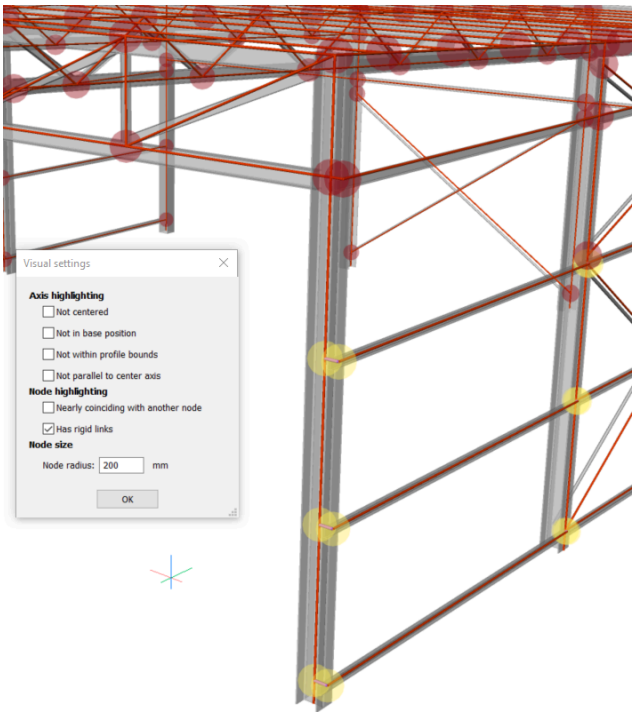
2 Choose **Any deviation** in the **Allowed deviations from center axis**. This allows all kinds of freedom to avoid any rigid links. In the preview of the analytical model, you will see that some axes have been tilted. For example, the small members in the trusses are not centered and the column axis is moved towards the profile bounds to find connections with smaller side members there.



3 Remove the unwanted tie rod nodes as in step 7 of the procedure with a strict deviation setting.

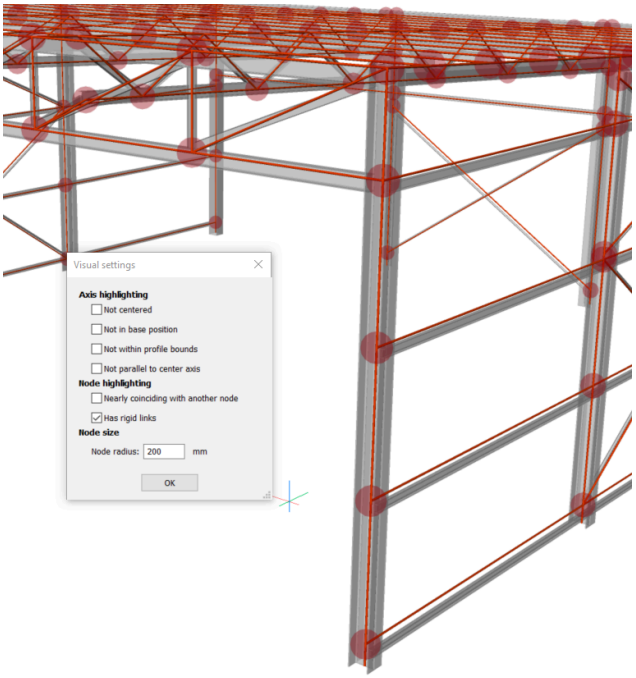
4 In the interactive preview of the analytical model, the column axes were moved towards the profile bounds. In this step, you will center the axis of the columns. You can do this by clicking on the axis of the column, click **Add restrictions** and click **To center**. This introduces rigid links very locally.


Options
Select connected nodes
Remove
Connect to node
Disconnect from node
Add restrictions
Relax restrictions



5 In the visual settings window, select the checkbox **Has rigid links**.

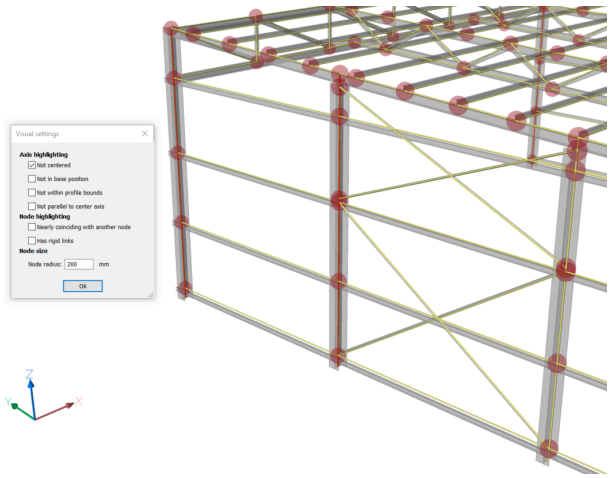
Type **R** in to choose **Recalculate**.



6 In the visual settings window, select the checkbox **Not centered**.

You will see that one of the columns is still not centered because the connection at its top includes not only the larger beams but also one of the smaller ones. You change this by imposing a center

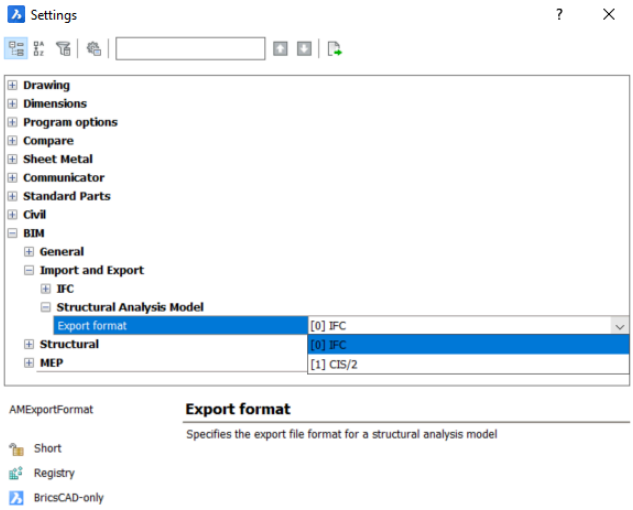
restriction on this column as well.



20.4.1.5 Procedure: exporting an analytical model

1 Before starting the command, open the **Settings** dialog.

2 In the Settings dialog, click the option **Structural Analysis Model** and choose the export format of the analytical model. You can choose between **IFC** and **CIS/2**.



3 Launch the BIMANALYTICALMODEL command.

4 Change the analytical model to your desire.

5 When you have finished the model, type **E** to choose Export and exit.

6 The document window appears where you can name and save the file in the desired folder.

Nota: There are some remarks about the exported result.

- The **node concept** used by BIMANALYTICALMODEL does not completely coincide with **CIS/2** and **IFC** node concept:

  In the BIMANALYTICALMODEL, only connections between linear elements are modeled with a node. For example, columns will not have a floor node if there is no connection to another

element there. During export, missing end nodes are added to be compatible with CIS/2 and IFC export.

In the BIMANALYTICALMODEL, a node always lies exactly on every connected axis. Eccentricities are modeled with explicit visual rigid links (purple bars in preview). During export, each cluster of BricsCAD nodes that are interconnected with rigid links is translated into a single connection node with eccentricities in **CIS/2** and **IFC**.

- The **axis concept** used by BIMANALYTICALMODEL does not always coincide with the **CIS/2** and **IFC** axis concept:

  In the BIMANALYTICALMODEL, there will be one single axis for each physical element. An axis can be attached to more or less than two nodes. During export, an axis that has multiple nodes along its length gets split up into several analysis members, each with one end and one start node, for the **CIS/2** and **IFC** export.
- The concept of **Base Position** in BricsCAD is similar to **cardinal points**. Base positions in BricsCAD form a subset of the cardinal points specified in the **CIS/2** standard. For now, only cardinal positions 1 to 9 are supported.
- In the analysis model, as calculated by BricsCAD, an axis can shift away from the center of the physical entity, essentially to any point within certain bounds. This means that the final position of the analysis axis does not necessarily coincide with a cardinal point or base position of the profile. During export, the section profile is linked to the analysis axis using the closest base position of the profile to the final analysis axis, using the cardinal point concept in **CIS/2** and **IFC**. This means that if you visualize the structure in a **CIS/2** or **IFC** viewer that shows physical members as extrusions of the profile section along the analysis axes, you will see that the structure shown there can slightly deviate from the original structure as modeled or imported into BRICSCAD. The structure as described by the **exported analysis model** is therefore an **approximation of the actual structure**.

model.dwg

20.4.2 AutoComplete BIM data

20.4.2.1 Commands

BIMAUTOMATCH

20.4.2.2 About the AutoComplete BIM data

Instead of assigning compositions to every element manually, BIMAUTOMATCH detects the difference between interior and exterior walls and their respective composition and automatically assigns it to other walls. It can also make distinctions between different floors, different sides of the building, walls with a balcony, etc.



Nota:

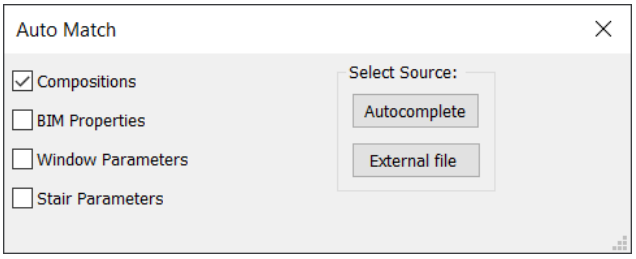
Before you use BIMAUTOMATCH make sure that you have Bimified ( ) everything. AutoMatch works best with a fully Bimified model.

20.4.2.3 AutoComplete BIM data in one file

1 Do one of the following:

- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:



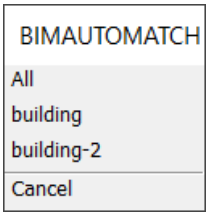
2 Select one of the following:

- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy window parameters.
- **Stair Parameters**: Copy Stair Parameters.

3 Under **Select Source**, choose **Autocomplete**.

4 f the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

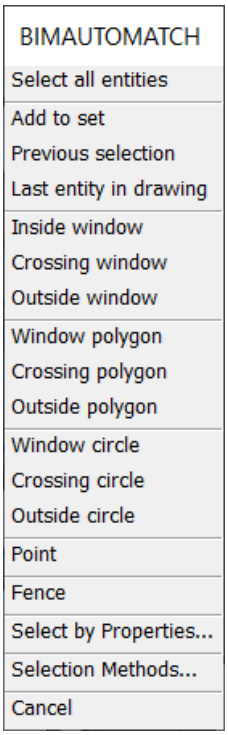
5 **You are prompted:**



Select the building from the menu.

6 If BIMAUTOMATCH finds a logical distinguishing feature between objects it will automatically use this. When it can not, BIMAUTOMATCH asks you to define the source composition or window.

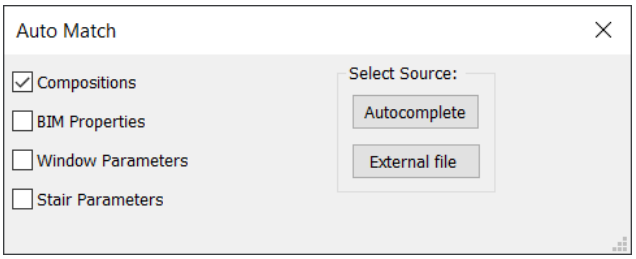
7 **You are prompted:**



20.4.2.4 AutoComplete BIM data with an external file

1 Do one of the following:
- Select **Automatch** from the **Classify** tab in the Ribbon.
- Select **Model** > **Automatch** in the Quad.
- Type BIMAUTOMATCH and press Enter.

You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:



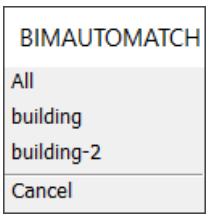
2 Select one of the following:
- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy parameters of windows.
- **Stair Parameters**: Copy properties of stairs.

3 Under **Select Source**, choose **External file**.

4 Select source file and click **Open**.

5 If the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

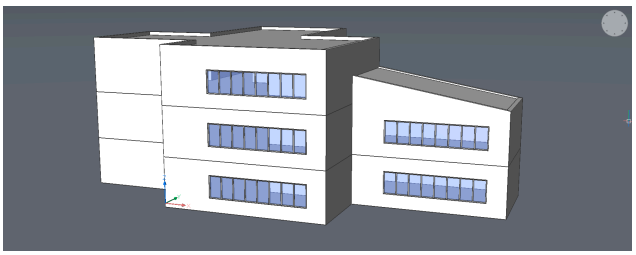
6 **You are prompted:**



7 Select the building from the menu.

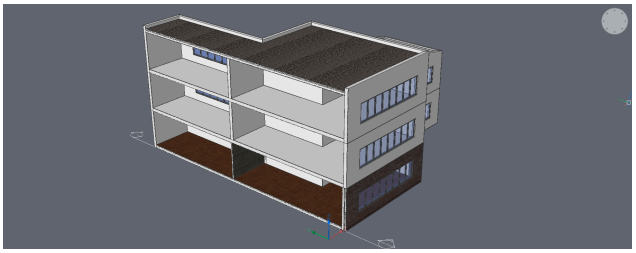
20.4.2.5 Procedure: using autocomplete in one file

1 Open up a file to start from. Here we use a basic rectangular building that looks like this:



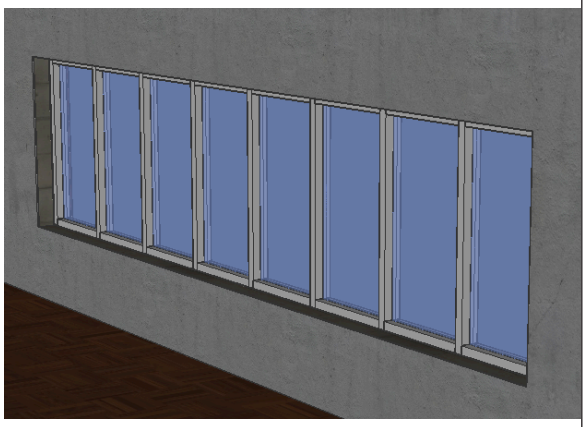
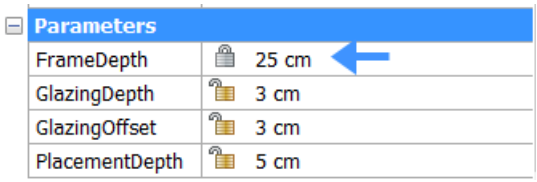
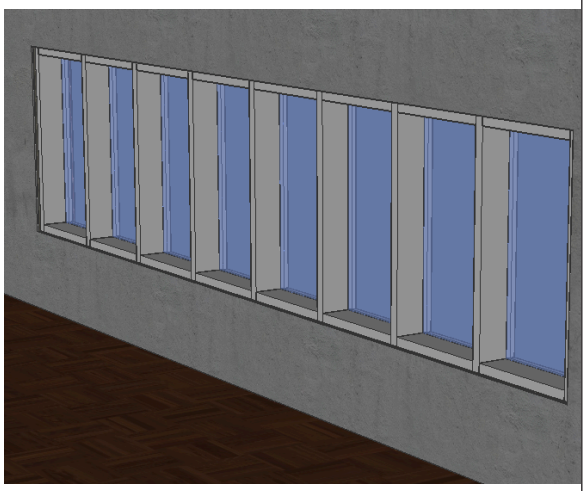
Nota: No BIM data or compositions should be added yet.

2 Add the compositions that you want. Don't do this for all your elements, but only for one element of each type. In our example we give a composition to 1 outer wall, 1 inner wall, 1 floor slab and the roof:



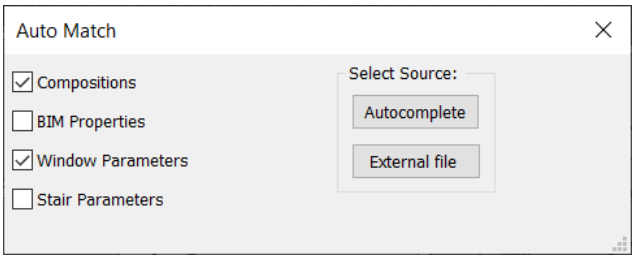
Nota: It could be that you have different types of outer walls, for example the wall facing north that needs more insulation. In this case, you give one of these northern walls a different composition.

3 In this example, we also want to adjust the parameters of our windows. Select only 1 of the windows and change the parameter you want in the properties panel.

	The window before changes.
	Changing the **FrameDepth** in the property panel of the window.
	The window after the changes.

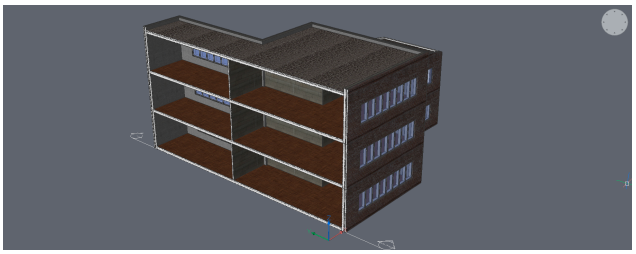
4 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

The following box appears:



We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

5 Select **Autocomplete**. The compositions and window parameters within your file are automatically copied. The drawing looks like this:

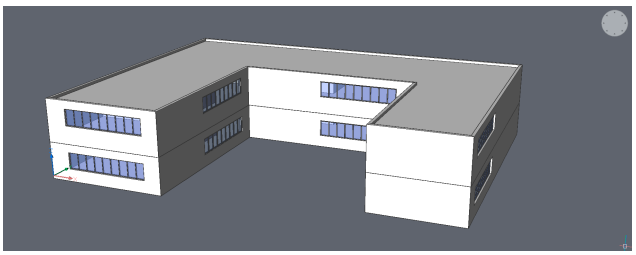


By using BIMAUTOMATCH, we very quickly have all our compositions and window parameters as we want them, without having to worry about forgetting anything.

Nota: You can see that different types of outer walls are no problem. All the walls facing north have been given the correct adjusted composition.

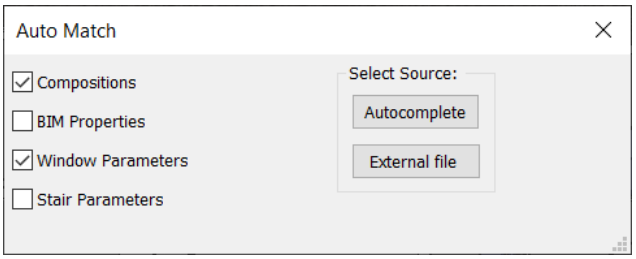
20.4.2.6 Procedure: using autocomplete with an external file

1 Open up a project that is similar in style to another project that you already finished. In this case, we take another rectangular building, similar to the one we looked at before:



2 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

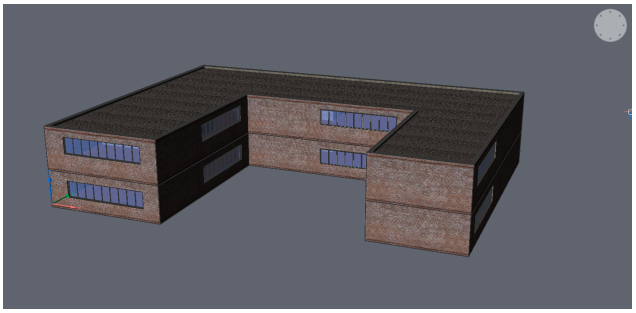
The following window appears:

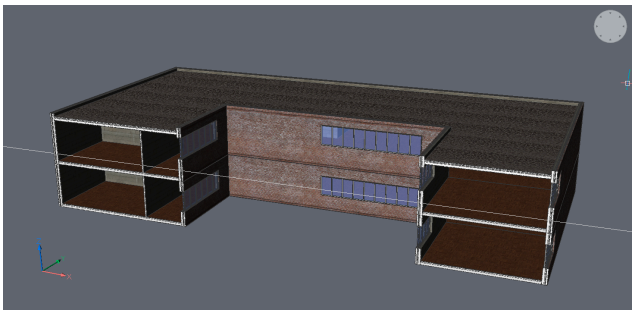


We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

3 Select **External file**.

4 Select the file that we made in the previous procedure. The compositions and window parameters from the external file will be copied to your new file. The drawing looks like this:





Nota: It can be handy to make a library of building styles with their typical compositions, in order to use BIMAUTOMATCH with external files in a very efficient way.

20.4.3 Attaching compositions

20.4.3.1 Commands

BIMATTACHCOMPOSITION, BLCOMPOSITIONS

20.4.3.2 About BIM attach composition

The BIMATTACHCOMPOSITION command allows you to easily apply a composition to a selection of solids. The assigned composition can be flipped or redefined to a different reference face. Optionally, you can align solids based on a reference face or the center of a composition.



See BIMATTACHCOMPOSITION for more information about attaching a composition.
See BIM Compositions for more information about managing compositions.

20.4.3.3 Attaching a composition

To attach a composition, you can:

- Use the BIMATTACHCOMPOSITION command.
- Drag a composition from the **BIM Compositions** panel on the solid(s).

Using the BIMATTACHCOMPOSITION command

1 Launch the BIMATTACHCOMPOSITION command.

1 **You are prompted:** Enter the composition name or [Dialog/Entity] <Dialog>.

2 Do one of the following:

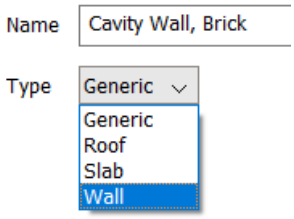
- Type a composition name in the Command line and press Enter.

  Nota: Composition names are case sensitive.

- Press Enter to display the **Compositions** dialog and click on a Composition in the **In Project** or **In Library** list to open the menu of the composition.

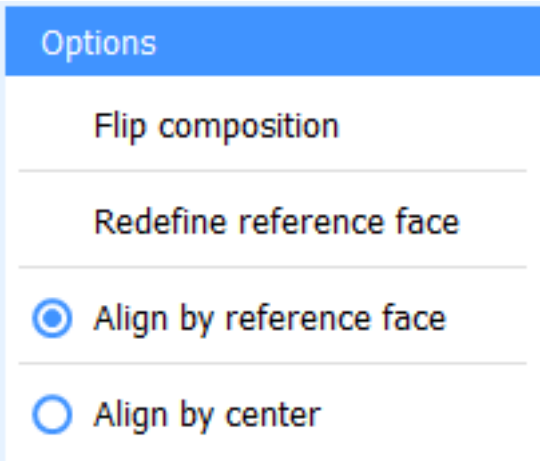
  If you want to select the composition without opening the menu, double-click it.

3 Choose a Composition **Type** from the drop-down menu and press **Select** to accept.



4 Select entities to attach the composition to and press Enter.

5 Choose the reference face. If you hover over the widget, a submenu displays showing different options. For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.



6 Press Enter to accept the reference face.

7 BricsCAD responds: The composition has been assigned to <xxx> element(s).

Nota: Using Bimify building elements are automatically classified. This makes it easy to select the right entities.

Dragging a composition on a solid

1 Select the composition in the BIM Compositions panel ( ). See Dockable panels.

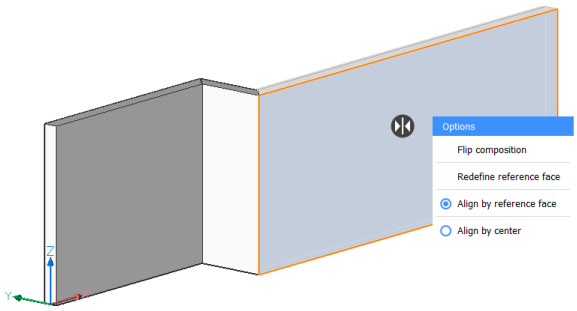
2 Hold down the left mouse button and drag the composition on the solid.

3 The solid face under the cursor highlights, indicating that this face is the reference face to attach the composition.

4 Release the left mouse button to attach the composition.

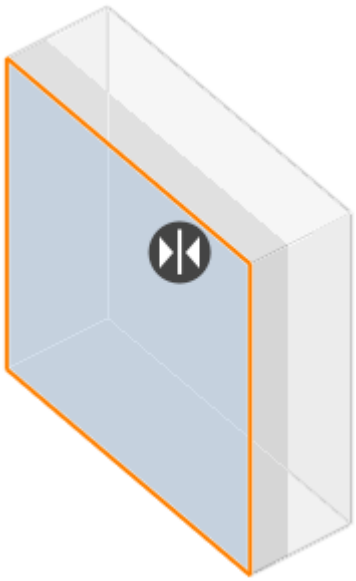
5 A widget appears that allows you to choose the reference face.

6 The reference face is fixed, while the opposite face moves if the thickness of the composition does not fit the current thickness of the solid.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

7 A darker volume indicates the volume and position of the solid after attaching the composition.



8 Press Enter to accept the current reference face and attach the composition.

Nota: When a single ply composition with a variable thickness is selected, it is not needed to select a reference face. The composition is attached to the selected solid.

Dragging a composition on a selection of solids

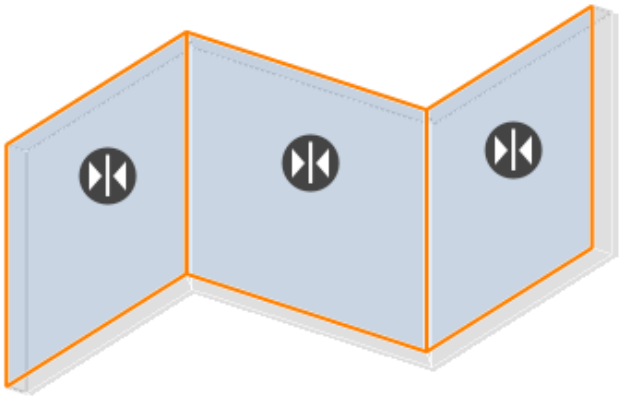
1 Select the solids you want to apply the composition to.

2 Select the composition in the **BIM Compositions** panel.

3 Hold down the left mouse button and drag the composition on any of the selected solids.

4 Release the left mouse button to attach the composition.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

Nota:

- BIM compositions are copied to the Project database upon attachment.
- Attaching a **Wall** composition to a solid, automatically classifies the solid as a wall.
- Attaching a **Slab** or **Roof** composition to a solid, automatically classifies the solid as a slab.

20.4.3.4 Copy compositions from other solids

1 Launch the BIMATTACHCOMPOSITION command.

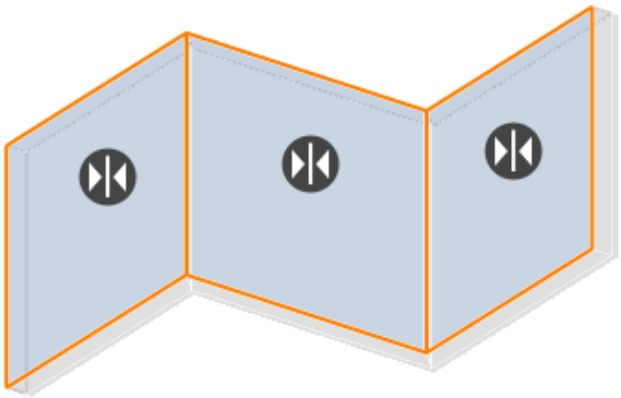
2 Select the **Entity** option.

Prompts you: **Select other entity to get composition:**

3 Select the solid to copy the composition from.

4 Select the solids(s) to attach the copied composition to.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

20.4.3.5 Removing compositions from solids

The **Unclassify** option of the BIMCLASSIFY command removes a composition from a solid.

1 Select the solid(s).

2 Choose **Unclassify** from the **BIM** command group in the Quad.

20.4.4 BIM classify

20.4.4.1 Commands

BIMCLASSIFY

20.4.4.2 About

If you have a conceptual model without any classification, you can turn these elements into classified BIM elements using BIMCLASSIFY.

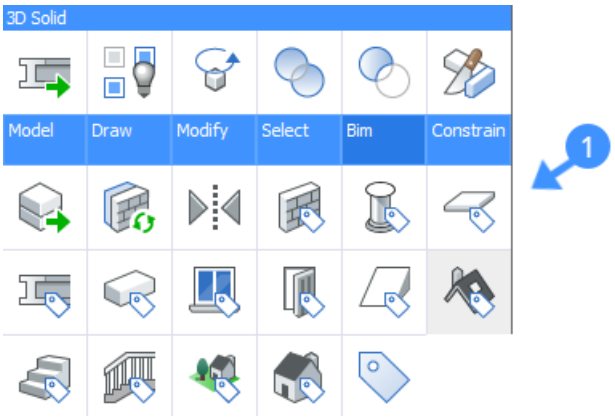


For more information about this command, visit the Command Reference article BIMCLASSIFY.

20.4.4.3 Classifying entities

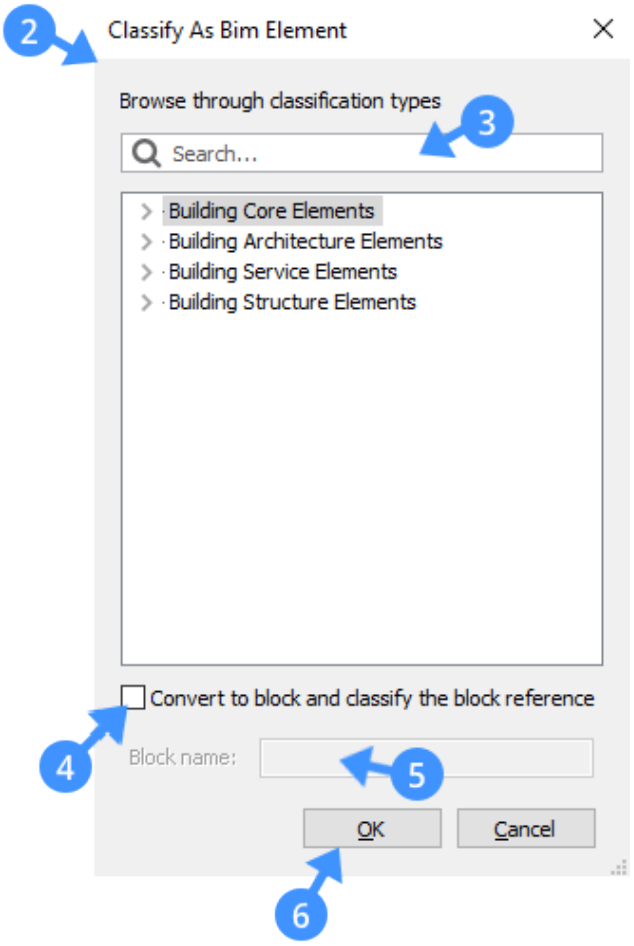
1 Select an entity in the drawing to classify.

2 Choose a classification type (1) in the **BIM** command group in the Quad.



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):



Nota: The classification types are grouped into the following categories:

- **Building Core Elements**
- **Building Architecture Elements**
- **Building Service Elements**
- **Building Structure Elements**
- **Spatial 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.

20.4.4.4 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:

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 transmittance	0
Solar heat gain transmittance	0
Thermal transmittance summer	0 W/m²·K
Thermal transmittance winter	0 W/m²·K

BIM	
Type	Window
Building Element Type	Window
Name	
Description	
Building	
Story	
Space	
Composition	
GUID	0shXs2TdfFB9RulemExNT4
Space 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	
Number	
Entity property sets	

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

Door Window Shading Type	
External shading coefficient	0
Internal shading coefficient	0
Inset shading coefficient	0

20.4.5 BIM compositions

20.4.5.1 Commands

BLCOMPOSITIONS, BIMCLASSIFY, BLMATERIALS, BIMATTACHCOMPOSITION

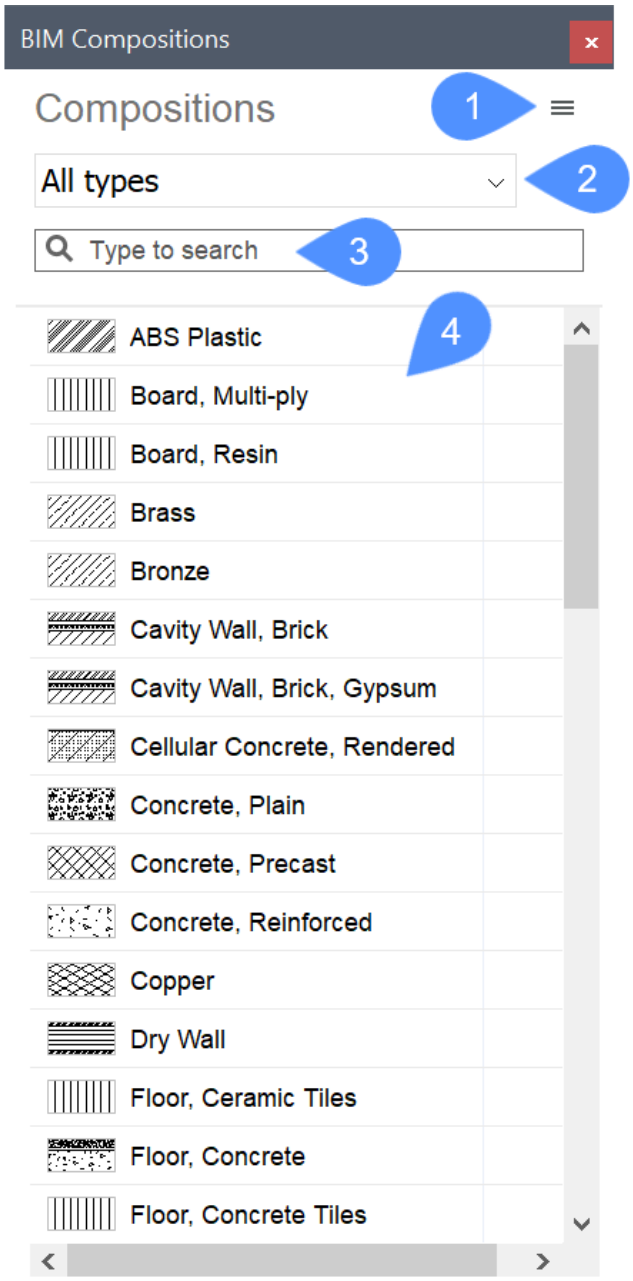
20.4.5.2 About

BIM compositions:

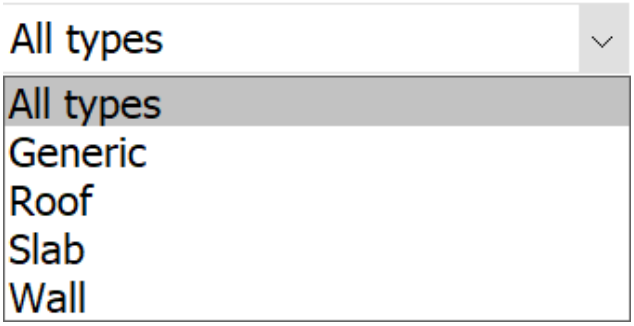
- Are stored in the library database.
- Are either multiply or single-ply.
- Ply materials are managed in the **Physical Materials** editor ( ).
- Can be assigned to any solid in a BIM model.
- Are listed in the **BIM Compositions** panel ( ).
- Are managed in the **Compositions** editor ( ).

For more information about Attaching Compositions, see the procedure article Attaching Compositions.

20.4.5.3 Using the BIM compositions panel



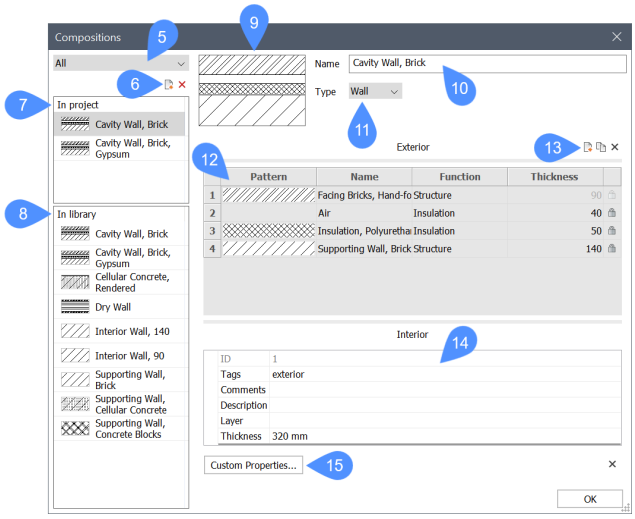
(1) Menu	Open material dialog... Open compositions dialog... Open project dialog... Show only compositions in project...

(2) Composition types list	
(3) Search field	Filters the list of the compositions. Only compositions that contain the current search term are displayed.
(4) Available compositions	All compositions of the selected type in both the **In library** and the **In project** database display, unless the **Show only compositions in project...** menu option is selected.

20.4.5.4 Using the BIM compositions editor

The BIM **Compositions** editor is a dialog box that contains options to create, modify and delete compositions. Using the **Compositions** editor, you can also sort the compositions by category and view the properties of the composition.

The BLCOMPOSITIONS command opens the BIM **Compositions** Editor:



(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 composition plies. The ply on top (exterior) is applied to the reference of the solid.
(13) Add ply	Opens the physical materials editor which allows you to choose a material from the library or project database. Here you can also create a new material. Inserts a copy of the selected ply.
(14) Properties grid	Shows the properties of the selected composition.
(15) Custom properties...	Opens the **custom properties** dialog.

20.4.5.5 Creating a new composition

You can create a new composition in either the **Project** or the **Library** database.

1 To open the BIM **Compositions** editor, do one of the following:
- Select the **Open the compositions dialog...** in the menu on the **Compositions** panel.
- Type BLCOMPOSITIONS in the Command line.

2 Click the **New** icon ( ) on the left of the compositions toolbar.

A new (empty) composition is added to the selected database.

3 To add and edit plies, go to **Edit a composition**.

20.4.5.6 Editing a composition

Step 1: Changing name and type

1 Select the new composition.

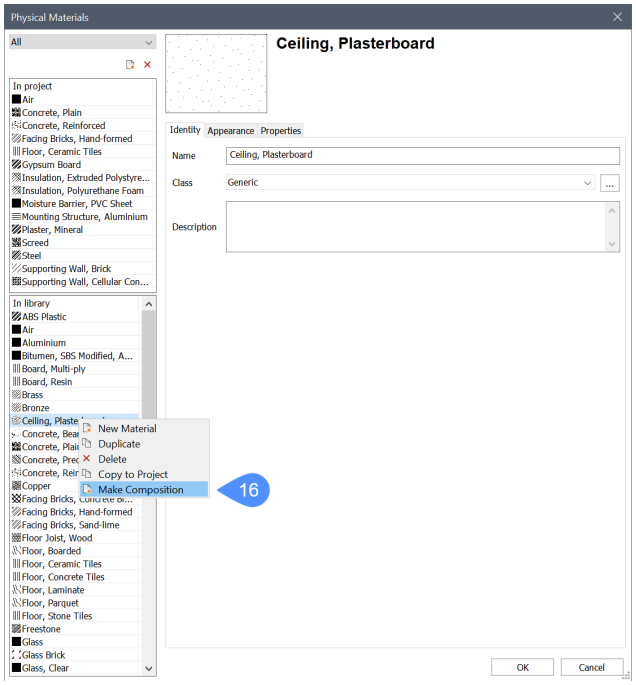
2 Type a new name in the **Name** field for the composition.

3 Use the Type drop-down box to specify the composition type: **Wall**, **Slab**, **Roof** or **Generic**.

Composition categories are used to filter the list of the compositions and the **Compositions** dialog.

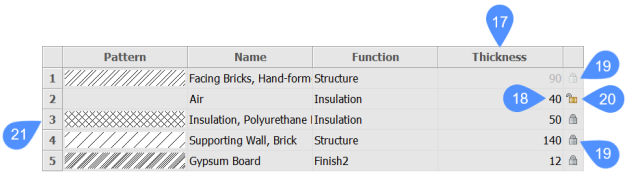
Step 2: Changing the plies

- **Adding, editing and deleting plies**
  a Click the Add Ply button ( ) on the right of the compositions editor.
  The **Physical Materials** dialog displays:



  b Select a material either in the Project **or** the Library database and double-click **or** press the **OK** button to assign the material to ply in the composition.
  c Double-click on the **Pattern** or **Name** of the ply to edit the material. See the **Physical Materials** article for more information.
  d To choose another material, pick a material in the Library **or** Project list and press **OK**.
  e To delete a ply, select the ply and click on the **Delete** tool button.
  f **(Optional)** Right-click the material and choose **Make Composition** (16) in the context menu to create a single-ply composition.

- **Modifying the thickness and changing the ply order**
  a Click the **Thickness** field (17) to modify the thickness of a ply in the Structure grid.
  You can only modify the ply thickness if the **Allow Custom Thickness** property of the material is set to **Yes**.



  b Enter a value in the field (18) and press Enter to accept it. The overall thickness of the composition is updated accordingly.
  c The plies of a composition are ordered from Exterior (top) to Interior (bottom). Drag the ply

number (21) on the desired position to change the ply order.

Nota: When a composition has been modified, the solids that have this composition attached are updated automatically. Use the BIMUPDATETHICKNESS command apply the changes in the model.

- **Locking/unlocking the composition thickness**

  The overall thickness of a composition is equal to the sum of the thicknesses of the composing plies.

  The thickness of a ply can be either locked or unlocked (see image above).

  - Locked thickness (19): the thickness of the ply is fixed.
  - Unlocked thickness (20): the thickness of the ply is variable.

  Nota: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

  - **Fixed:** all plies have locked thicknesses.
  - **Minimal:** the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
  - **Free:** a single ply composition has an unlocked thickness.

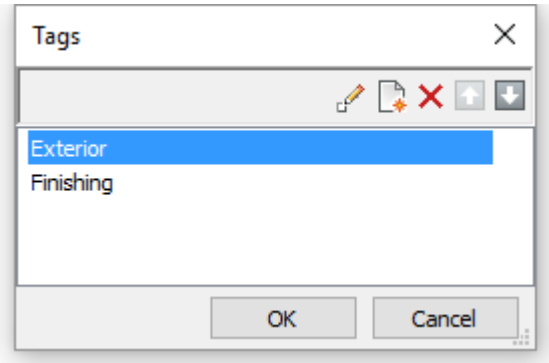
  The thickness of a ply can only be unlocked if the **Allow Custom Thickness** property of the material is set to **Yes**.

- **Adding Tags**

  a Select the **Tags** field.

  b Type the tags, separated by commas. Tags are case sensitive.

  c Click the **Browse** button ( ) next to the tags field in the **Compositions** dialog box to open the **Tags** dialog.



  d Click the **New Item** button ( ) to add a new tag at the bottom of the list.

  e Select a tag in the list and click a tool button:

  -
  -
  -
  -

  f Click the **OK** button to save the changes.

Step 3: Creating new materials

You can create new materials from scratch or copy them from an existing material. See **Physical Materials** for more details.

20.4.5.7 Displaying compositions

Whether the various plies of a composition show in the 3D model is controlled through the LEVELOFDETAIL user preference. The RENDERCOMPOSITIONMATERIAL user preference controls whether the ply material is rendered or not.

Click the icons in the **Home/View** panel in the ribbon or the **BIM** toolbar to toggle the status of the composition display:

- Level of Detail: 
- Render Composition Material: 

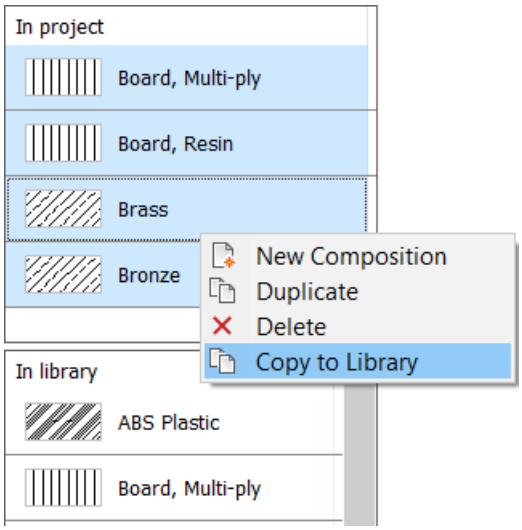
When LEVELOFDETAIL=0, the major faces of a solid are rendered using the material of the outside plies. The minor faces (for example top, bottom, start and end face of a wall) are rendered using the material of the exterior ply.

20.4.5.8 Copying compositions

You can use the following procedure in both the **BIM Compositions** panel and the **Compositions** editor:

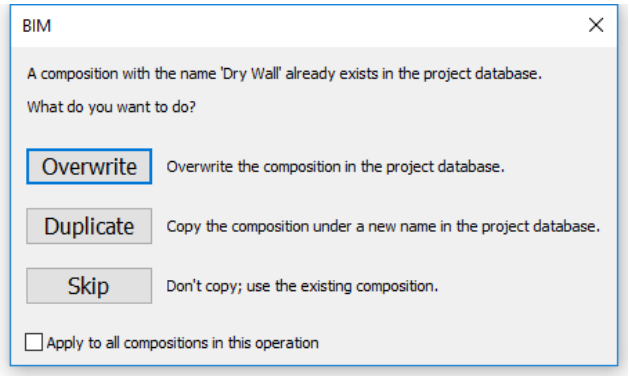
1 Select the composition(s) in the source database. Press and hold the Ctrl key to select multiple compositions.

2 Do one of the following:
   - Drag the selection to the other database.
   - Right-click and choose **Copy to Library** or **Copy to Project** in the context menu.



If the value of the COPYOVERWRITE user preference is 0 (zero), a dialog box displays if the composition already exists in the target library. You can choose to overwrite, duplicate or skip existing compositions.

(Optional) Check the **Apply to all compositions in this operation** option.



20.4.5.9 Deleting compositions

1 Select the compositions in the **In Project** or **In Library** list on the **BIM Compositions** panel 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).

Nota: Compositions that are used in the project cannot be deleted from the project database.

20.4.6 BIM data structure

20.4.6.1 Commands

BIMSPATIALLOCATIONS, BIMATTACHCOMPOSITION, BLCOMPOSITIONS, BIMPROFILES, BLMATERIALS, BIMPROPERTIES, BIMPROJECTINFO

20.4.6.2 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 in the **Roamable** root folder. When a BricsCAD update is installed, the **BricsCAD User File Manager** 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** dialog.

20.4.6.3 Materials and compositions

A BIM database consists of Building Materials 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

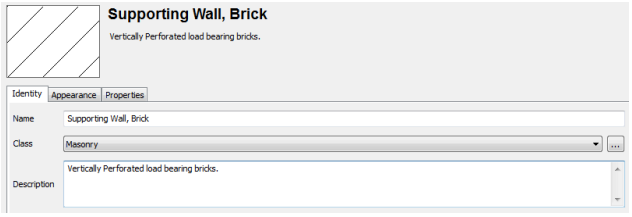
The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic** Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical 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.



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.

20.4.7 BIM ply editing

20.4.7.1 Commands

LEVELOFDETAIL, RENDERCOMPOSITIONMATERIAL, DMPUSHPULL

20.4.7.2 About

The **BIM Ply Editing** function allows you to manipulate all the composition plies in your model. In BricsCAD, each drawing has settings stored inside to control the composition plies, called LOD (LEVELOFDETAIL), and the render materials, called RENDERCOMPOSITIONMATERIAL. These two settings can be turned on/off separately.

20.4.7.3 Composition plies and render materials

Composition plies display only when LEVELOFDETAIL = 2 (high).

Building elements do not need to be selected, they will be automatically adjusted. When LEVELOFDETAIL = 0 (low) the material definition of the solid is used.

To toggle the value of the LEVELOFDETAIL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:



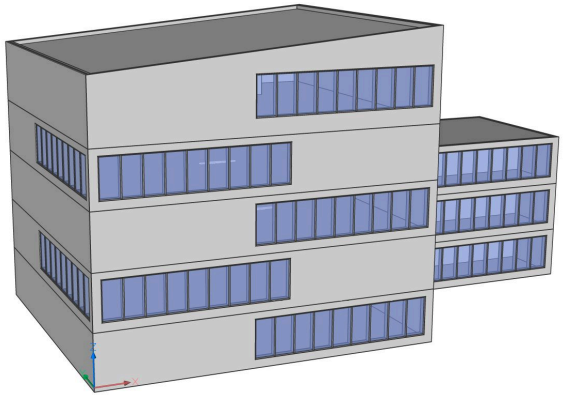
For more information about this setting visit the article LEVELOFDETAIL.

Render materials display only when RENDERCOMPOSITIONMATERIAL = 1 (on). The render material of each ply is defined in the 3D Render Material setting of the material definition.

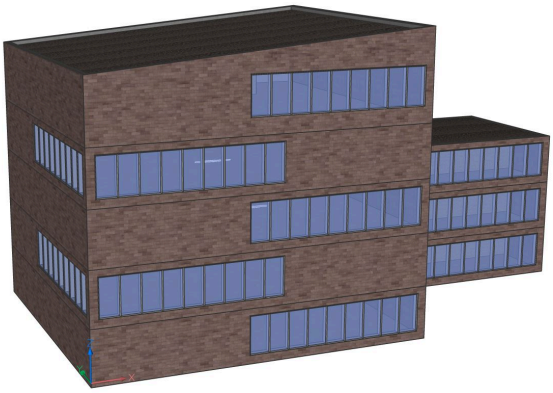
To toggle the value of the RENDERCOMPOSITIONMATERIAL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:



For more information about this command, visit the article RENDERCOMPOSITIONMATERIAL.



RENDERCOMPOSITIONMATERIAL = 0 (off)



RENDERCOMPOSITIONMATERIAL = 1 (on)

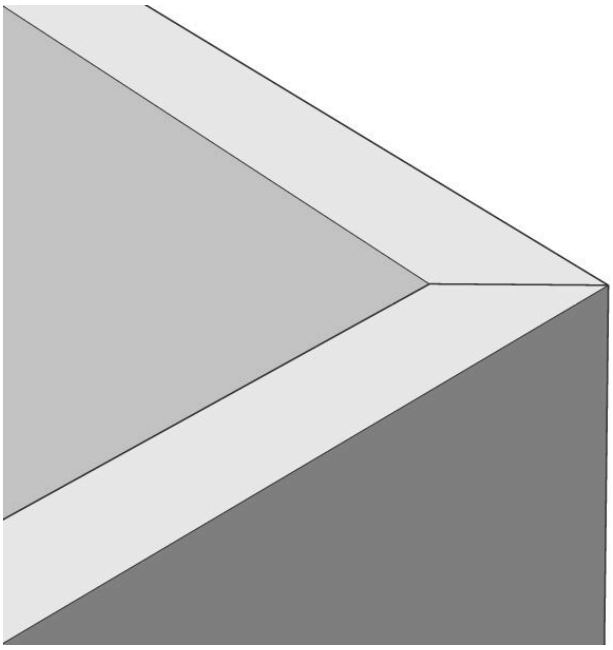
Nota: Only when Visual Styles have their **Face Settings / Material Display settings** set to **Materials and Textures** (e.g. **BIM**, **Modeling**, and **Realistic**), a visual representation of materials is displayed. Composition plies are displayed in the model as well as render materials.

Nota: If the render materials of the selected composition are not downloaded or the composition is incorrect, the entity will be displayed in red.

There are different combinations possible with LOD and RENDERCOMPOSITIONMATERIAL turned on or off.

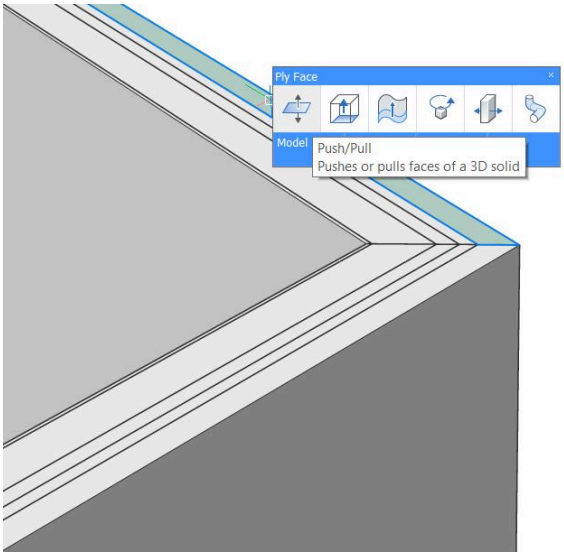
- **LOD =0, RENDERCOMPOSITIONMATERIAL = 0**

  The material definition of the solid is used and no materials are displayed.



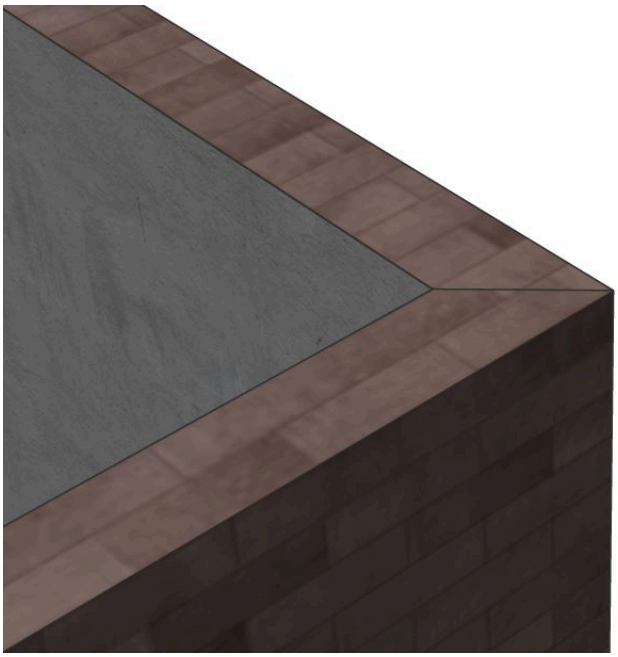
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 0**

  The solid is subdivided into plies, no materials are displayed. When **face detection** is turned on, you can select the faces of different plies to **push/pull** for example.



- **LOD =0, RENDERCOMPOSITIONMATERIAL = 1**

  The material definition of a solid is used. The materials of the exterior and interior face are displayed and the material of the exterior layer will be wrapped around the object.



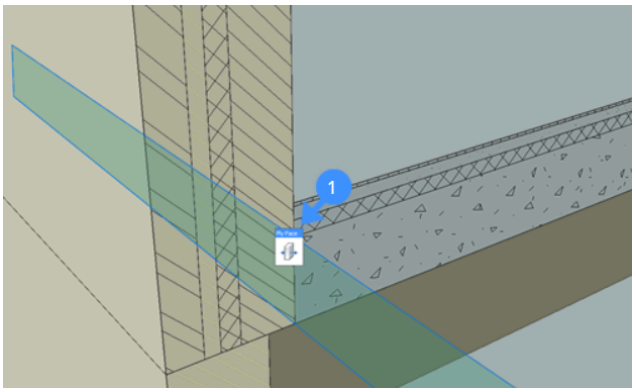
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 1**

  The solid is subdivided into plies. Each ply will be displayed with its own material. When face detection is turned on, you can select the faces of different plies to push/pull for example.

20.4.7.4 Manipulating the composition plies

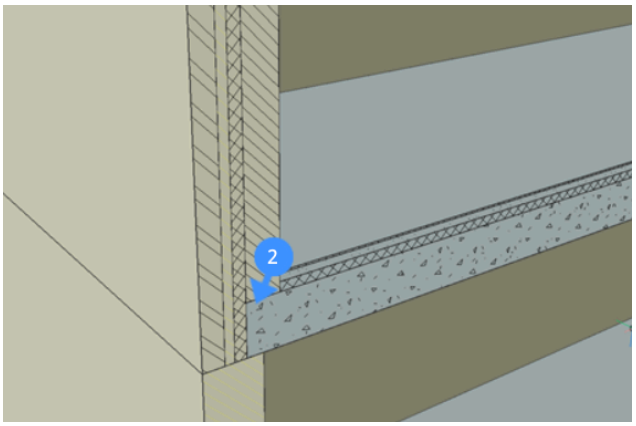
1 Make sure the value of **LEVELOFDETAIL = 2 (High)**.

2 Place your cursor near the composition ply you want to edit and press the Tab key until the **Ply face** (1) highlights.



Nota: The highlighted ply face is displayed in the detected boundary color (default = 95). Face detection must be enabled to do this.

3 Select **Push/Pull** from the Quad and drag the ply to a new position (2).



Nota: Make sure DMPUSHPULLSUBTRACT = 1. This ensures that plies being push/pulled are subtracted from any of the other 3D solids they intersect.

Nota: Make sure ESNAP is turned on to snap to the edge of a ply.

20.4.7.5 Procedure: creating a rooftop detail

It is also possible to create more complex details with extra solids added apart from the existing compositions. As an example, a rooftop detail will be created, starting from a rooftop edge.

1 tart with this roof-wall connection.

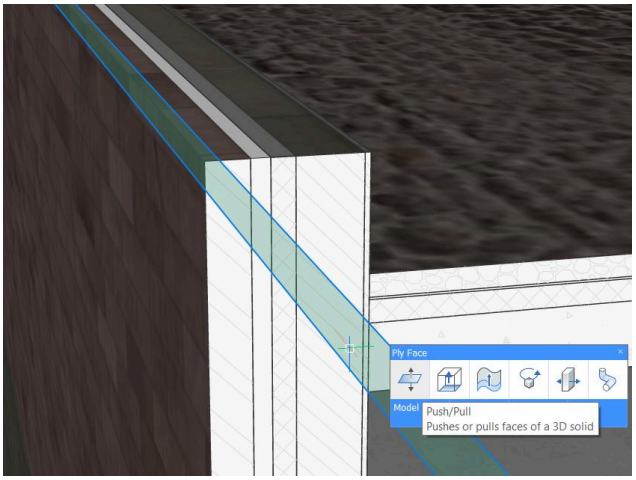


2 Click the **Face detection** icon in the ribbon to enable face detection.

3 Turn on LOD to be able to select ply faces. Click the icon in the ribbon **or** type LEVELOFDETAIL in the Command line, type **2** and press Enter.

4 Hover over the edge of the concrete ply of the floor slab and press Tab to select the face of the ply.

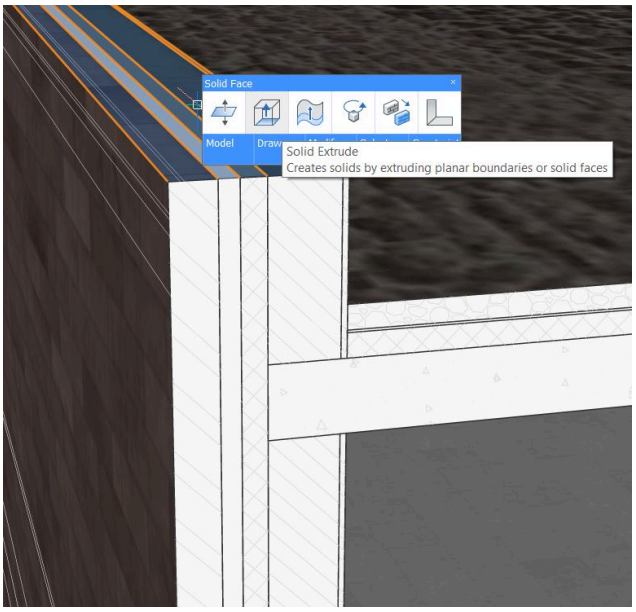
5 Choose **Push/Pull** in the **model** tab of the Quad.



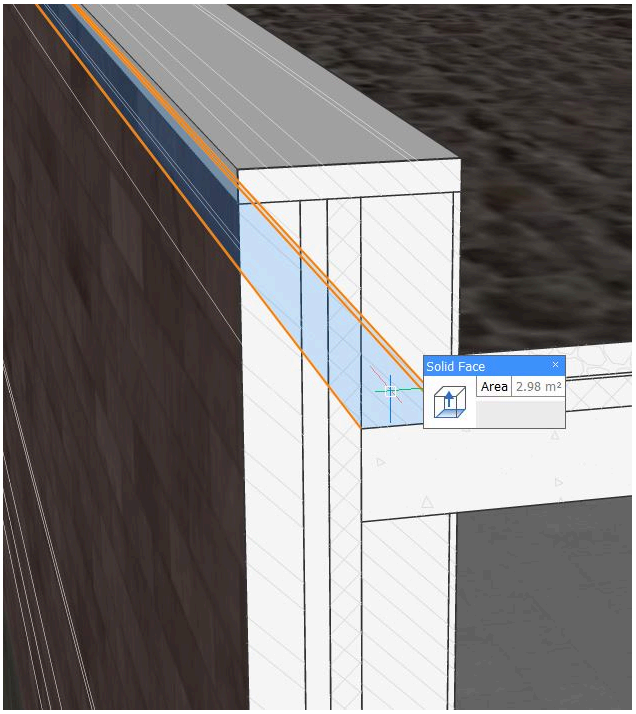
6 Push/Pull the face to the next ply of the wall.



7 To create a wall cap, select the top surface of the wall by pressing Tab and extrude the surface up for 50mm. The created cap isn't classified yet.



8 Delete the part where the insulating building block will come. Hover over the surface of the supporting wall and the gypsum board and press tab to select the surface. Then Extrude till the ply of the wall cap.





9 **Extrude** the same surface again, now a new solid will be created.



20.4.8 BIM project info

20.4.8.1 Commands

BIMPROJECTINFO

20.4.8.2 About

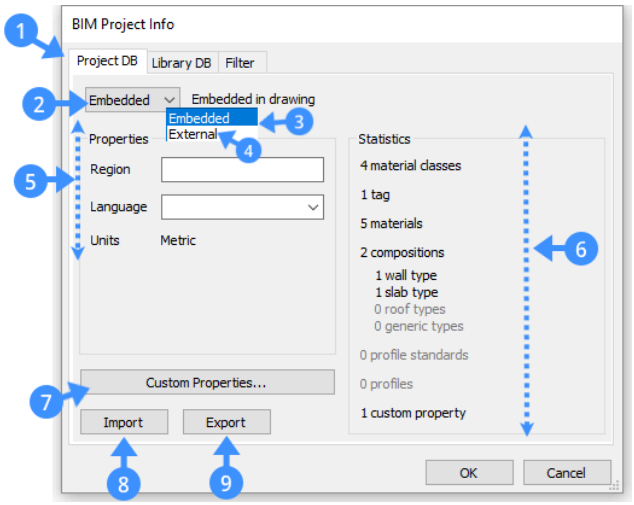
The **BIM Project Info** allows you to control the location, properties, and content of the Project and Library databases. When you launch the BIMPROJECTINFO command in BricsCAD, the BIM project info displays in a dialog box that includes certain components and tabs. Project and library databases of BIM models can be changed using this dialog box.

For more information about this command, visit the Command Reference article BIMPROJECTINFO.

20.4.8.3 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.



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 and .csv 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.

Nota: 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.

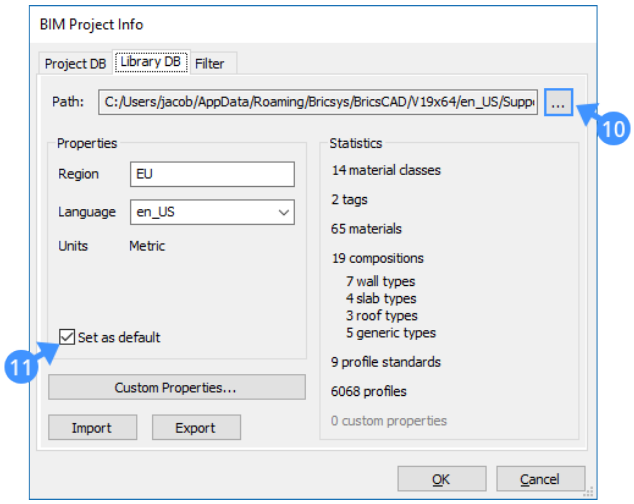
3 To specify the Project Database properties, such as Region and Language, fill out the **Region** field and choose one of the language options in the drop-down list.

4 Optionally, use the custom properties button to add material and composition custom properties in the Project Database.

5 Press the **OK** button to accept it.

20.4.8.4 Procedure: setting the library database

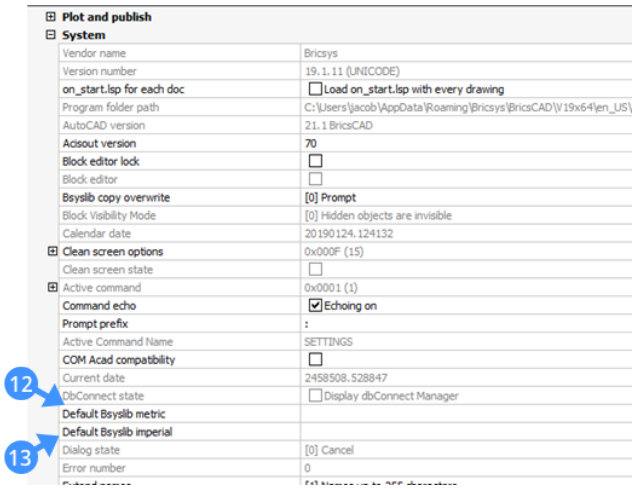
1 Click the **Library DB** tab page in the **BIM Project Info** dialog box. The following illustration shows the Library Database section in the dialog box.



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.

Nota: You can edit the default library settings (12, 13) in the **Settings** dialog under **Program options** > **System**.



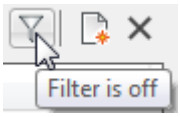
20.4.8.5 The project filter

To each Building Material or Composition, any number of tags can be attached. Any string can be entered as a tag.

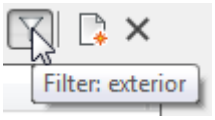
Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed.

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 **Compositions** editors).
- The project filter is defined, but not active.



- The project filter is defined and active.

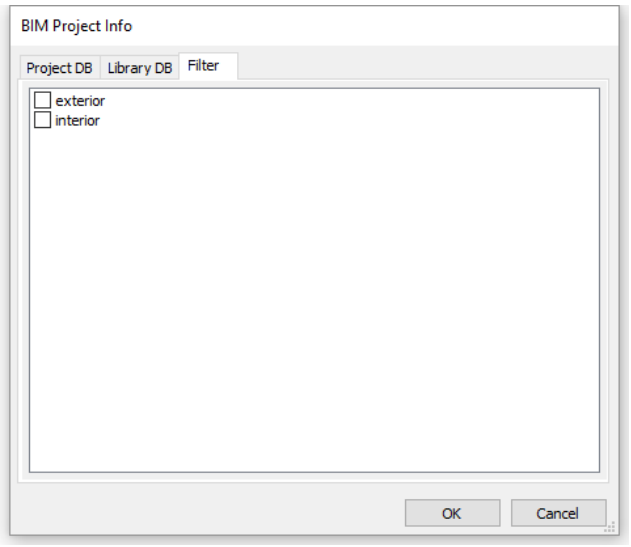


A tooltip shows the current project filter.

20.4.8.6 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.



20.4.9 Using Python scripts (beta)

20.4.9.1 Commands

BIMPYTHON

20.4.9.2 About BIMPYTHON

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a suitable .py Python script.

The Python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

BricsCAD does not ship with a Python Shell, so the scripts would need to be prepared in a text or code editor application.

The Python Programming Language, together with its standard libraries, is embedded within BricsCAD BIM so there is no need for you to install them separately unless you have custom packages and libraries which you wish to use in your scripts.

Before using BIMPYTHON, you need to first turn on the virtual Python environment within BricsCAD.

20.4.9.3 Procedure: activating the Python environment in BricsCAD

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMACTIVATEPYTHON in the Command line and press Enter.

3 **You will be prompted:** New current value for BIMACTIVATEPYTHON [1 for ON/0 for OFf] <1 for ON>:

4 By default, the value should display **<1 for ON>**. That means the Python environment is already activated.

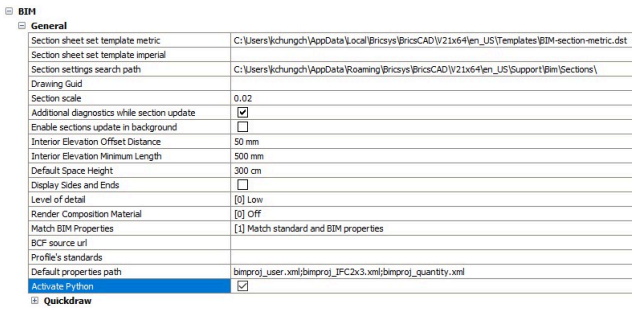
However, If the value shows **<0 for OFF>**, type **1**in the Command line and press Enter to confirm.

Or

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Access the **Settings** dialog box.

3 Tick the box where it says **Activate Python** under the **General** subcategory within **BIM**.



4 Close the dialog box to confirm your updated settings.

20.4.9.4 Procedure: setting up a Python script with the BIMPYTHON module

1 Before gaining access to the API, import or 'call' any desired modules you wish to have in the script.

1 If they are not part of the Python Standard library, make sure they have been installed separately beforehand.

You can "import" a standard module, such as math and "import as" an external one, for example Pyplot, which is a collection of functions in the Matplotlib package:

```
import math
import matplotlib.pyplot as plt
```

2 With your desired libraries "called" in, the next step is to gain access to the API in order to query the elements within the BricsCAD model. This is the range of bim objects and serves as an entry point to the model.

3 Import the briqpy.bim_model:

```
from briqpy import bim_model
```

4 Now, you can query the model with these example statements.

```
# Display info about the lengths of walls in the model
lengths = [wall.prop('Length') for wall in bim_model.filter(Type='Wall')]
print(f'wall lengths. max: {max(lengths)}, avg: {sum(lengths)/len(lengths)}')
```

```
# Create a selection and print the objects
bim_model.filter(Type='Wall', IsExternal=True).select()
for wall in bim_model.filter(Type='Wall', IsExternal=True,
 Length=max(lengths)):
    print(wall)
```

The mappers of the briqpy.ObjectRange are chainable. This is an example to get the parts of the roof that are close to a wall:

```
# Get all parts of roof within 40cm range of all walls
roof_parts = bim_model.filter(Type='Roof').parts()
```

```
roof_parts_close_to_wall = bim_model.filter(Type='Wall').within_distance(40, 'cm',
 search_range=roof_parts)
```

Whereas this is an example of a function statement to filter:

```
# Filter roof parts longer than 50 project units
def is_long(obj):
return obj.prop('Length') > 50
roof_parts.filter(is_long)
```

You can also export and show data in various formats:

```
# create a dictionary list
wall_info = [
    {   'Handle': wall.prop('Handle'),
        'Length': wall.prop('Length'),
        'Height': wall.prop('Height')
    } for wall in bim_model.filter(Type='Wall')]
```

```
# export to .json
import json
file = open('path/to/file.json', 'w+')
file.write(json.dumps(wall_info, indent=4))
file.close()
```

```
# plotting a histogram
import matplotlib.pyplot as plt
import pandas as pd
df = pd.DataFrame(wall_info)
df.hist();
plt.show()
```

```
# export to .csv
df.to_csv(r'path/to/file.csv', index = False, header=True)
```

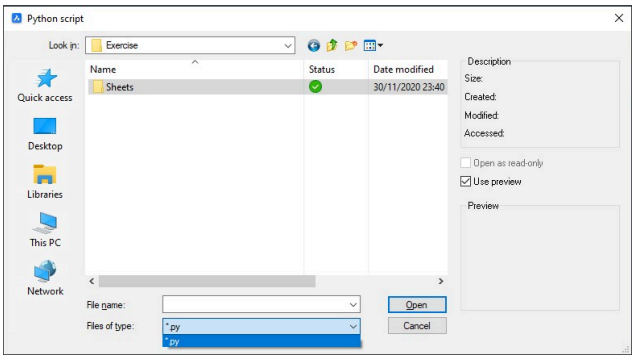
For more information on the BricsCAD API and its various classes, please visit the API chapter below.

20.4.9.5 Procedure: executing the Python script

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMPYTHON in the command-line and press Enter.

3 A dialog box is displayed where you can only select a Python script file (*.py). Select it and click **Open** to execute the file.



4 Unless you have specified the data to be exported or displayed in an external program outside BricsCAD, BricsCAD reports the output in the Command line panel.

20.4.9.6 API dictionary

Classes and syntaxes

class briqpy.Object- defines BricsCAD BIM Objects

```
prop(prop_name)
```

- Returns the property value of the Object with the given name.

```
distance_to(  other_obj, units='mm', distance_mode='exact' )
```

- Calculate the distance between two Objects.
- Options for argument *units*: any units value form insunits ('Centimeters', 'Feet', 'Parsecs' etc.) and these abbreviations: 'mm', 'cm, 'm', 'km', 'ft'.
- Options for argument *distance_mode*: 'bbox_center', 'bbox', exact'.

```
parts()
```

- Return the ObjectRange containing the sub-elements of this object.

```
parent()
```

- The opposite of **parts**(), returns the parent object of this sub-element.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Return the ObjectRange of objects that are within the distance of argument *distance*.
- Options for argument *units*: see distance_to.
- Options for argument *distance_mode*: see distance_to.

```
select()
```

- Add Object to selection

```
deselect()
```

- Remove Object from selection.

```
__eq__() and __hash__()
```

- Makes Object interoperable with e.g. python set or dictionary.

class briqpy.ObjectRange - defines collection of BricsCAD BIM Objects

```
filter( function )
```

- Filtering this range with a function parameter.

```
filter( **conditions )
```

- Filtering this range with conditions given as keyword arguments.

```
parts()
```

- Return the ObjectRange of all the parts of all the elements in this range.

```
parents()
```

- Return the ObjectRange of all parent objects of elements in this range.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Returns the ObjectRange of objects that are within distance to any object in this range.

```
select()
```

- Add Objects to selection.

```
deselect()
```

- Remove Objects from selection.

```
__len__ ()
```

- Return the number of Objects in this range.

20.4.10 Classification codes

20.4.10.1 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.



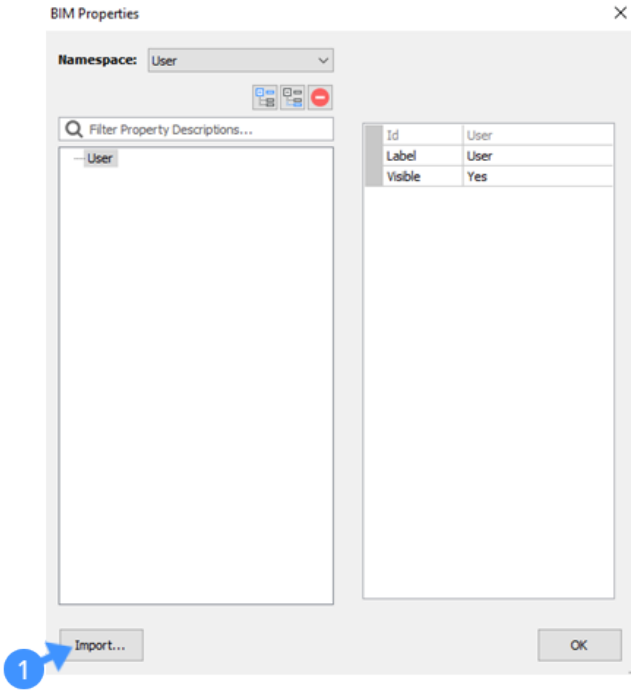
For more information about this command, visit the Command Reference article BIMPROPERTIES.

20.4.10.2 Assigning classification codes to the BIM

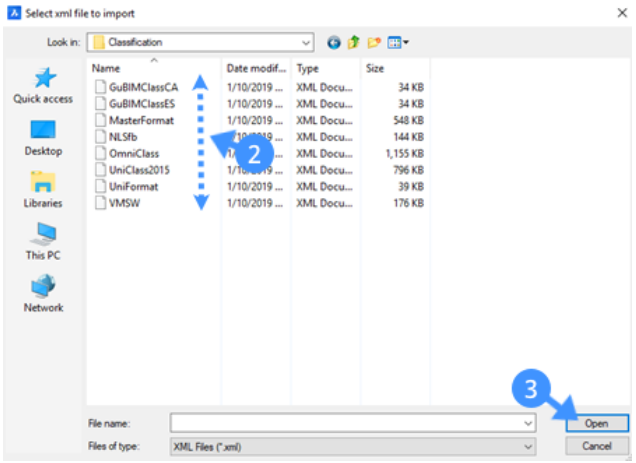
1 Open the BIM Properties dialog box from the Quad (**BIM** > **Model**).

Nota: Quad should be in **No Selection** state:

A dialog box appears:



2 Click the **Import** (1) button at the bottom-left corner of the dialog box. A file dialog displays:



3 Choose the classification system on the list (2), and click **Open** (3) to import it into the BIM.

Nota: If the file dialog does not show the **Classification** folder, you should manually enter this path in the address bar: C:\Users\<User>AppData\Roaming\Bricsys\BricsCAD V19x64\en_US\Sup Support \Bim\Classification.

Available classification systems:

GUBIMClassCA	Classification system for the BIM Users group of Catalonia.
GUBIMClassES	Unified classification system of building elements designed for the AEC industry in Spain.

Master Format	Standards for commercial 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 building specifications, 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.

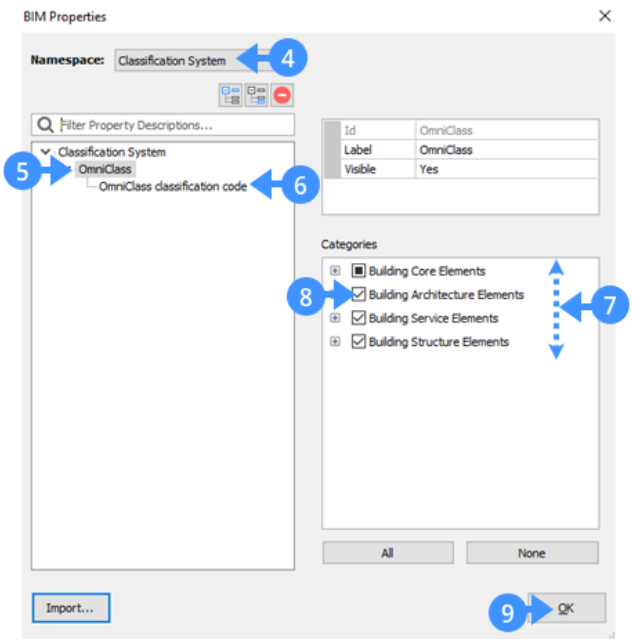
By default, the classification system is applied to all sorts of building elements (7).

Optional: Deselect the categories of building elements to which the classification system is applied.

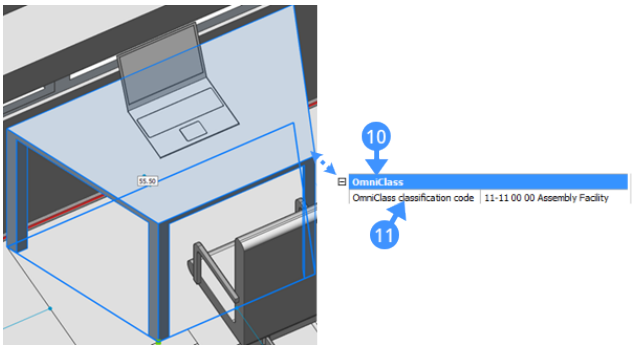
To do so, check the box next to the category name (8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only.

Click the **OK** button (9) to finalize the procedure.

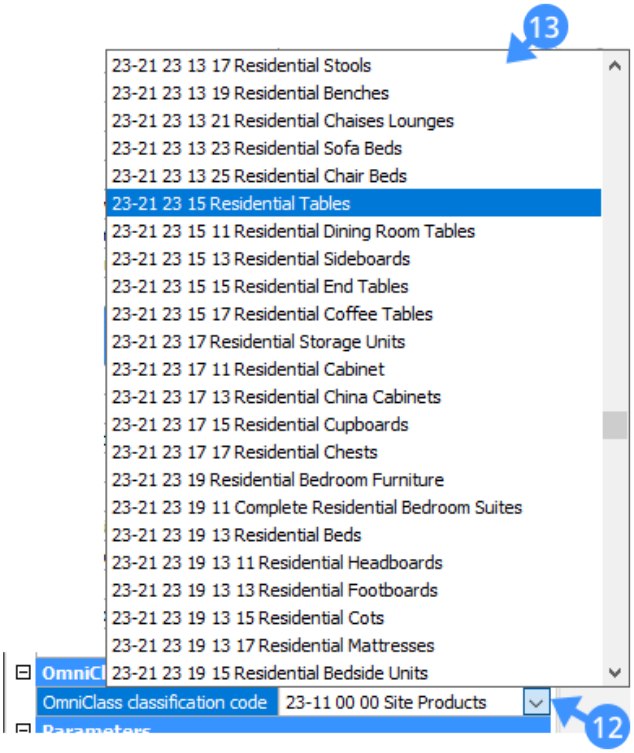
Orimport another classification system following the previous steps.



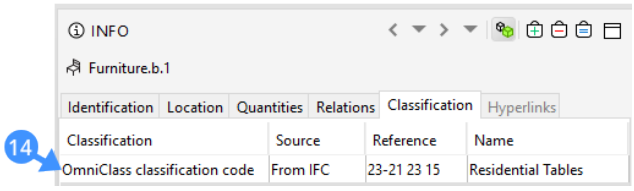
5 Select the element to which you want to assign a classification code, and open the properties panel. The classification system (10) and code (11) should be available in the properties panel.



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).



Nota: 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.



20.4.11 Creating and editing BIM spaces

20.4.11.1 Commands

BIMSPACE, BIMUPDATESPACE, BIMIFY

20.4.11.2 About

You can create BIM Spaces from enclosed spaces. When the wall elements enclose a space in your model, you can use the **BimSpace** tool to define a BIM Space. These spaces are solids so they can be easily manipulated. The 3D space geometries added to the current model are associated with space tags. The space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. The values of each attribute can be changed using the properties panel.





For more information about this command, visit the Command Reference article BIMSPACE.

20.4.11.3 Defining a BIM space

1 Select the **Space** tool under **Home** > **Classify** > **Space or** type BIMSPACE in the Command line.

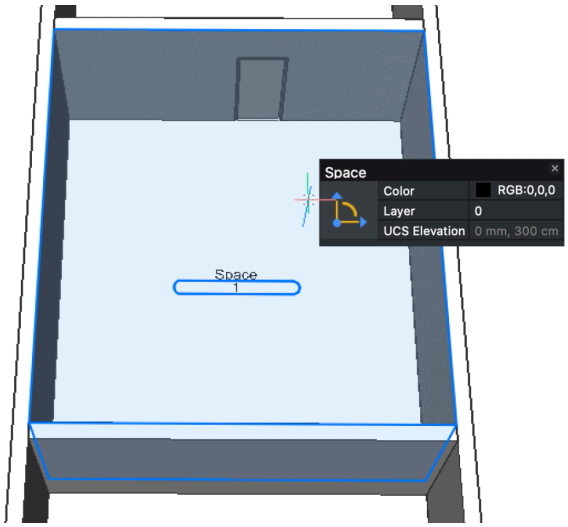
You are prompted:*Pick a point inside the space or [Edit];*

2 To define a BIM space, pick a point inside the space area in your BIM model.

3 Move the cursor inside the space area to pick a point, the **Dynamic UCS** enables you to define the bottom plane of the space. A green checkmark will appear next to the cursor to indicate a possible location. A red cross will appear if the plane isn't available.

4 In this plane, an area enclosed by wall elements or any space bounding elements is found and a space tag is placed in the center of the space.

5 **Space Bounding elements** can be Walls, curtain walls, slabs, columns, but also hatches and polylines can be counted as space bounding elements.



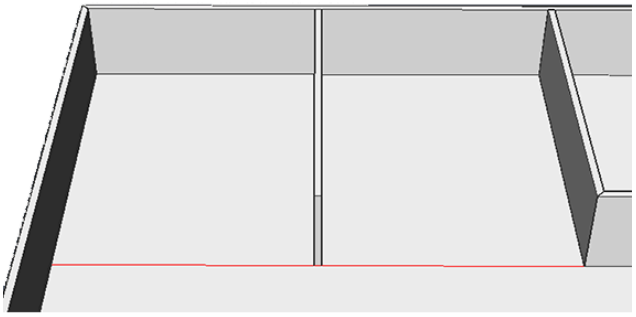
6 Edit the properties of the BIM Space in the properties panel.

7 Change the representation from solid to footprint for a less notable visualization.

20.4.11.4 Procedure: creating BIM spaces from virtual boundaries

There are cases in which a virtual boundary is needed. For example, separate parking spaces, large auditorium spaces for energy analytics, etc.

1 Divide the Space with linear entities. This can be hatches or lines.

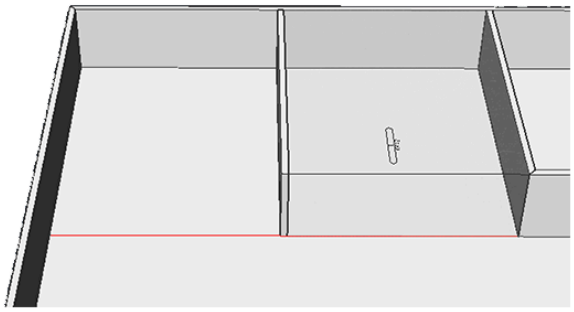


2 Select the dividing elements (the red line).

3 Classify them as a space bounding element: **Home** > **Classify** > **Wall**.

Orin the Quad under **BIM**.

4 Type BIMSPACE in the Command line and pick a point inside the space.



Nota: If the line doesn't divide the space, go to the **Properties** panel and then to **BIM**. Check if **Space bounding** is turned on.

20.4.11.5 Creating BIM spaces automatically

BIMIFY and BIMQUICKBUILDING create BIM spaces and automatically assign information to them. For example, the story and building of the space (see BIMSPATIALLOCATIONS). The commands will search for space bounding elements and create spaces wherever possible.

These spaces act like manually made spaces and can be edited according to your own needs.

20.4.12 Custom properties

20.4.12.1 Commands

BIMPROPERTIES

20.4.12.2 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.

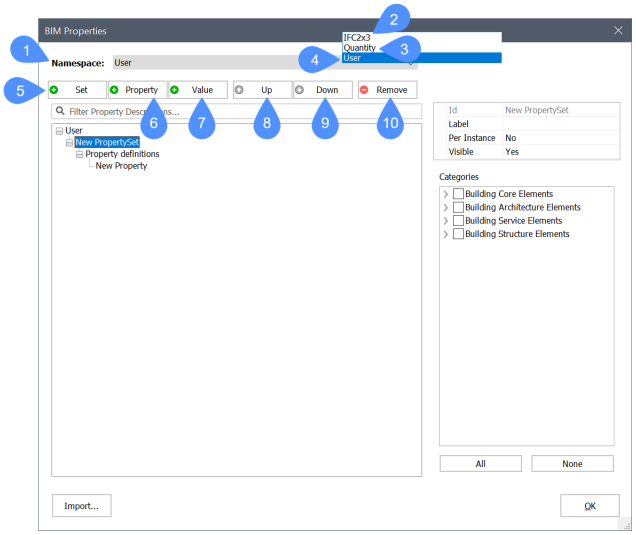
For more information about this command, visit the Command Reference article BIMPROPERTIES.

To learn more about creating user-defined quantity definitions, visit the Procedure article for Customizing BIM Quantities.

20.4.12.3 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.



(1) Namespace	Groups property sets in namespaces.
(2) IFC2X3 namespace	Shows the IFC2x3 properties.
(3) Quantity namespace	Shows some hardcoded quantities ***Nota:*** These have been deprecated and replaced by the Quantities in the **IFC2x3** namespace.
(4) User namespace	Shows the custom properties.
(5) Add property set button	Creates a new property set.
(6) Add property button	Creates a new property definition.
(7) Add value button	Creates a new value definition.
(8) Move up button	Moves the selected property or value up the list.
(9) Move down button	Moves the selected property or value down the list.
(10) Remove button	Removes the selected property set, property definition or value definition.

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:

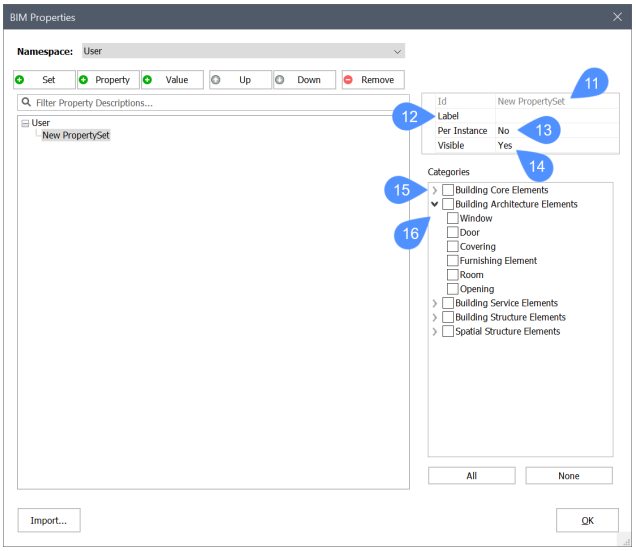
- Type a name in the **Id** (11) field.
- Type a name in the **Label** (12) field.

The **Label** name appears in the properties tree. If the **Label** name is not defined, the **Id** name displays. The **Id** must be unique.

Optionally, you can choose to have the custom property be only applied to specific instances (13), and choose the visibility (14).

- Select the building element categories (15) the property set applies to.

When you expand one of the categories, a different type of building elements (16) 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.

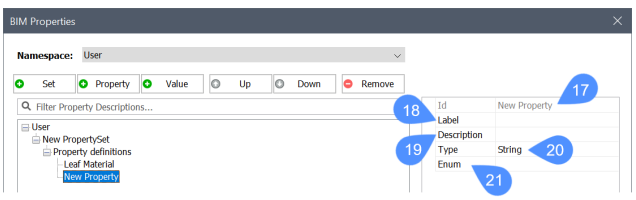


3 To add a property to the currently selected property set, first, select a property set, then click the Add Property button to add a new property.

4 To edit the property do the following:

- Type a name in the **Id** (17).
- Type a name in the **Label** (18).
- Optionally, type a description in the **Description** (19).
- Select the property type. The options are **Boolean**, **Integer**, **Real**, and **String** (20).
- Optionally, define enumerated values (21).

The following illustration shows the **BIM Properties** dialog box after inserting a new property.



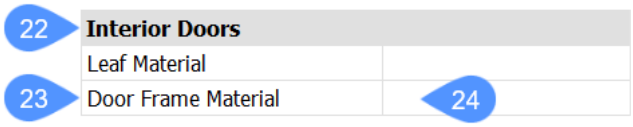
5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button to remove.

6 Press the **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 (22) are placed in a new category (23) 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 (24).

The following illustration shows the custom properties in the properties panel.



20.4.13 Customizing BIM quantities

20.4.13.1 Commands

BIMPROPERTIES

20.4.13.2 About

Entities have a set of properties which can be viewed in the **Properties** Panel. If an entity has been assigned a certain **BIM classification** or **Type** (e.g. **Wall** or **Slab**), additional properties will be displayed such as IFC Common properties and Base quantities. Which properties are displayed in the panel is managed in the **BIM Properties** dialog, which is opened by the BIMPROPERTIES command.

20.4.13.3 Namespaces

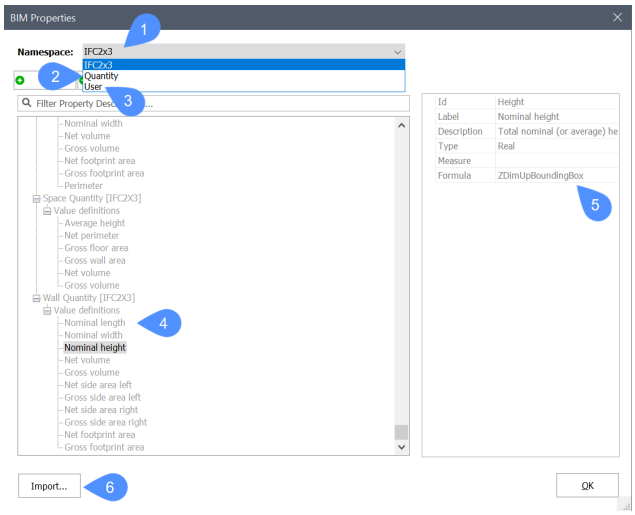
In the BIM Properties dialog, by default three namespaces are available: **IFC2x3** (1), **Quantity** (2) and **User** (3). The **IFC2x3** namespace contains properties that are defined by the IFC2x3 schema. This contains both common properties (e.g. Wall Common properties, such as **Fire Rating** and **Acoustic Rating**), and Quantities (e.g. **Wall Quantity** > **Nominal lenght** (4)). These Quantities are based on so-called Core quantities (5) of the solid. See Core quantities for more information.

The **Quantity** namespace contains some hard-coded quantities of BIM objects also based on the Core quantities, much like in the **IFC2x3** namespace.

> ***Nota:*** This **Quantity** namespace is deprecated. It has been replaced by the Quantities in the **IFC2x3** namespace, however it is still available in the **BIM Properties** dialog for backward compatibility with versions V20 and older.

The **User** namespace exists for the user to create their own property definitions. See also the Procedure article Custom Properties for more information on custom properties.

It is also possible to import namespaces, e.g. the **IFC4** namespace. This can be done by clicking the **Import** button (6) on the bottom left.



Nota: using this method the namespace will only be imported for this particular project. If you want namespaces to be imported over all your projects, open the **Settings** dialog and under the **BIMGeneralDefault properties** path you can add a new path. By default, it should take you to *C:\Program Files\Bricsys\BricsCAD Vxx en_US\Support*, where you can find for instance the **bimproj_IFC4.xml** containing the **IFC4** namespace.

20.4.13.4 Core quantities

Core quantities are quantities of solids that are calculated based on the geometric features of that solid, and that can be used in user-defined quantities. These core quantities are also the basis for the quantities defined in the **IFC2x3** and **IFC4** properties as described in the previous section.

For example, under **Wall Quantities** in the **IFC2x3** namespace, the **Nominal Height** of a wall is defined as the **ZDimUpBoundingBox** of the solid, which is a Core quantity. A full list of Core quantities can be found below.

Core Quantity	Description	Measure	Type
NumberOfPlies	Number of plies in the attached composition.		**Integer**
XDimUpBoundingBox	X dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
YDimUpBoundingBox	Y dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
ZDimUpBoundingBox	Z dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**

Core Quantity	Description	Measure	Type
XDimFreeBoundingBox	X dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
YDimFreeBoundingBox	Y dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
ZDimFreeBoundingBox	Z dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
DistanceBetweenMajorSurfaces	Distance between major surfaces.	**Length**	**Real**
FirstMajorSurfacePerimeter	Perimeter of the first (largest) major surface.	**Length**	**Real**
SecondMajorSurfacePerimeter	Perimeter of the second major surface.	**Length**	**Real**
LinearAxisLength	Length of the linear element's axis.	**Length**	**Real**
VariablePlyThickness	Thickness of a variable ply.	**Length**	**Real**
FirstMajorSurfaceNetArea	Area of the first (largest) major surface after subtracting openings.	**Area**	**Real**
FirstMajorSurfaceGrossArea	Area of the first (largest) major surface before subtracting openings.	**Area**	**Real**
SecondMajorSurfaceNetArea	Area of the second major surface after subtracting openings.	**Area**	**Real**
SecondMajorSurfaceGrossArea	Area of the second major surface before subtracting openings.	**Area**	**Real**
IsMajorSurfacesParallel	Indicator if the major surfaces are parallel.		**Boolean**
TotalSurfaceNetArea	Area of the solid's surfaces after subtracting openings.	**Area**	**Real**
TotalSurfaceGrossArea	Area of the solid's surfaces before subtracting openings.	**Area**	**Real**

Core Quantity	Description	Measure	Type
FootprintNetArea	Area of the bottommost solid's surfaces after subtracting openings.	**Area**	**Real**
FootprintGrossArea	Area of the bottommost solid's surfaces before subtracting openings.	**Area**	**Real**
ProjectedNetArea	Area of the solid's surfaces projected to global XY plane after subtracting openings.	**Area**	**Real**
ProjectedGrossArea	Area of the solid's surfaces projected to global XY plane before subtracting openings.	**Area**	**Real**
CrossSectionNetArea	Area of the cross section for linear elements after subtracting openings.	**Area**	**Real**
OuterSurfaceNetArea	Area of the outer surface for linear elements after subtracting openings.	**Area**	**Real**
NetVolume	Volume of the solid after subtracting openings.	**Volume**	**Real**
GrossVolume	Volume of the solid before subtracting openings.	**Volume**	**Real**

20.4.13.5 User-defined quantities

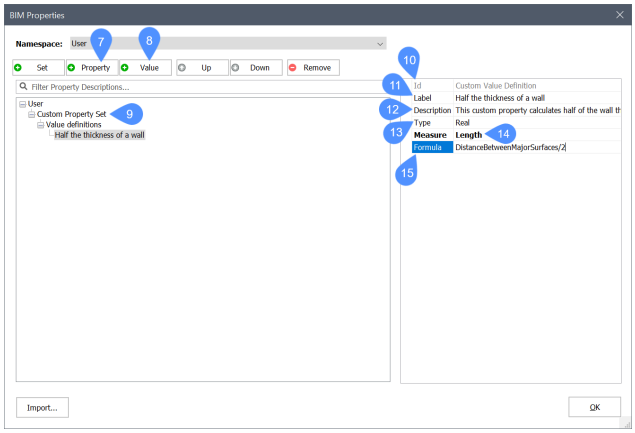
In the **User** namespace it is possible to create custom properties inside a custom property set using the **Property** (7) button, see Procedure article Custom Properties.

It is also possible to create user-defined quantities, using the **Value** (8) button. In the image below, a user-defined quantity was created in a **custom property set** (9). This user-defined quantity (or Value Definition) has the following attributes:

- **Id** (10): a unique identifier.
- **Label** (11): the name of quantity which will show up in the properties panel. This can be the same as the ID, however **the ID must be unique**.
- **Description** (12): a description of the quantity.
- **Type** (13): the type of the quantity. Typically, a quantity will be defined as a **Real** value.
- **Measure** (14): the measure of the quantity, i.e. is it a **length** value, an **area**, **volume** or **mass**?

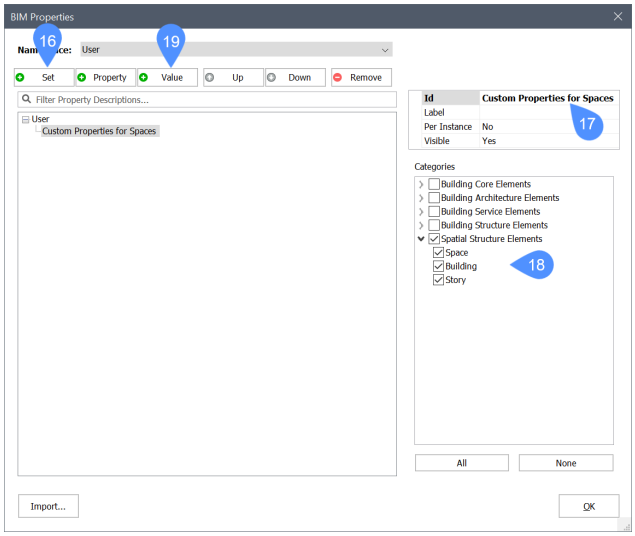
- **Formula** (15): the actual expression of the quantity. Here Core quantities can be used, as well as mathematical expressions as defined in the Procedure article Working with Parameters and Constraints.

In this example, a user-defined quantity is defined that displays half the thickness of a wall, which is calculated as **DistanceBetweenMajorSurfaces/2**.



20.4.13.6 Procedure: creating a user-defined quantity

In this part you will learn how to make user-configurable quantities. By way of example, we will define a quantity of Spatial objects such as a room. The quantity will be the **Compactness** which can be calculated as the volume of the space divided by the sum of the areas on all sides of the space.



1 Open the **BIM Properties** dialog by typing BIMPROPERTIES in the Command line.

2 Set the **User** namespace as current namespace.

3 Create a new Property Set by clicking the **Set** button (16), and give it a name (17).

4 Assign the Property Set to one or more categories (18). In this example we will assign it to spatial elements.

5 Create a new Value Definition by clicking the **Value** button (19).

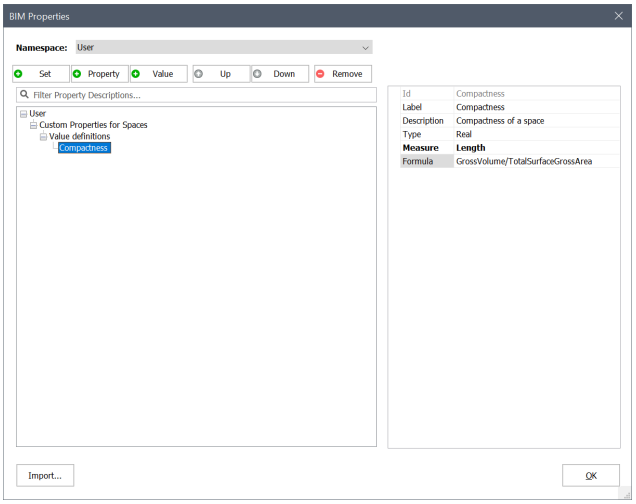
6 Give this Value Definition an **Id** and a **Label**, e.g. **Compactness**.

7 **(Optional)** Give this Value Definition a description.

8 Assign the correct **Type** to this Value Definition. Because our Value Definition will be based on a calculation of two **Real** values, this will also be a **Real** value.

9 Assign a **Measure** to the Value Definition. Because the result of the calculation will be a volume divided by an area, we will end up with a **Length** measure.

10 Fill in an expression in the **Formula** field. The volume of a space is a Core quantity with the name **GrossVolume**. The sum of areas of the space on all sides is also a Core quantity with the name **TotalSurfaceGrossArea**. Hence, we can define the **Compactness** = **GrossVolume/ TotalSurfaceGrossArea**.



11 Click **OK** to close the **BIM Properties** dialog.

12 Create a volume that represents a space, either by creating surrounding elements like walls and a slab, and by defining a space inside using the BIMSPACE or BIMIFY commands, or by creating a solid and using BIMCLASSIFY to classify it as a **Space**.

13 Select the space. In the **BIM Properties** panel you should now see your user-defined property under a separate property set.

20.4.14 Load bearing direction for slabs

20.4.14.1 Commands

BIMSETLOADBEARINGDIRECTION

20.4.14.2 About BIMSETLOADBEARINGDIRECTION



In BricsCAD BIM, the BIMSETLOADBEARINGDIRECTION command allows you to define 0, 1, or 2 load bearing directions for a slab.

After defining the load bearing direction in the 3D model, you can indicate the load bearing direction in the floorplan using the **Manual** option of the BIMTAG command.

20.4.14.3 Specifying the load bearing direction(s)

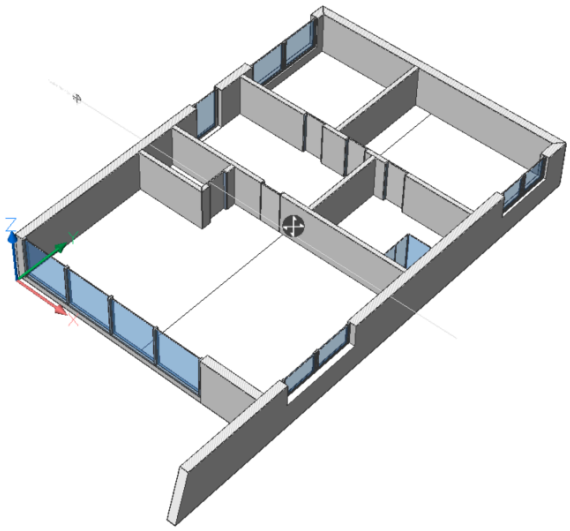
Only solids which are classified as a **BIM Slab** can be processed by the BIMSETLOADBEARINGDIRECTION command. Use the BIMIFY or BIMCLASSIFY command to classify a solid as a **BIM Slab**.

1 Launch the BIMSETLOADBEARINGDIRECTION command.

1 **You are prompted:** Select a slab

2 Click the slab. It may help to set the **Clip Display** property of a plan section to select the slab.

A widget displays at the center of the slab.



3 Repeatedly click the widget. Press Enter or right-click to confirm the loadbearing direction(s).

20.4.14.4 Adding a load bearing symbol in plan section

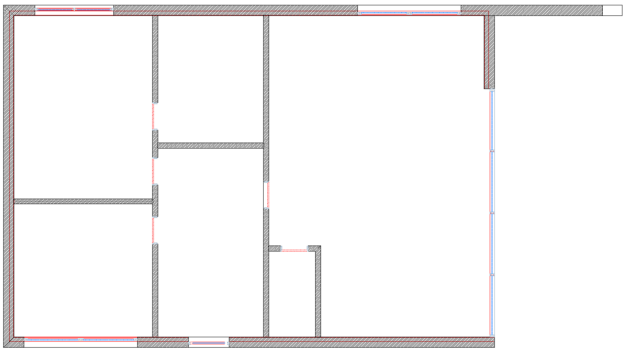
1 When you insert the load bearing tag in a plan section, you need to select a line, which is generated by the slab. Since slabs usually overlap with walls, you need to set the **Hidden Lines** > **Show** property of the **BIM Section** entity to **Yes**.

Hidden Lines

Show	Yes
Color	Red
Layer	0
Linetype	Continuous
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer

Next, execute the BIMSECTIONUPDATE command of the plan section.

2 Open the **Plan** section.



3 Launch the BIMTAG command.

4 Choose the **Manual** option.

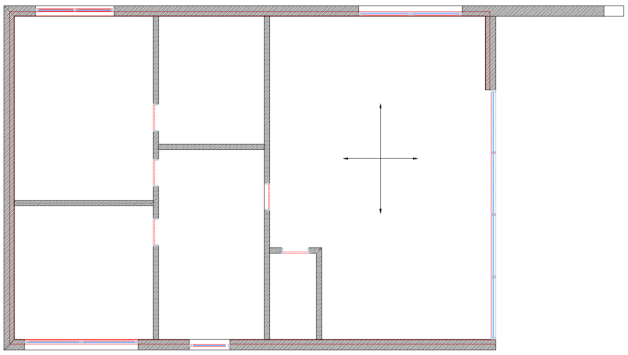
5 Choose the **Change Tag Type** option.

6 Choose the **Load-Bearing Direction** tag type.

7 Click on a point on the edge of the slab.

8 The tag displays, attached to the cursor.

9 Click to place the tag.



20.4.15 Physical materials

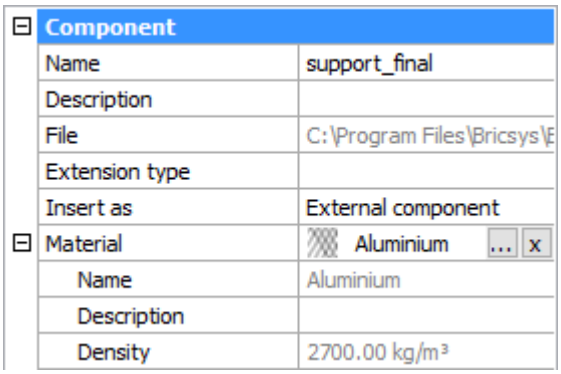
20.4.15.1 Commands

BLMATERIALS

20.4.15.2 Opening the physical materials editor

Do one of the following:

- Launch the BLMATERIALS command.
- Click the **Browse** button (…) of the **Material** property of a component in the **Mechanical Browser**.



20.4.15.3 Creating physical materials

You can create physical materials in either the Project or the Library database. You can create new materials from scratch or copy them from an existing material.

Creating a physical material from scratch

Select a material in either the **In Project** or the **In Library** material list.

Do one of the following:

1 Click the **New Material** button ( ) on the **Physical Materials** dialog.

2 Right-click and choose **New Material** in the context menu.

A material named **New** is added to the selected database. Now you can edit the new 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 the following:

1 Right-click on the material you want to copy and choose **Duplicate** in the context menu.

2 A material named **<existing_material>-2** is added in selected database.

3 Adjust the copy of the material.

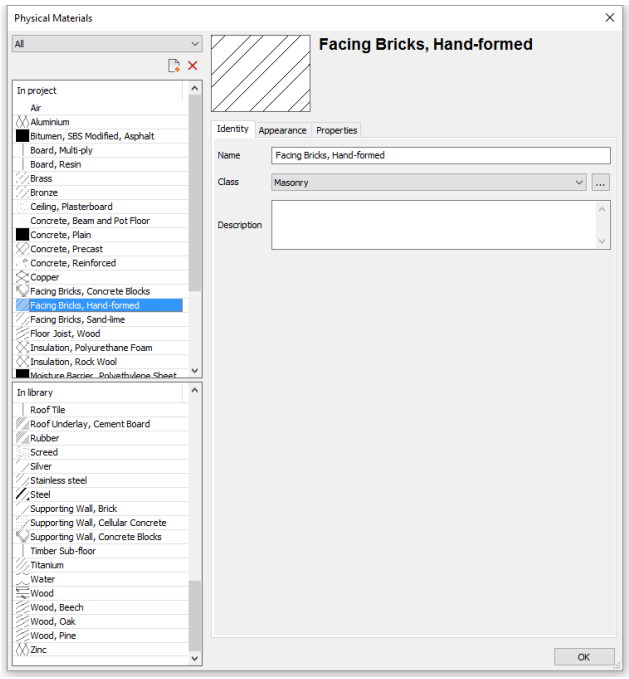
20.4.15.4 Editing a physical material

The various settings of a physical material definition are divided into three groups under three tabs on the **Physical Materials** 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 you want to edit in either the **In Project** or in the **In Library** list on the **Physical Materials** dialog.

Identity tab page

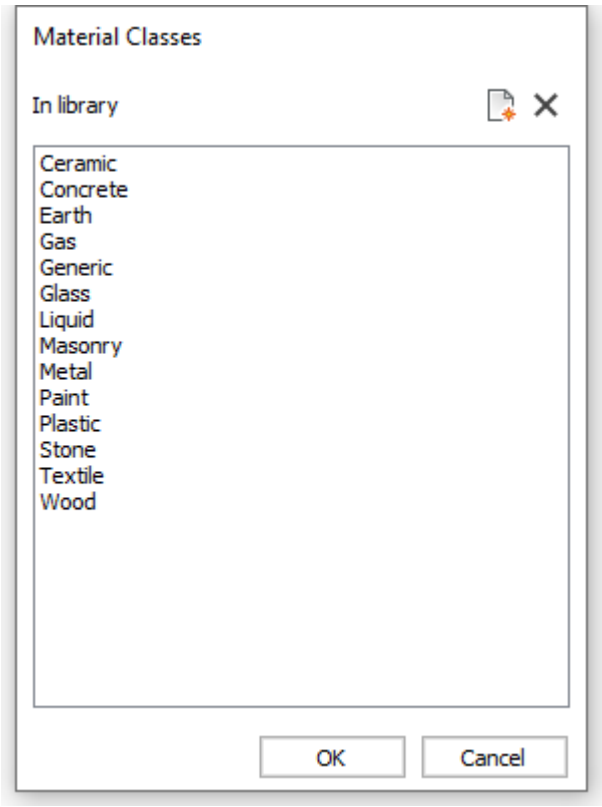


1 Type a **Name** for the material.

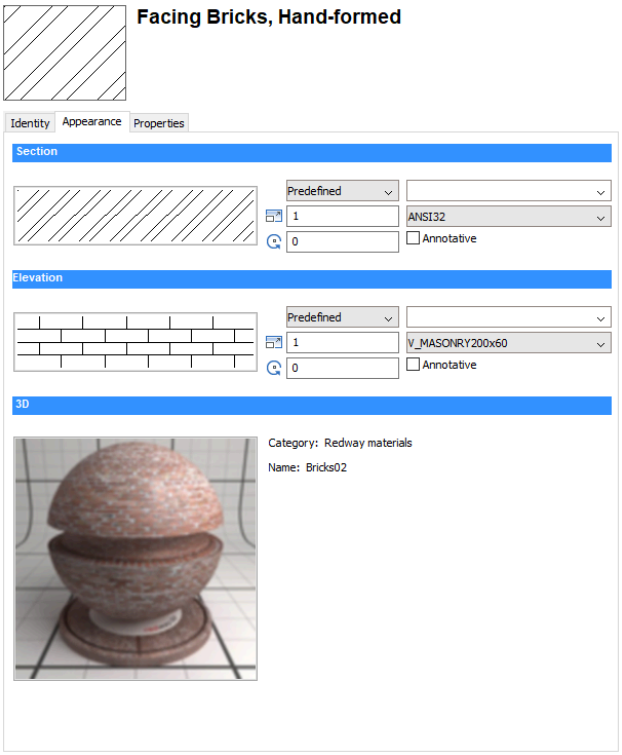
2 Select a **Class** from the drop-down list*.

3 **(Optional)** Type a **Description**.

* Click the **Browse** button (...) to open the **Material Classes** dialog where you can create new material classes.



Appearance tab page



- **Section settings**

The **Section** settings define the hatching of the section boundary lines.

a Click the **Hatch Type** button and select the type of the hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom**.

b Do one of the following to select a predefined or custom hatch pattern:

- Click the **Hatch** preview and select the pattern in the the **Hatch Pattern** dialog box.
- Click the **Hatch Pattern Name button** and select the pattern in the list.

c 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 **Scale** property of the **BIM** section. As a result, the printed scale of the hatch pattern is independent of the section plane scale.

d Set the **Rotation** angle.

e In the **Layer** field, either type a name or click the arrow button and select a layer in the list.

f Optionally, check the **Annotative** option.

- **Elevation settings**

  The **Elevation** settings define the surface hatching of the section background.

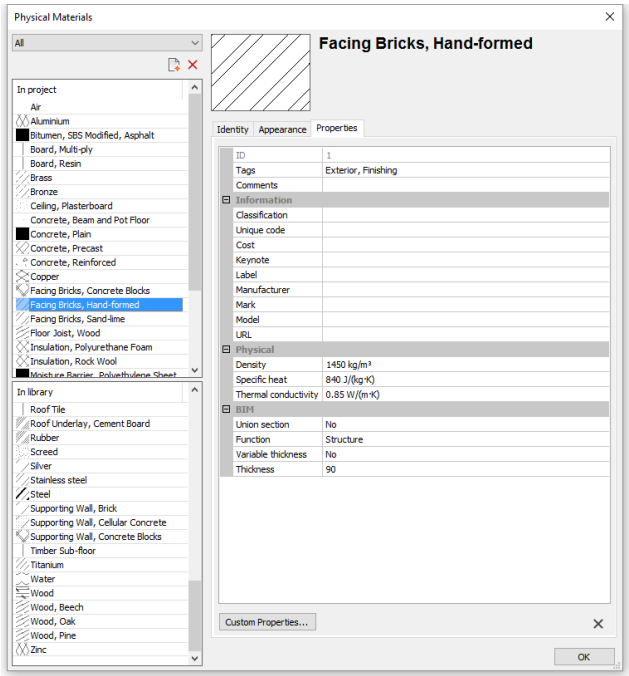
  See **Section** settings above, except for the **Scale** setting. The elevation hatch is independent of the **Scale** property of the **BIM** section.

- **3D settings**

  The **3D** settings define the rendering material.

  Double-click the preview image and select a material in the **Select Render Material** dialog box.

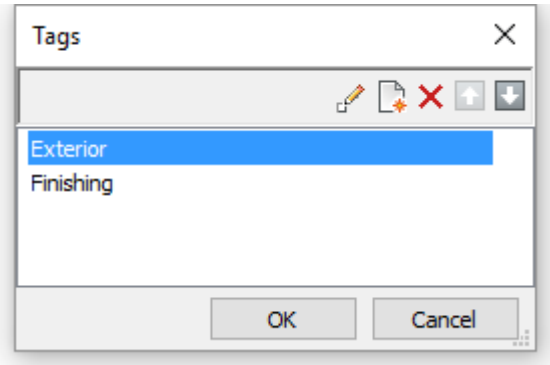
Properties tab page



1 Edit the **Tags** field: tags are separated by commas.

Tags are case sensitive.

Click the **Browse** button (...) to open the **Tags** dialog.



2 Click the **New Item** button () to add a new tag at the bottom of the list.

3 Select a tag in the list and click a tool button:

- : Edit the selected tag.
- : Delete the tag.
- : Move the tag up.
- : Move the tag down.

4 **(Optional)** Expand the **Information** properties and fill out the necessary fields.

5 **(Optional)** Expand the **Physical** properties and specify the physical properties of the material.

6 Expand the **BIM** properties.

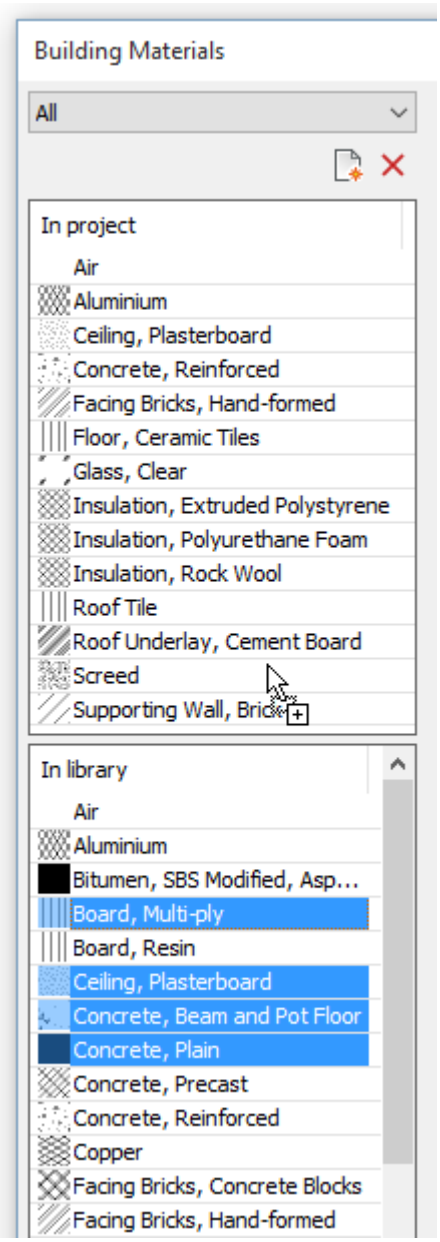
- **Union Section**: Select **Yes** to remove connection lines in sections between different solids of this material. Select **No** to keep the connection lines.
- **Union Elevation**: Select **Yes** to remove connection lines in elevations between different solids of this material. Select **No** to keep the connection lines.
- Select a **Function** from the drop-down list.
- **Variable Thickness**: select **Yes** to allow a variable thickness. Select **No** to not allow a variable thickness.
- Type a default thickness in the **Thickness** field.

7 Click **OK**.

20.4.15.5 Copying materials between the project and library databases

1 Select the material(s) in the source database.

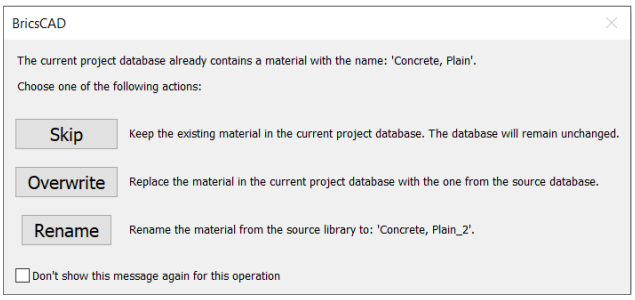
1 Press and hold the Ctrl-key to select multiple materials.



2 Drag the materials to the target database.

3 The selection set is copied.

If one or more materials already exist in the target database, an alert box displays:



Click the appropriate button to either overwrite, duplicate or skip the existing material. If the **Apply to all materials in this operation** option is selected, the option applies to all existing materials. Duplicated materials have an index number added to their name.

20.4.15.6 Deleting physical materials

1 On the **Physical Materials** dialog box, select the material(s) in the Project or Library database.

2 Click **Delete**.

Nota: You cannot delete materials that are used in a composition or a mechanical component.

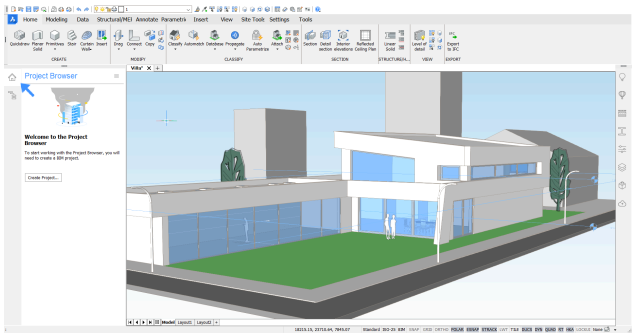
20.4.16 Project browser

20.4.16.1 Commands

DATAEXTRACTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

20.4.16.2 The project browser panel

You can find the **Project Browser** panel on the left-hand side of the screen.



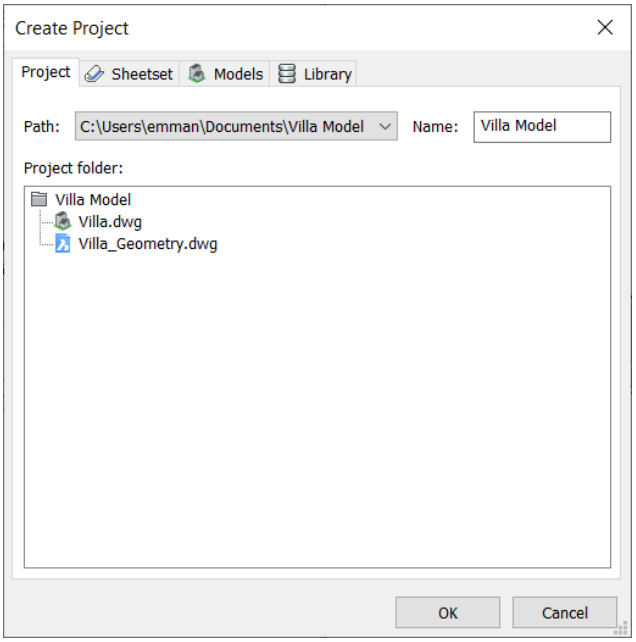
Nota: If the **Project Browser** doesn't display in the panel, right-click in the panel and select **Panels** > **BIM Project Browser**

20.4.16.3 Create a BIM project

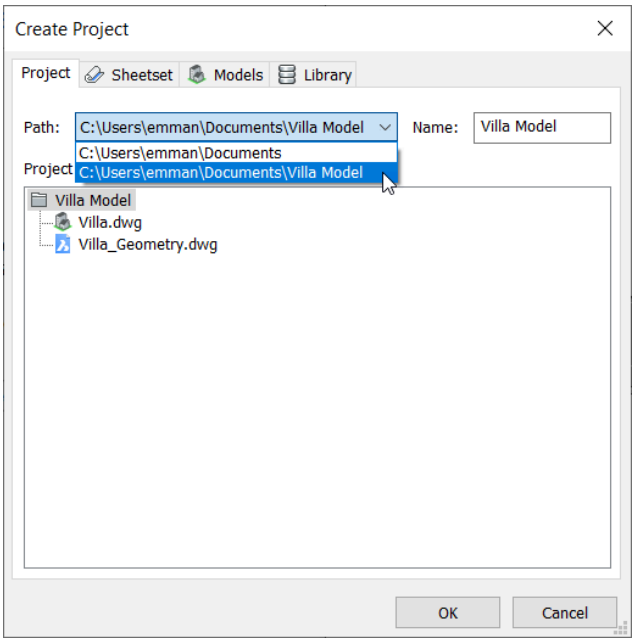
To create a BIM project, you must select one or more model drawings.

Project

1 Click the **Create Project...** button in the **Project Browser**. A dialog box appears.



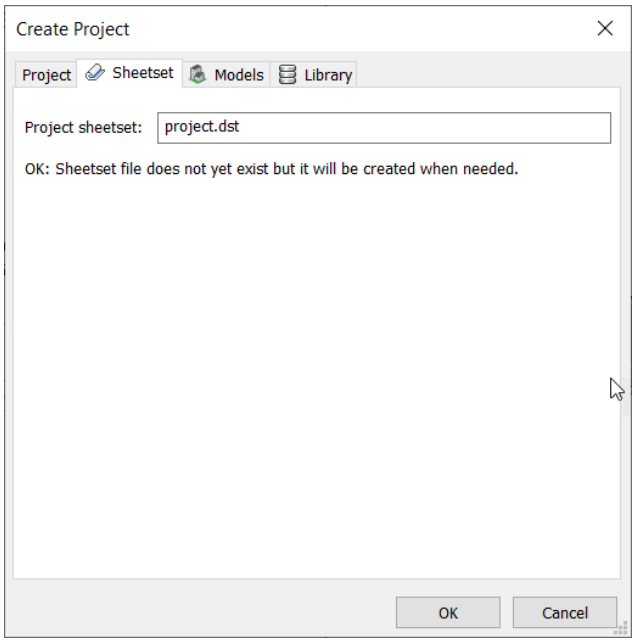
2 **(Optional)** Click the **Path** list button and select a parent folder.



3 **(Optional)** Type a name in the **Name** field (the default name is the name of the parental folder).

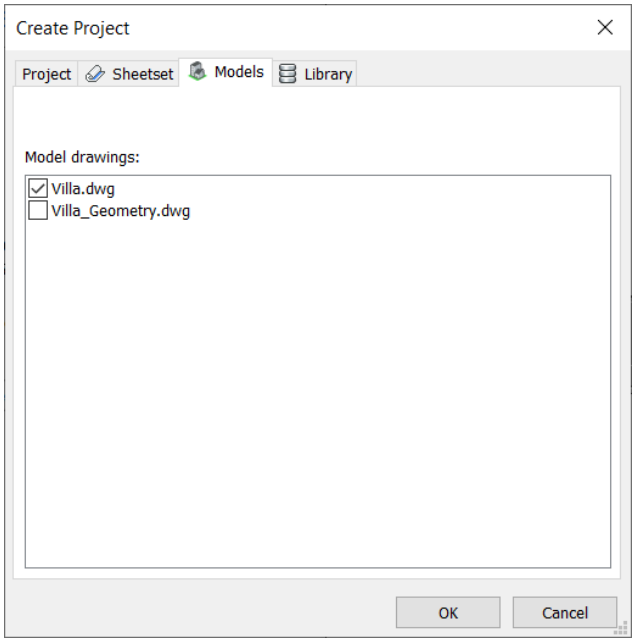
Sheetset

Click the **Sheetset** tab. Enter a name for the **Project sheetset** (the default name is project.dst).



Models

Click the **Models** tab. Here, the files in the selected parent folder (and subfolders) are displayed. Check all the models you want to incorporate in your drawing.



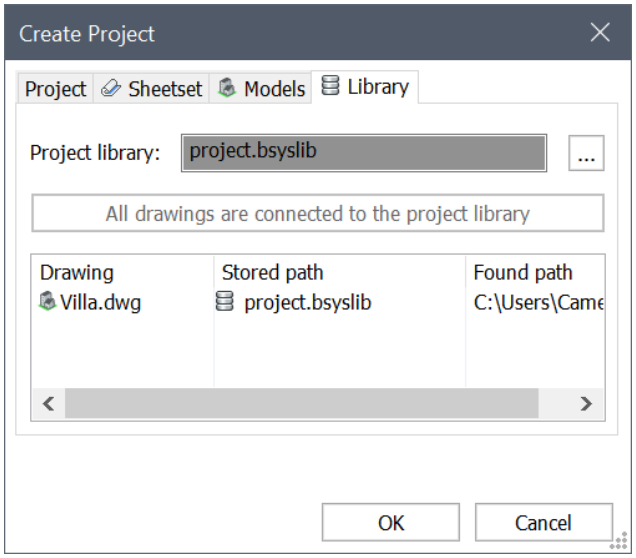
Library

The **Project Library** can either be **external** or **embedded**. Sharing the **project library** means that you can use the same library over different drawings, this is called an external library. The shared library is stored in an external BIM Database File with the .bsyslib extension. An embedded library stores the **Project Library** in the drawing file.

- To make an external library:

Nota: By default, the library is set to **Embedded**.

a Go to the **Library** tab. Click the **Browse** button (...) of the **Project Library** option. Enter a name for the **Project library** (the default name is project.bsyslib).



b Click **OK** to display the **Sheetset Setup** dialog box.

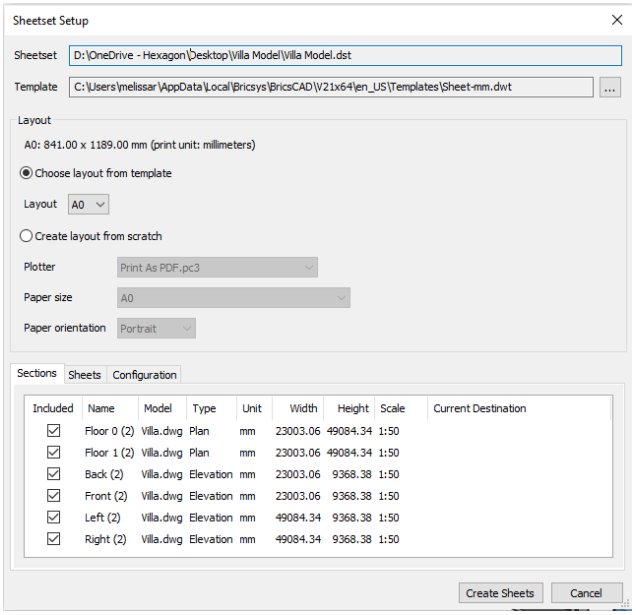
20.4.16.4 Sheetset setup

Sheetset: Shows the path of the sheetset file.

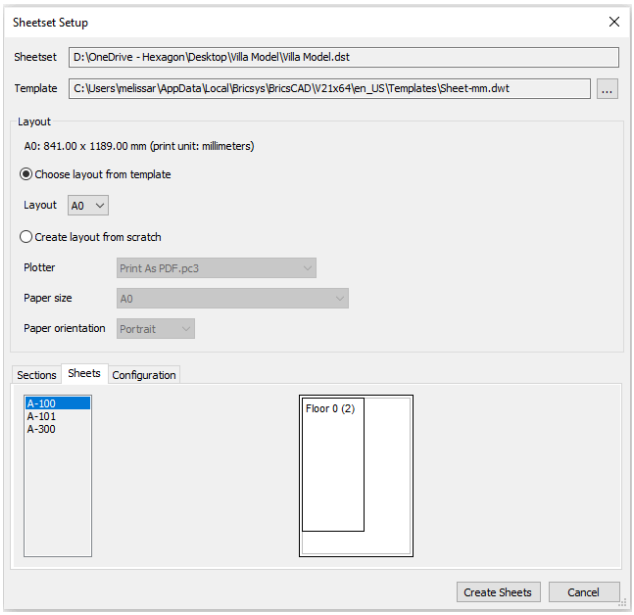
Template: Shows the path of the current template. Click **Browse** (...) to change the template.

Layout: Selects which layout (different paper sizes) the section results will be generated.

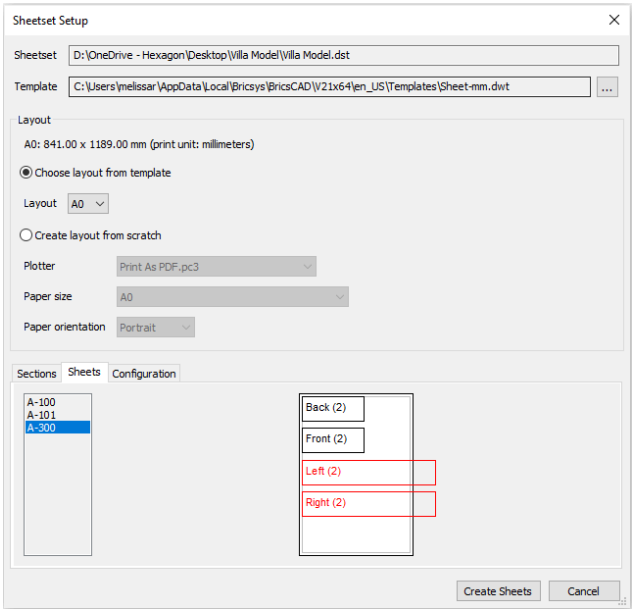
1 The **Sections** tab lists all sections. For each section with an empty **Current Destination** field, a viewport and sheet view are generated.



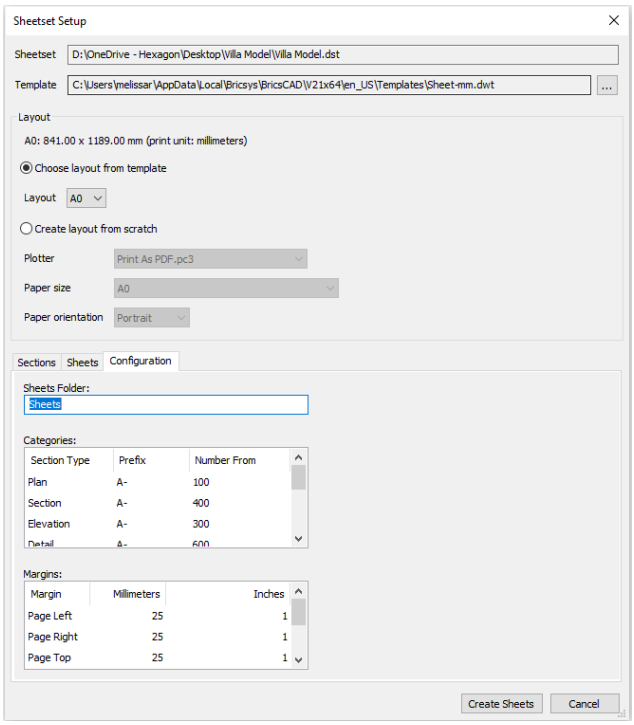
2 Click **Sheets** to preview each sheet (within the selected layout).



Nota: When the viewport is larger than the layout, it displays in red.

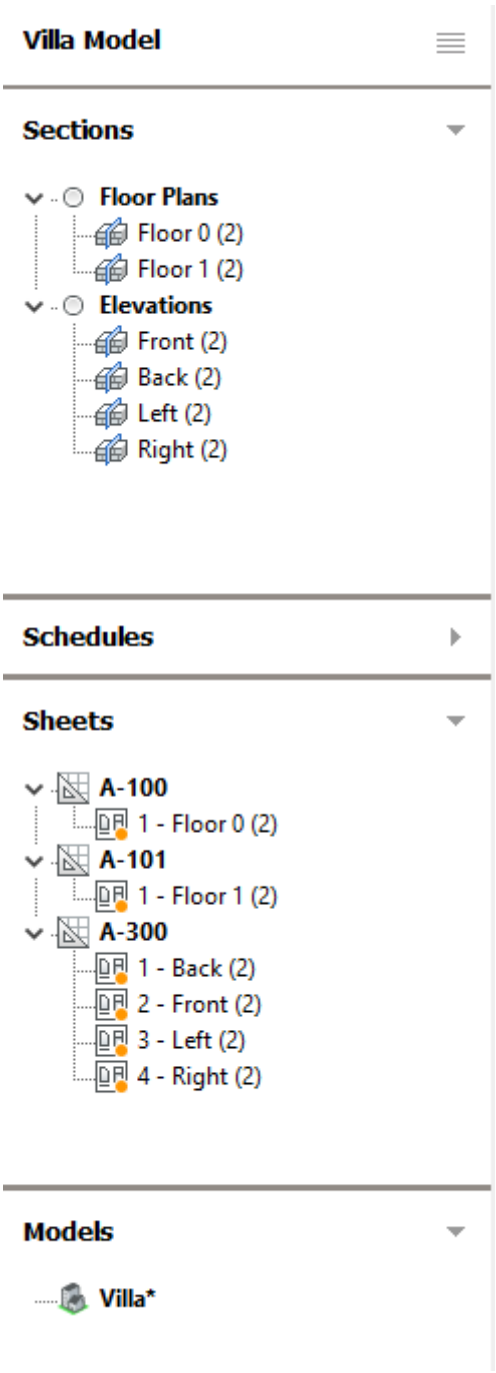


3 Click the **Configuration** tab. Slowly double-click (or click and hit Space/Enter) to change the properties. Under **Sheets Folder** you can change the name of the folder where the sheets will be stored.



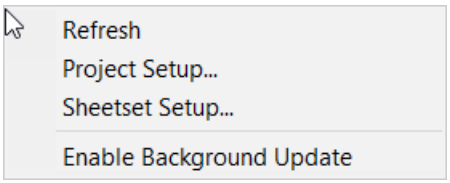
4 Click **Create Sheets**. An overview of the project displays and the sheets are generated.

Nota: The drawings with an orange fill circle icon are out of date. Right-click on them and select **Update** to match changes in the models included in the project.



20.4.16.5 Editing a project

Click on the **Menu** button (☰). A context menu displays:



- **Refresh**: Refreshes the Project Browser to match changes in the model.
- **Project Setup...**: Opens the Project Setup dialog box.
- **Sheetset Setup...**: Opens the Sheetset Setup dialog box.
- **Enable Background Update**: Calculates section results automatically (without interrupting user workflow). Sections are assigned a color; Queued: blue, Outdated: red and In progress: green.

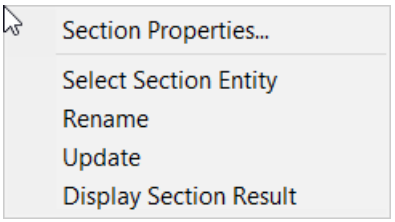
20.4.16.6 Creating schedules

See the Schedules article for more information about Schedules.

20.4.16.7 Working in the project browser

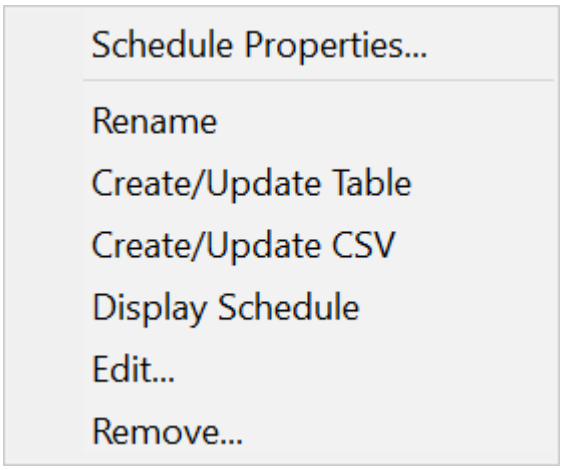
Right-click an item in the BIM **Project Browser** to display a context menu.

Sections



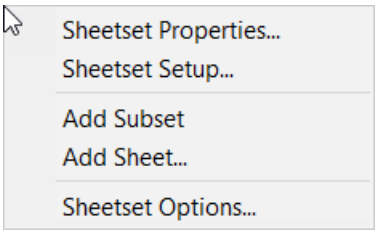
- **Section Properties...**: Opens the **Section Properties** dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: Selects the section entity in the project model drawing.
- **Rename**: Renames the section entity.
- **Update**: Recalculates the section result.
- **Display Section Result**: Displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

Schedules



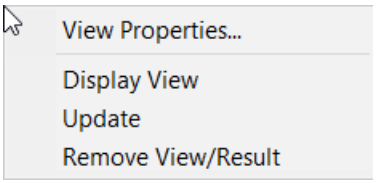
- **Schedule Properties...**: Opens the **Schedule Properties** dialog box that allows you to modify the properties of a schedule.
- **Rename**: Renames the schedule.
- **Create/Update Table**: Creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: Creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: Displays the schedule in the resulting drawing. If not already open, the drawing is opened.
- **Edit**: Opens the **Wizard Page** dialog box that allows you to edit the schedule.
- **Remove...**: Displays the **Remove** dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

Sheets



- **Sheetset Setup...**: Opens the **Sheetset Setup** dialog box that allows you to create a layout from scratch.
- **Add Subset**: Adds a subset in the **Sheets** tab.
- **Add Sheet...**: Adds a new sheet in a subset in the **Sheets** tab.
- **Sheetset Options...**: Opens the **Sheet sets** tab in the **Settings** dialog box that allows you to modify the sheet sets options.

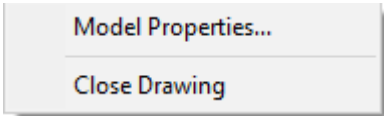
Sheet views



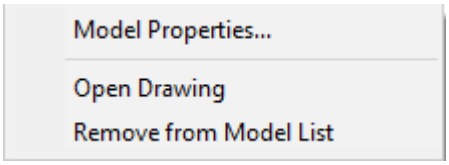
- **View Properties...**: Opens the **Sheet View Properties** dialog box that allows you to modify the properties of a sheet view.
- **Display View**: Displays the corresponding result and zooms in on its viewport.
- **Update**: Updates the sheet view. It executes the BIMSECTIONUPDATE command for the related **BIM** section.
- **Remove View/Result**: Removes the view and result from the corresponding sheet.

Model drawings

Context menu of a currently open model:



Context menu of a currently closed model:



- **Model Properties...**: Opens the **Model Properties** dialog box that allows you to see more information about the model.
- **Close Drawing**: For a currently open model, it closes the model.
- **Open Drawing**: For a currently closed model, it opens the model.
- **Remove from Model List**: Removes the drawing from the project model drawings.

20.4.17 Spatial locations

20.4.17.1 Commands

BIMATTACHSPATIALLOCATION, BIMSPATIALLOCATIONS

20.4.17.2 About

Spatial locations allow you to specify building and its stories in a project. In BricsCAD, a BIM model file can contain one site and multiple buildings and a building can contain multiple stories. When the entities are classified as a building element in the project, these classified entities have a **Building** and a **Story** property. Every building element in the BIM model resides in a particular story within a particular building.



For more information about this command, visit the Command Reference article BIMSPATIALLOCATIONS.

20.4.17.3 Procedure: assigning a building and story to a building element

1 Launch the BIMSPATIALLOCATIONS command.

2 Click the **New Building** icon ( ) to add a new building to the model.

3 Fill out the properties grid to specify a new name or value of the existing property.

4 Click the **New Story** icon ( ) to add a new story in the selected building.

5 Fill out the properties grid to specify a new name or value of the existing property.

6 **(Optional)** Click the **Delete** icon ( ) to remove the selected building or story.

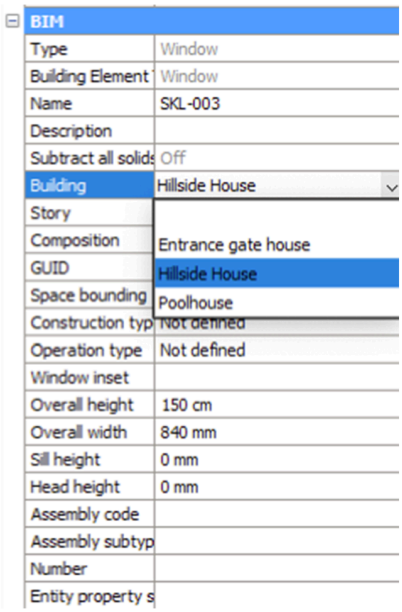
7 **(Optional)** Click **Import Spatial Locations** button ( ) to import a spatial location from a .txt file.

8 Select building element(s) in the drawing area to associate them with the appropriate spatial locations.

9 The properties of the selected building elements display in the **Properties** panel.

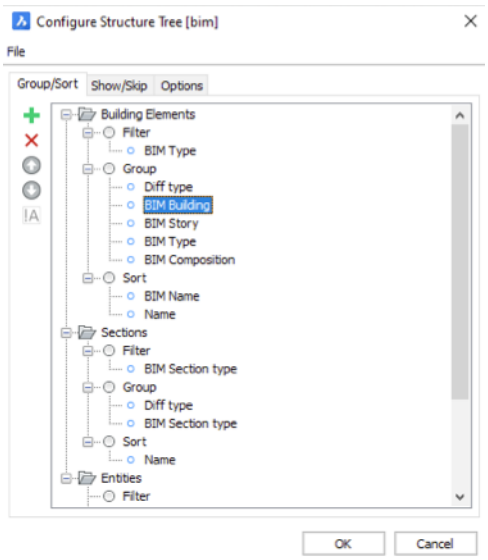
10 Do the following to assign the building and story to these building elements:

- Expand the **BIM** section in the **Properties** Panel.
- Click the **Building** field, then use the arrow to select a building.
- Click the **Story** field, then use the arrow to select a story.



20.4.17.4 Viewing the spatial locations 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. Right click in the **structure browser** and click **Configure**, a new window will open. See the STRUCTUREPANEL command for more info.



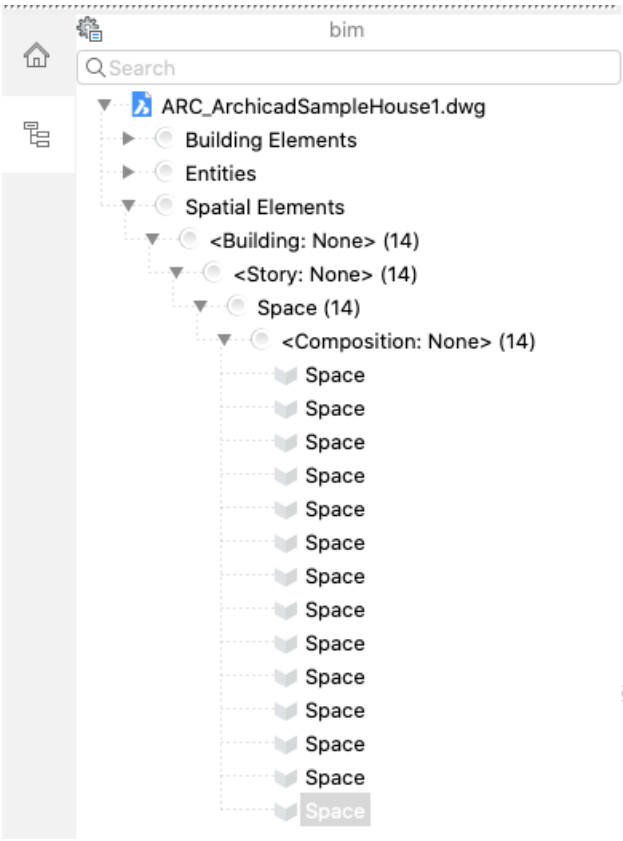
If there is no **BIM Building** or **BIM Story** in the **structure browser**, you can add them manually. See Structure Panel for more info.

20.4.17.5 About BIMATTACHSPATIALLOCATION



Assigning new Spaces can be done using the BIMSPACE command. Spaces are not always assigned to the according buildings because they don't have a spatial location yet. Use the BIMATTACHSPATIALLOCATION command to assign a spatial location to spaces.

For more information about this command, visit the Command Reference article BIMATTACHSPATIALLOCATION.



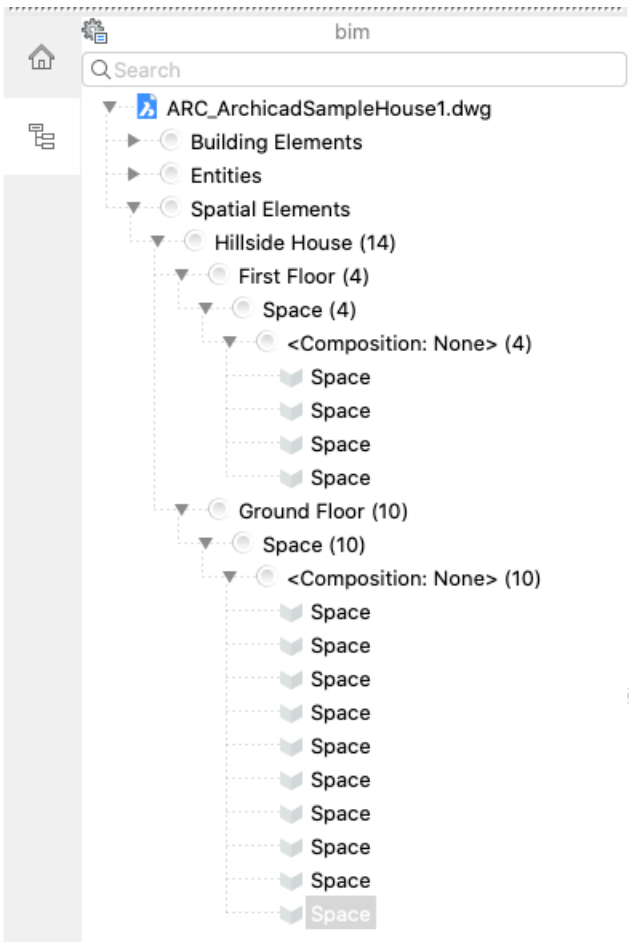
20.4.17.6 Procedure: attaching spatial locations

1 Launch the BIMATTACHSPATIALLOCATIONS command.

2 **You are prompted:***Enter location number or [Auto attach locations/Unattach current location]:* Press **A** to auto attach the spaces to their according building and floor level.

3 Select the spaces to update and press Enter.

4 **(Optional)** Press U to unattach the spatial location.



20.4.18 Structure browser

20.4.18.1 Commands

STRUCTUREPANEL

20.4.18.2 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 line, then press Enter.

20.4.18.3 Procedure: using the structure browser

1 Open the **structure browser**.

By default, the **structure browser** displays on the left-hand side on your screen. With a **search** field at the top of the panel (1).

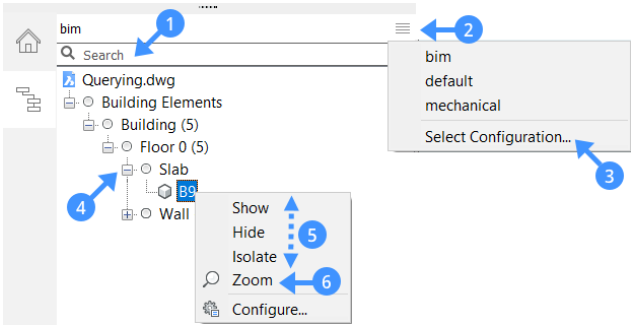
2 Click the horizontal stripes button (2) to load a configuration file in the list. Click **Select Configuration...** to select a .cst file, which specifies a configuration (3). The current configuration displays on the left. Based on the selected file the BIM model will be organized in a tree-structure.

3 Click the expand/collapse icon (4) of a node to expand or collapse the node. Right-click on a group name to select **Expand All** or **Collapse All** and expand or collapse all the subnodes at once.

You can **Show**, **Hide** or **Isolate** the entities, selected in the **structure browser**, in your model space, by right-clicking on the selected names (5).

This context menu also provides you to **Zoom** (6) into a specific entity in the model.

The following illustration shows the **Structure Browser**.



20.4.18.4 Procedure: querying in the structure browser

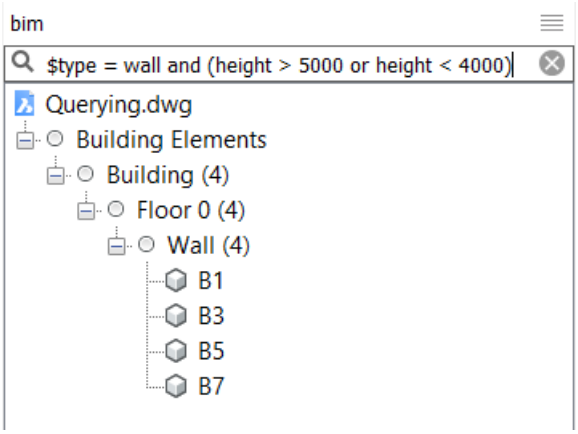
1 Typing something in the **Search** field at the top of the **structure browser** will issue a simple Text search: e.g. if you type 'wall' in the search field and press **Enter**, the **structure browser** will be filtered to everything with 'wall' in its name, including walls and curtain walls.

2 Text searches can be useful for a quick search, but more powerful queries are possible using expressions.

3 An expression starts with the **$** E.g. if you enter *$type = wall* and press **Enter,** the browser will be filtered on entities that have the BIM type = wall. Thus, using this expression, the curtain wall doesn't show up in the browser anymore.

4 Expressions can be more complex, such as filtering on multiple properties. E.g., you can enter *$type = wall and height > 5000* to filter on all walls with a height over 5000 mm.

5 Logical expressions can be nested using brackets: *$type = wall and (height > 5000 or height < 4000)*.



20.4.18.5 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).

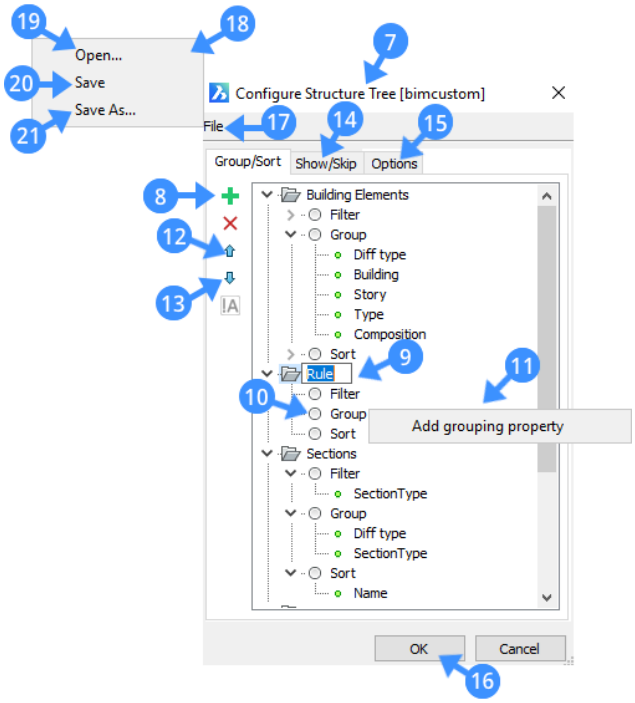
5 Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the **Structure Browser**.

6 Select the **Options** (15) tab to set the entities action in the model using the structure tree. You can here specify whether or not selecting an entity in the **Structure Browser** also selects the entity in the model space.

7 Do one of the following to save the configured tree:

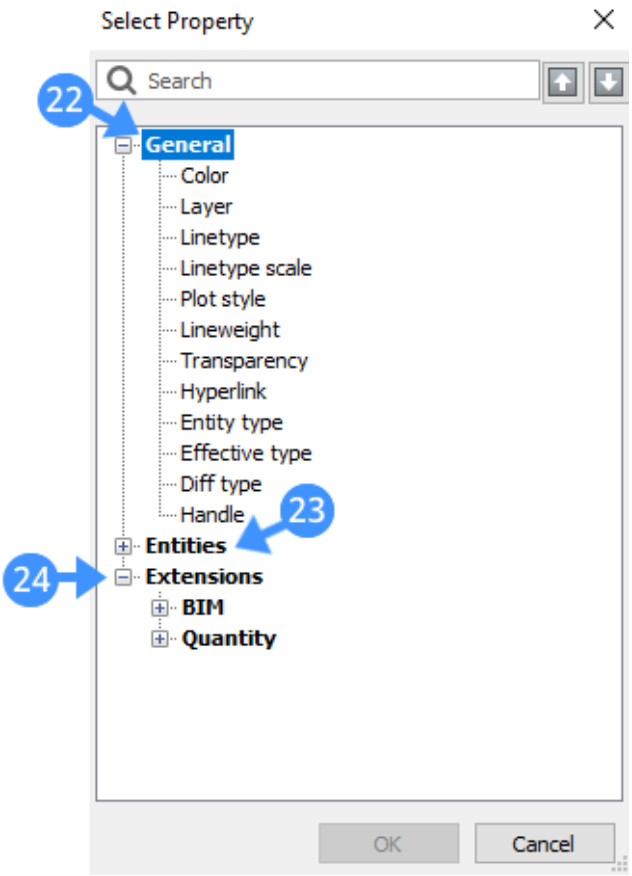
- Click the **OK** (16) button.
- Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

The following illustration shows the **Configure structure tree** dialog box.



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.



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.

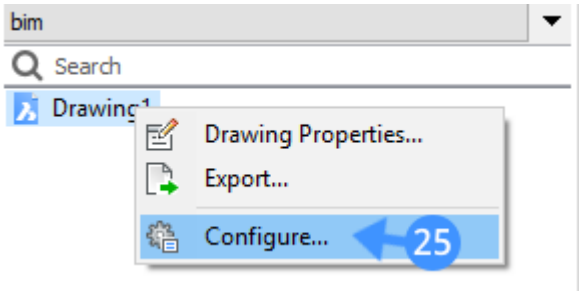
20.4.18.6 Procedure: managing the tree-structure with custom properties

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. Note that custom properties, 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.



The **Configure Structure Tree** dialog box displays.

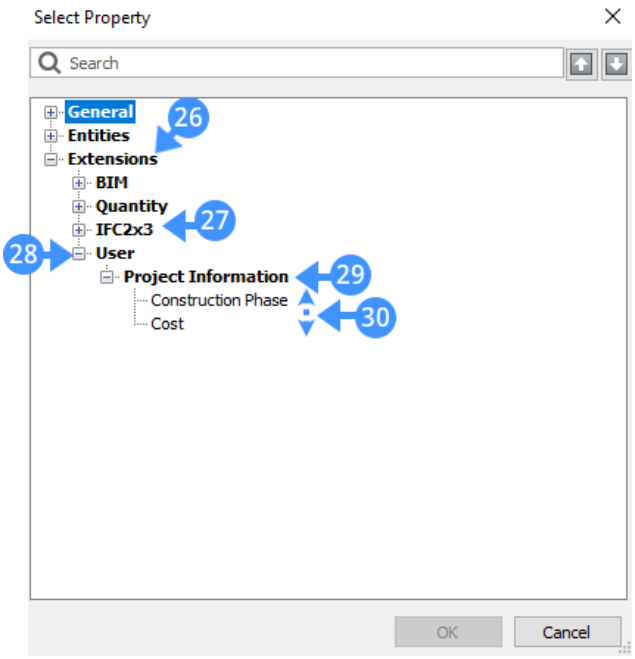
2 Select the **Group** node of the rule, and click the **Add rule or property**(+) icon.

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.

4 The following image shows the **IFC2X3** (27) and **User** (28) namespaces in the **Select Property** dialog.

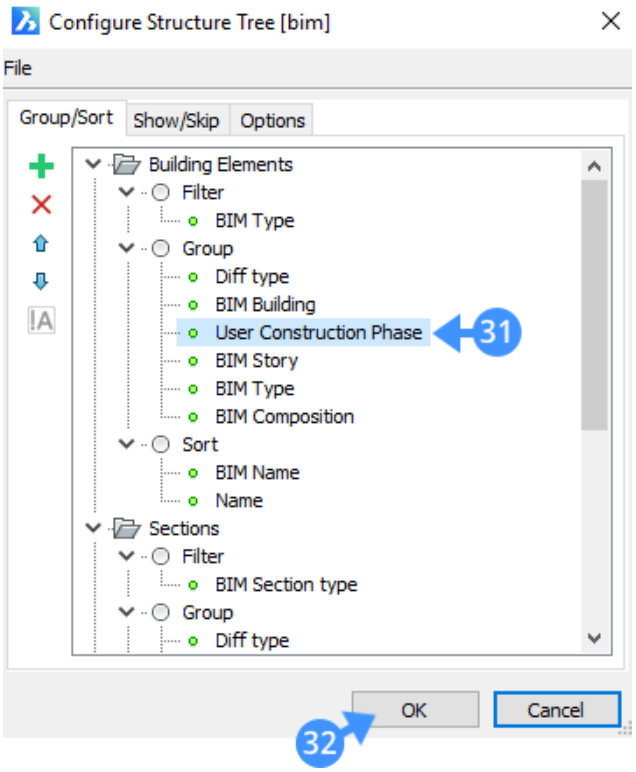
For example, the **User** namespace is expanded to display the property set (29) and properties (30) that are in this node.



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 property for grouping entities in the **Structure Browser**.



20.5 Assistência ao Projeto

20.5.1 Using bimify

20.5.1.1 Commands

BIMIFY

20.5.1.2 About BIMIFY

BIMIFY analyzes a model and will automatically classify spatial locations, spaces, buildings, and stories. It also detects and classifies the external and internal walls.

BIMIFY can automatically detect whether a model is an Architectural, Structural or MEP model. BricsCAD uses this information during the autoclassification process. Define this information before you launch BIMIFY, to improve the accuracy of the automatic classification.

The solids are auto-classified as columns, beams or members. If their profiles match a definition in the profile library, the profile name will be automatically added to the entity's properties as meta-data. If no matching profile definition is found, a new one is created in the library.

BIMIFY can be used in a model that has both classified and unclassified entities. The classified entities are also linked to the appropriate spatial locations.

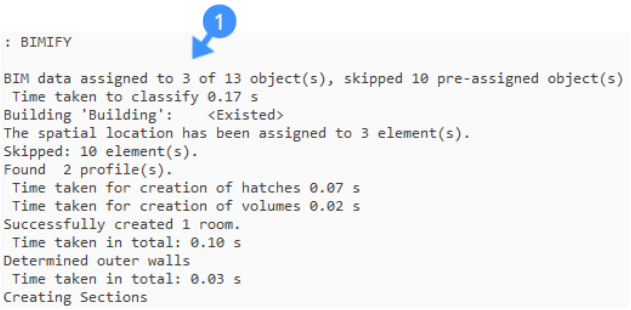


For more information about this command, visit the Command Reference article BIMIFY.

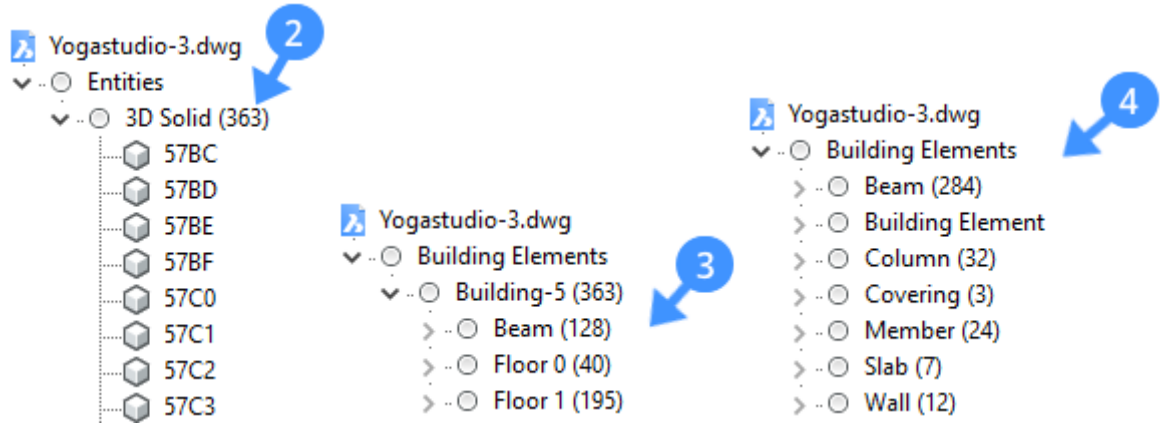
20.5.1.3 Procedure: bimify your entire drawing

Choose BIMIFY from the Quad or type BIMIFY in the Command line and choose **Entire model**.

When the BIMIFY process is completed, an overview of the result (1) is provided in the **Prompt History** panel.



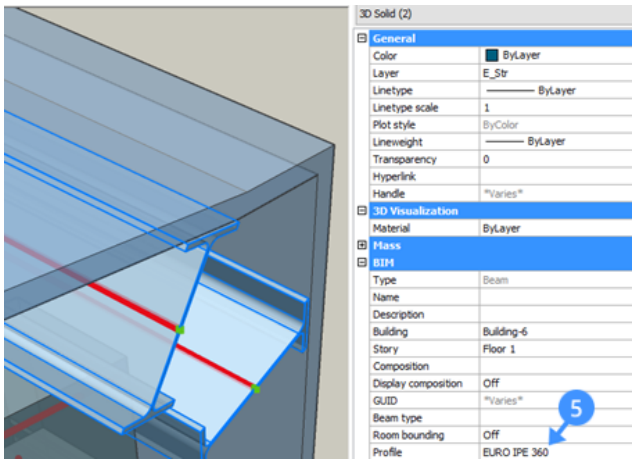
Bimify creates new spatial locations if necessary.



Left to right:

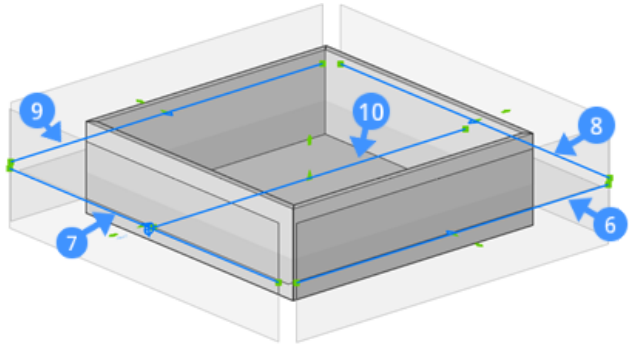
- ***Before Bimify:*** *The structure tree displayed only 3D solids (2)*
- ***After Bimify:*** *The entities are sorted into their corresponding building, story (3) and building element type (4).*

Nota: The **structure browser** can be customized to control the content of the tree.



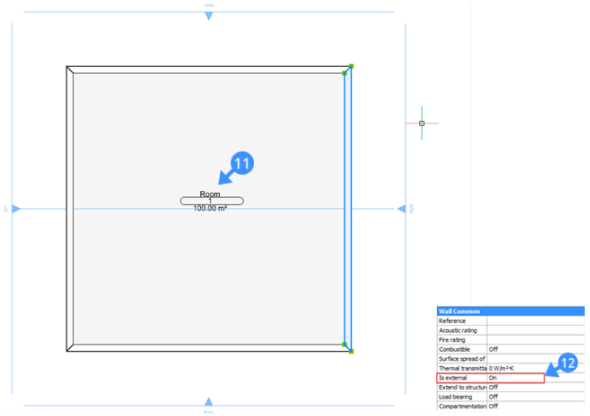
The profile data of a beam in the properties panel (5). BIMIFY recognized the profile of these model elements as structural members and detected the axis location of the profile.

The elevation views and floor plan sections are created last.



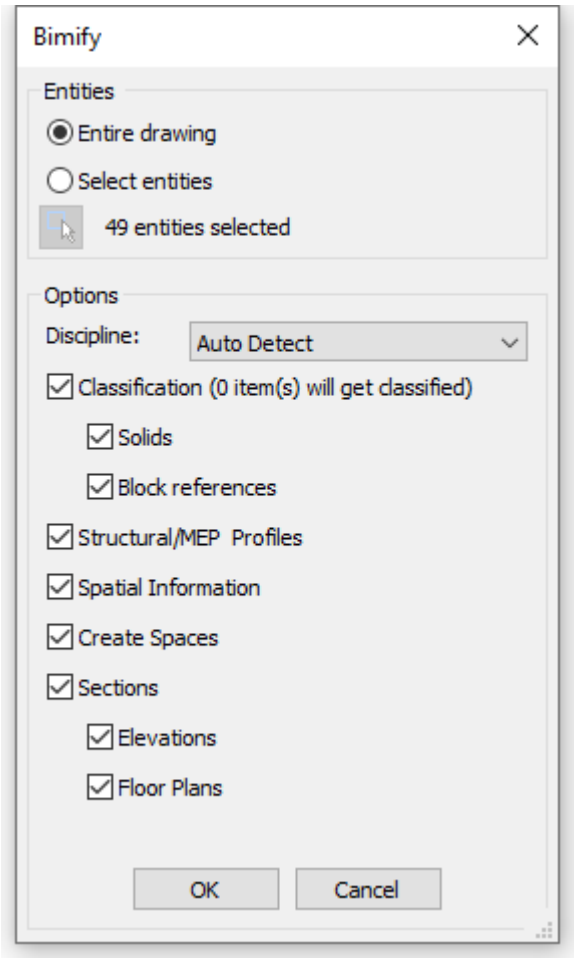
This building contains front (6), Left (7), Right (8) and Back (9) views as well as a floor plan (10).

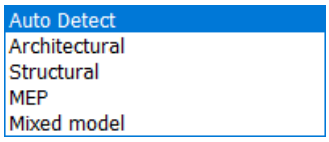
For this simple building, one room and 4 external walls have been detected.



*The BIM room (11) and **Is External** (12) property under the **Wall Common** properties.*

If you choose to use the **Advanced** option of BIMIFY, a dialog box will appear:



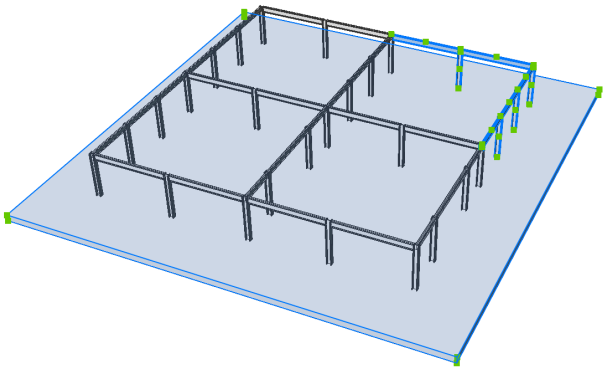
Discipline	 Defines the discipline of the drawing manually. This information helps improve the accuracy of Bimify.
Classification	Classifies **Solids** and **Block References** automatically.
Structural/MEP profiles	Assigns columns, beams, members or flow segments profiles if available in the BricsCAD profile library. This option is available when **Classification** is selected only.
Spatial Information	Assigns spatial locations such as buildings and floors, detects spaces and identifies internal and external walls. This option is available when **Classification** is selected only.

Sections	Creates an elevation view and a plan section per floor. This option is available when **Spatial Information** is selected only.

Nota: To disable the autoclassification option for solids or blocks simply uncheck the related checkbox. By default, all checkboxes are ticked. However, BricsCAD remembers your previous settings. Ticking the Spatial Information checkbox automatically includes the Classification option. This information is needed to detect the other elements.

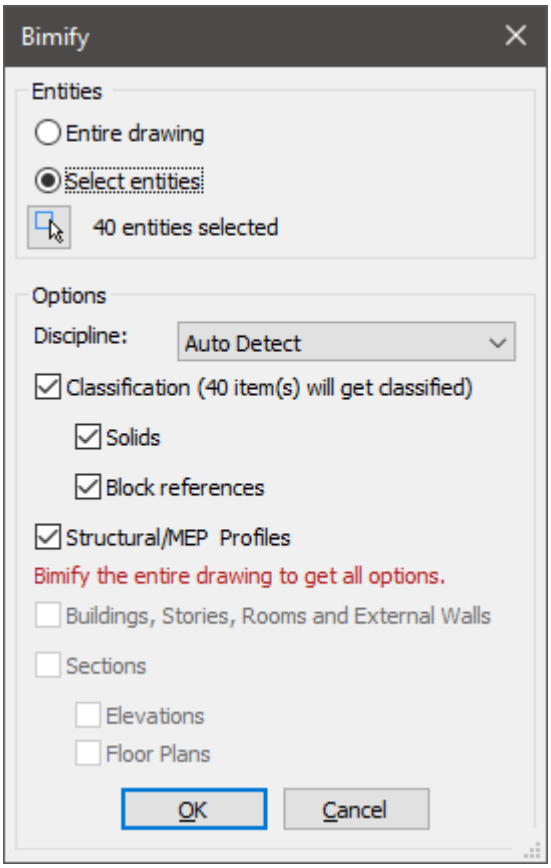
20.5.1.4 Procedure: bimify a selection of your BIM Model

1 Select the part of your drawing you want to Bimify.



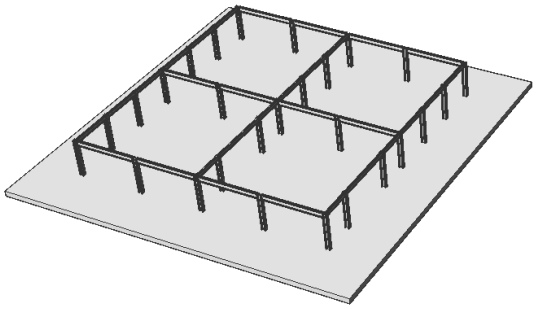
2 Type BIMIFY in the Command line and type in **A** for Advanced.

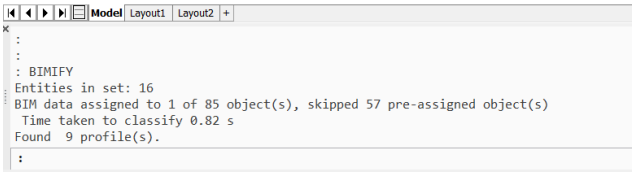
A dialog box displays:



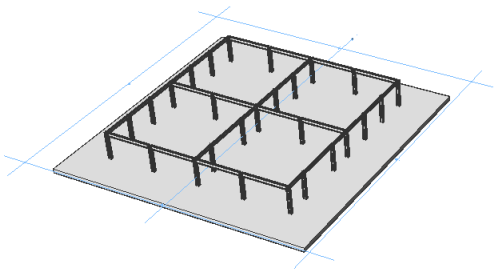
Nota: If you only select a part of your model, only solids, blocks and profiles can be classified. BricsCAD can only detect entities like stories, spaces, and sections when the whole drawing is selected.

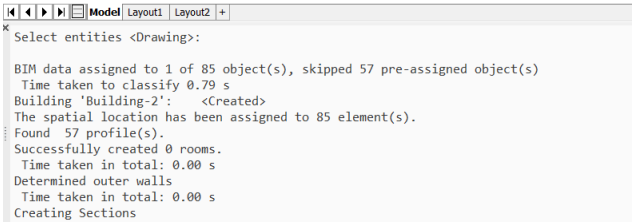
3 When the Bimify process is completed, an overview of the result is provided in the **Prompt History** panel.





BIMIFY classified the selected part of the model. No sections for floor plans or elevations are created.





This is the result if BIMIFY was launched on the whole drawing and all the boxes in the dialog box were checked. The entire model is classified additionally there are sections for the floor plan and the 4 elevations of the model.

20.5.2 Using propagate

20.5.2.1 Commands

BIMPROPAGATE

20.5.2.2 The five flavors

Before getting into the BIMPROPAGATE workflow it is important to note that there are five variants of BIMPROPAGATE, each working on a limited set of situations. These five variants (or flavors) were designed to guide new users through the process of BIMPROPAGATE. 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 the original BIMPROPAGATE command, so every problem you can solve with a particular flavor you can **also** solve using the standard BIMPROPAGATE command. A more advanced user will even be able to solve problems that can not be solved by any of the particular flavors.

The BIMPROPAGATE variants are:

- BIMPROPAGATEPLANAR: Propagation of connections between planar elements such as walls, slabs and roofs.
- BIMPROPAGATELINEAR: Propagation of connections between linear elements such as beams, columns, pipes and ducts, and maximum one planar element.
- BIMPROPAGATEPATTERN: 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.
- BIMPROPAGATEEDGES: Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
- BIMPROPAGATECORNER: Propagates details connected to multiple planar reference solids coming together in one node.

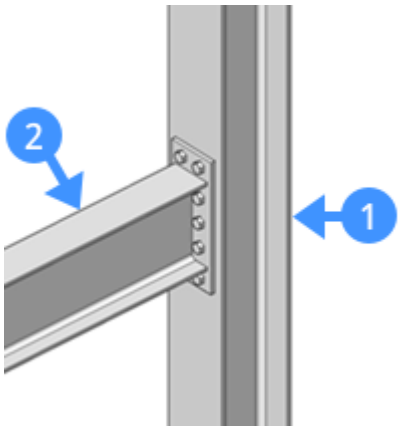
The general workflow of BIMPROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

20.5.2.3 Defining the detail volume

You are prompted:

Select reference solids. These will not be copied, but used to match other locations. [selection options (?)]:

The first step of every propagation is specifying the **reference 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 modeled 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 reference solids in this case will be the column (1) and the beam (2). Later in the process, BIMPROPAGATE scans the model for situations where similar solids occur, and tries to map the detail onto these locations.



Nota: This is an example of a situation that can be solved by BIMPROPAGATELINEAR.

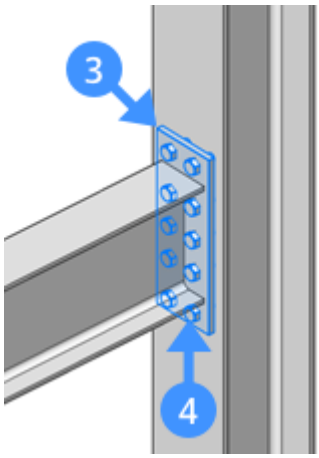
Press **Enter** to stop selecting reference solids.

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 reference solid can be propagated by selecting its faces).

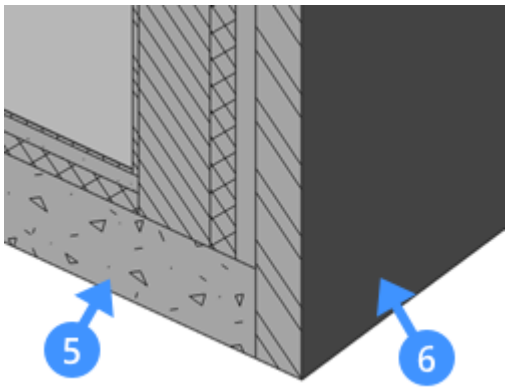
You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

In the example at hand the detail objects will be the end plate (3) and the bolts (4).



Nota: 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 reference 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.



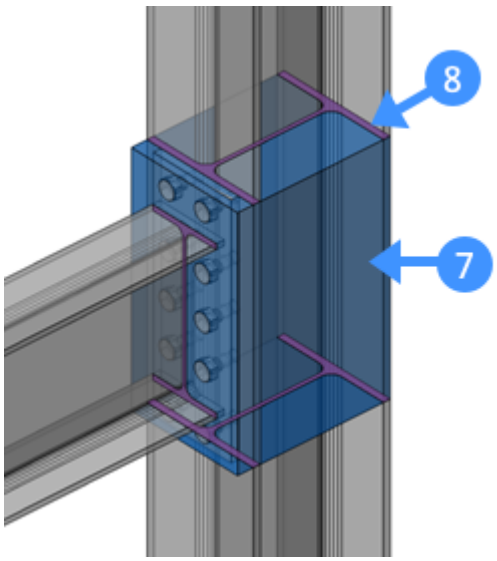
If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects.

Nota: This is an example of a situation that can be solved by BIMPROPAGATEPLANAR.

Once the **reference solids** and the **detail objects** are specified, a detail volume is defined automatically. The visual style 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 reference solid will be copied to the other locations when executing BIMPROPAGATE.

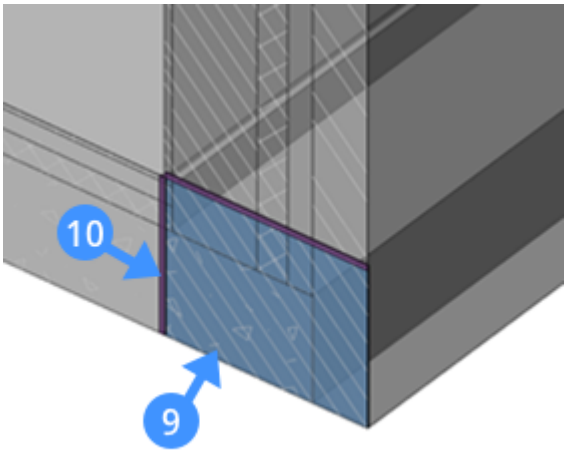
A detail volume can be either a **3D volume** or a **2D slice through a linear detail**.

In our example of the column-beam connection, the detail volume will be as shown in the image below. This is an example of a 3D volume. The connection of the reference solids to the detail volume is highlighted in purple (8).



Something interesting to note: In this particular detail, there are holes in the column where the bolts are bolted into the column. These holes are not necessarily detail objects, but since they are part of a reference solid and they are inside the detail volume, they will be copied as well. Thus, in this respect, BIMPROPAGATE does more than a 'dumb' copy of objects.

The detail volume of our wall-slab connection will look different. As shown in the image below, the detail (9) will be planar, and displayed in a section view. This is because the detail can be propagated as an extrusion of this 2D detail slice. Here as well, the locations where the reference solids connect to the detail volume are highlighted in purple (10).



Nota: The detail of BIMPROPAGATEPLANAR and BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

You are prompted:

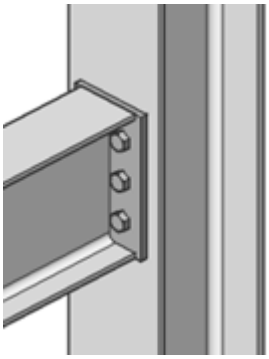
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

You are prompted: Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>:

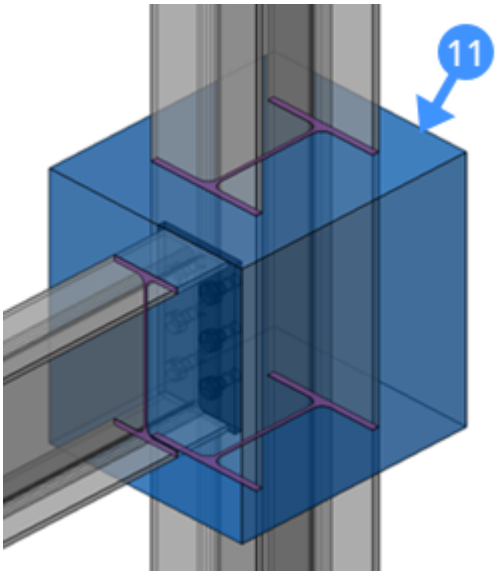
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Usually, the detail volume as it is proposed by BIMPropagate will be sufficient for what the user is trying to achieve. However, it is possible to inflate the detail to include a larger volume. This feature has two particular use cases:

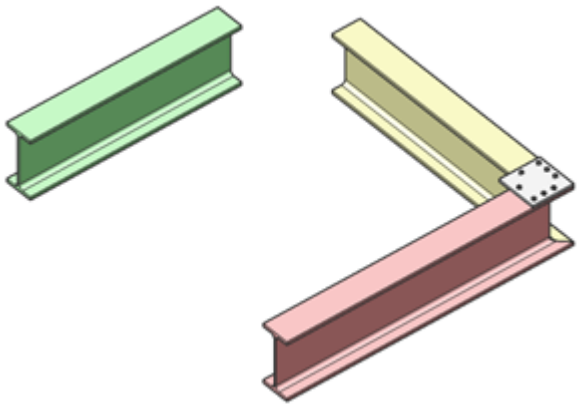
- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following:

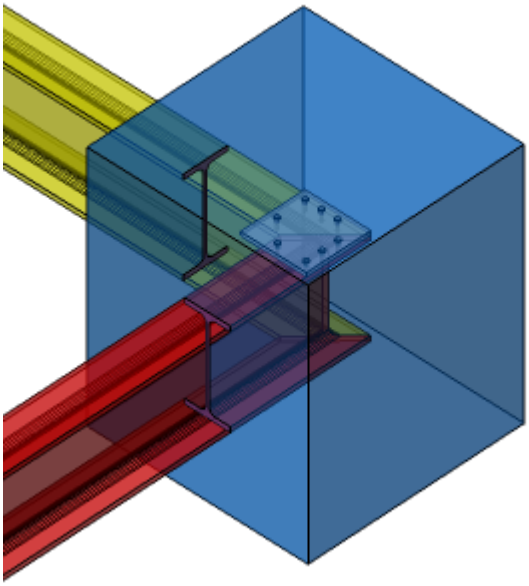
  In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail:

- You might want to include a larger part of the reference 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** beam have a similar connection.



To achieve this, we need to inflate the detail volume so that it overlaps the end faces of the red and the yellow beam.



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**.

20.5.2.4 Choosing where to apply the detail

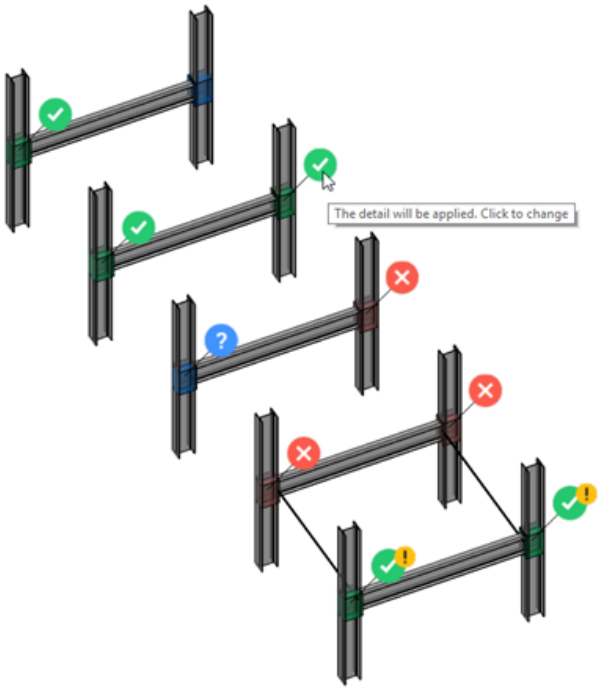
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

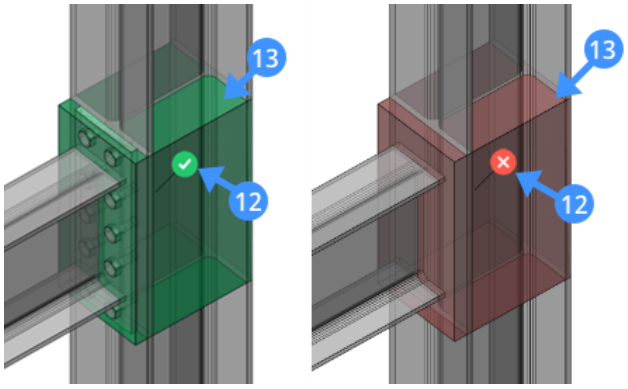
In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

	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.



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).



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 currently selected:**

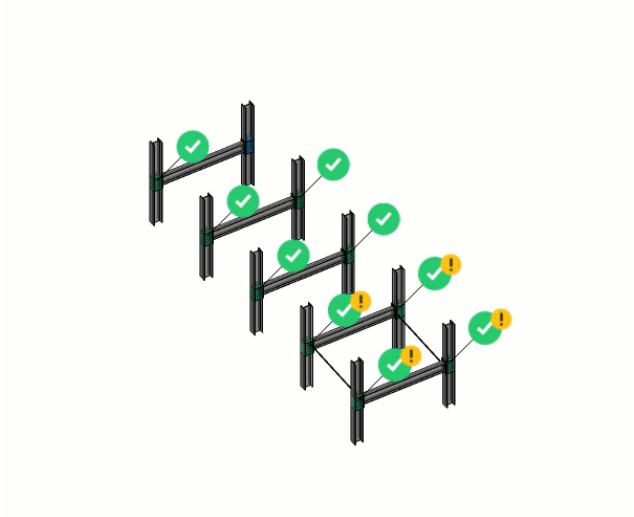
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

This procedure can be used to gradually solve all the suggestions.

In the video below, the following steps are taken:

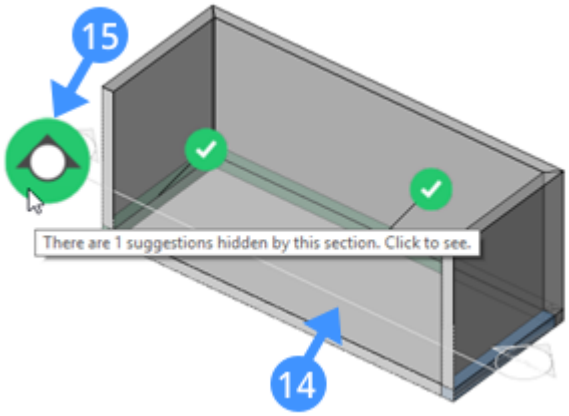
- Four widgets are selected and toggled off
- Enter is pressed: the four selected suggestions were all toggled off, so they are not applied, and the four suggestion are removed from the view
- Four more suggestions are selected
- Enter is pressed: the four selected suggestions are applied, and removed from the view
- There is one suggestion left; Enter is pressed to accept this suggestion, and the BIMPROPAGATE command is terminated.



It is important to emphasize that BIMPROPAGATE only processes entities that are currently visible. As a result, hidden entities, or entities on a hidden or frozen layer are not included when BIMPROPAGATE is scanning the model for locations where the detail can be applied. This can be a useful when executing BIMPROPAGATE in a large model; e.g. when you want to propagate a steel connection, only turn on those layers that contain steel members. This will improve the propagation performance.

It is possible to activate or move existing section planes during BIMPROPAGATE.

- A section plane (14) can be activated by double-clicking it.
- Left-click it once to move it.
- When a section plane is activated, a widget (15) might appear next to it indicating that there is one or more suggestion currently hidden because of the active section. Click the widget to deactivate this section and show the hidden suggestions.



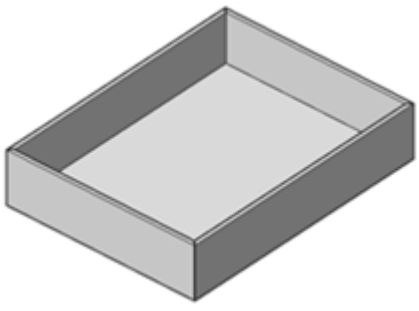
20.5.3 Propagate planar

20.5.3.1 Commands

BIMPROPAGATEPLANAR

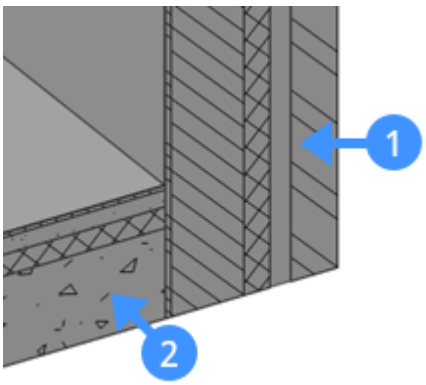
20.5.3.2 Procedure: propagating a wall-slab connection

1 Consider the following situation of four walls and one slab, created with **QuickDraw**.

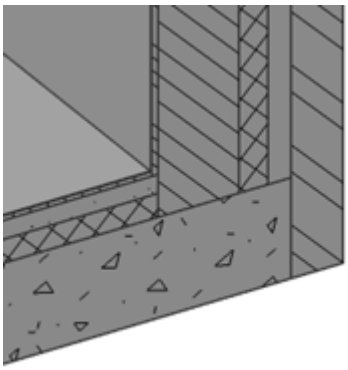


2 The four walls were given a composition, as well as the floor slab. Without any further editing or modeling, creating a vertical section through the building will show the connection one of the walls (1)

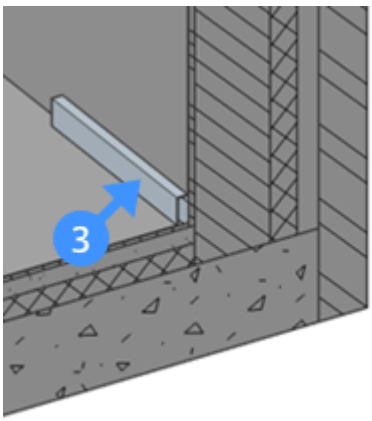
and the slab (2) to be something similar to the following image.



3 While this section is active, we can **Push/Pull** the different ply faces to obtain the desired connection between the wall and the slab:



4 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**.



5 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.

6 **You are prompted:**

Select planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

Select the wall and slab, and press Enter.

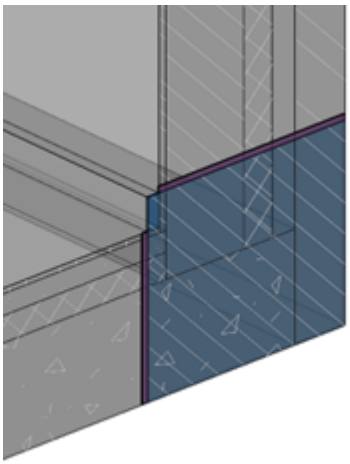
7 **You are prompted:**

Optional. Select detail objects to be copied as part of the connection. [selection options (?)]:

Select the skirting board solid, and press Enter.

8 **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 modeled 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.



9 **You are prompted:**

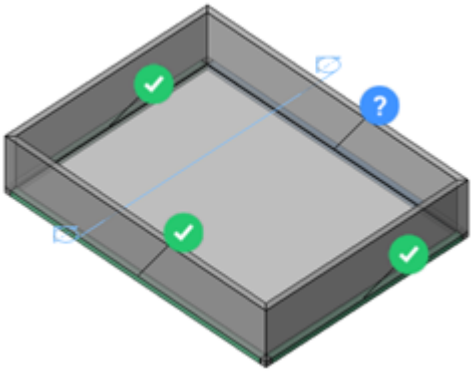
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

10 **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:

You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green checkmark 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 modeled 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 checkmark.

11 Press **Enter** to complete the command.

Now, all four walls should have this particular connection with the floor slab, including the skirting board.

20.5.4 Propagate linear

20.5.4.1 Commands

BIMPROPAGATELINEAR

20.5.4.2 About

BIMPROPAGATELINEAR is one of the five Propagate flavors. 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 the Using Propagate article.

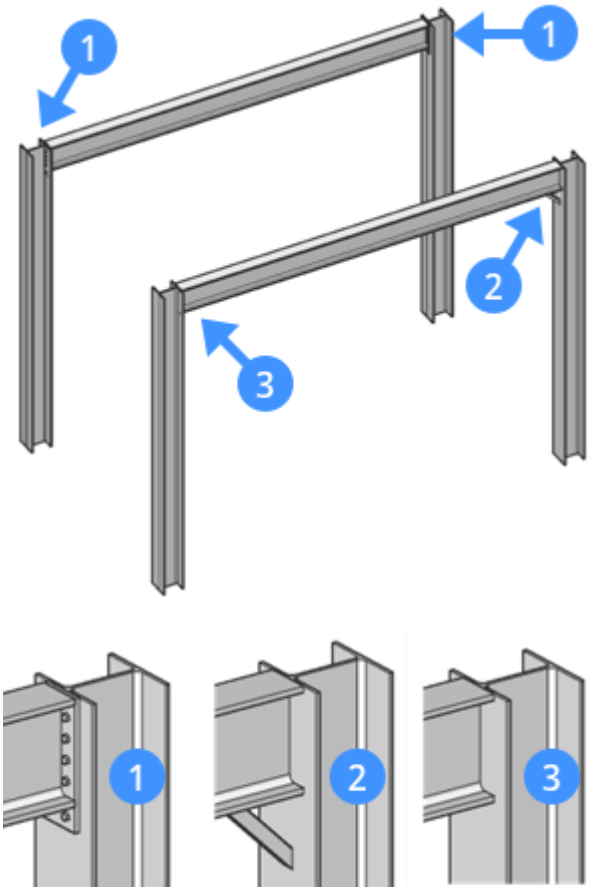


For more information about this command, visit the Command Reference article BIMPROPAGATELINEAR.

20.5.4.3 Procedure: propagating a column-beam connection

1 Consider the following situation of four columns and two beams. Two of the connections have been designed with a plate and bolts through 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).



2 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.

3 We will launch BIMPROPAGATELINEAR.

4 **You are prompted:**

Select the linear or planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

At least one should be linear.

Select the column and the beam that are connected by the diagonal member, and press **Enter.**

5 **You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the connection. [selection options (?)]:

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.

6 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain

Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

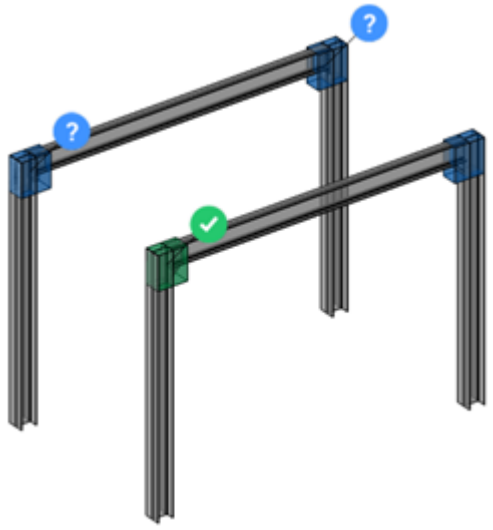
You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the Using Propagate article. Choosing **No** will exit the command.

Press **Enter** to accept.

7 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:

8 **You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



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.

9 If we now press **Enter**, all connections should be replaced by the diagonal member.

20.5.5 Propagate pattern

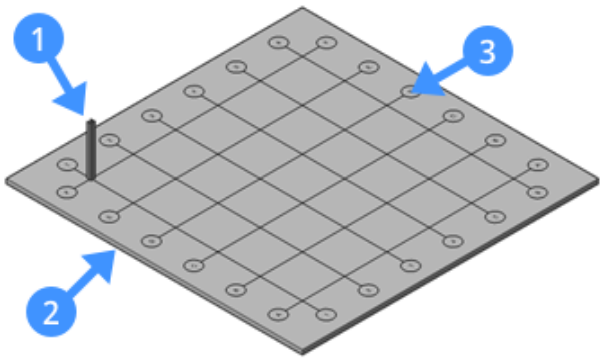
20.5.5.1 Commands

BIMPROPAGATEPATTERN

20.5.5.2 Procedure: propagating columns on top of a slab

1 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).



2 We want to place this column on specific axis intersections. We could manually copy the column but, Propagate allows us to do this faster.

3 Let's launch BIMPROPAGATEPATTERN.

4 **You are prompted:**

Select planar reference solid to which the detail is related. The detail will be copied to similar solids. [selection options (?)]:

Select the slab, and press **Enter.**

5 **You are prompted:**

Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

Select the column, and press **Enter.**

6 A 3D detail volume is created.

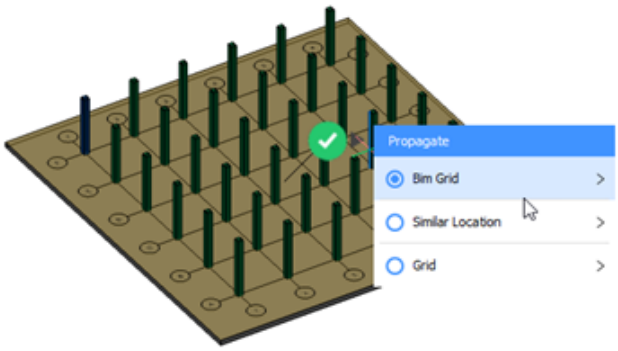
7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command.

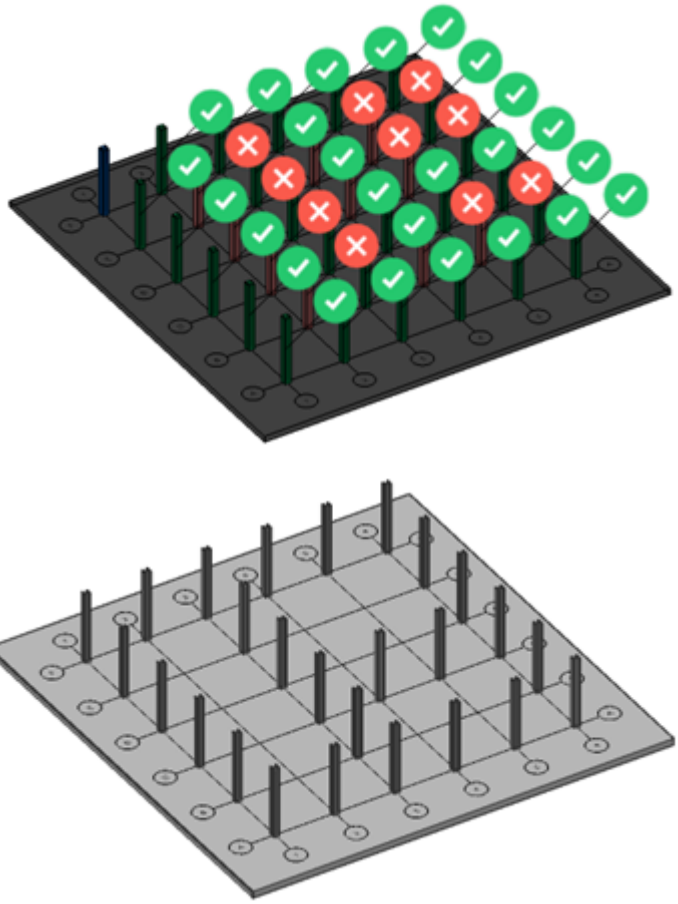
Press **Enter** to accept.

8 A widget appears. Hovering over this widget reveals a flyout menu with several options:



- **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.

9 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:



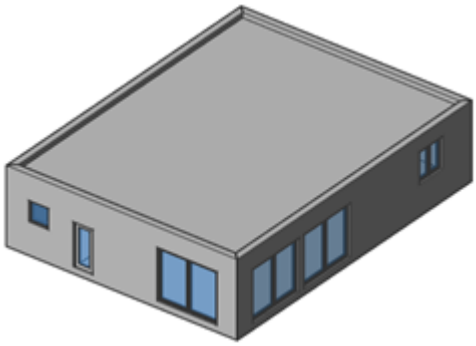
20.5.6 Propagate edges

20.5.6.1 Commands

BIMPROPAGATEEDGES

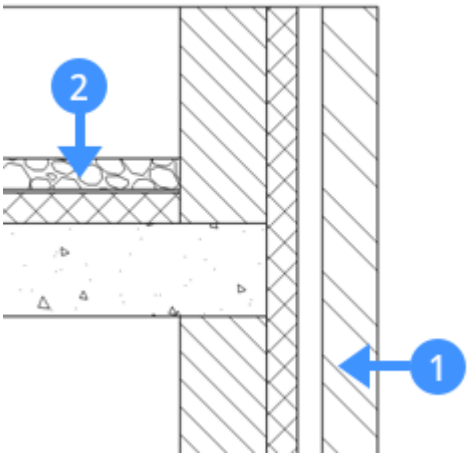
20.5.6.2 Procedure: propagating a wall cap

1 Consider the following situation of a one-story building with a flat roof.

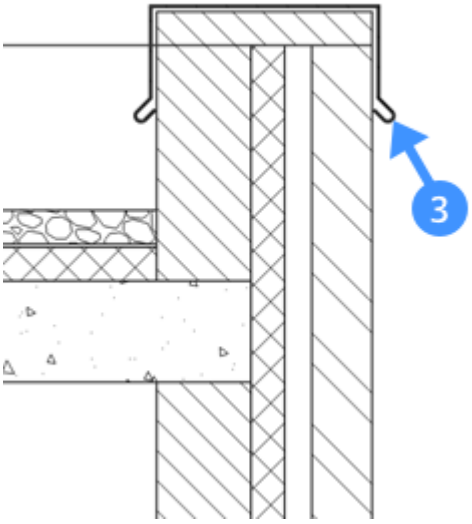


2 Let's create a vertical section plane to cut through our building and take a look at where the wall (1)

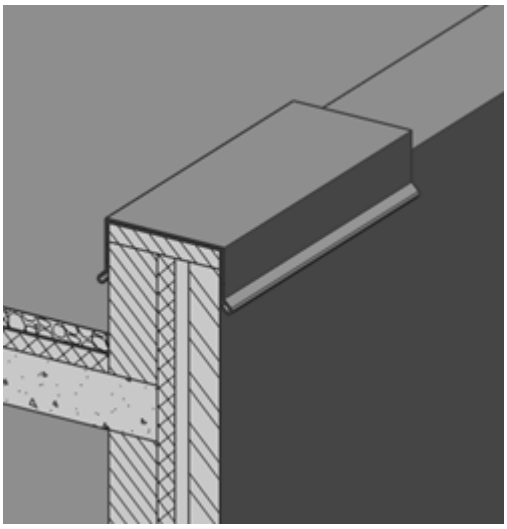
and the roof slab (2) come together.



3 We want to create a capping (3) on top of this wall to get the following result:



We can start by creating this locally on top of one wall, either by using direct modeling tools or inserting a block from somewhere else and exploding it. In our section it might look like this:



4 ow we can use Propagate to copy this detail over to all applicable locations. Let's deactivate the section and launch BIMPROPAGATEEDGES.

5 **You are prompted:**

Select planar reference solid to which the edge detail is related. The detail will be copied to edges of similar solids. [selection options (?)]:

Select the wall on top of which the capping was modeled, and press **Enter.**

6 **You are prompted:**

Select detail objects to be copied. These should be solids or faces that are parallel to an edge of the reference solid. [selection options (?)]:

Select the objects that make up the capping, and press **Enter.**

7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

8 **You are prompted:**

Propagated 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.

9 **You are prompted:**

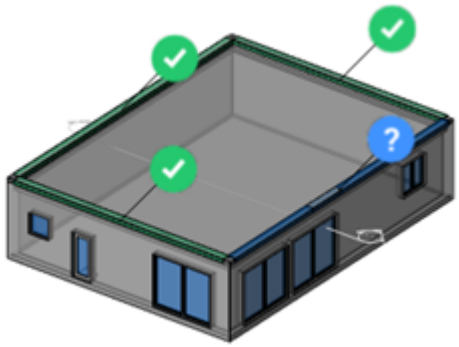
Choose location of detail relative to edge: [Outside reference solid/Inside reference solid] <Outside reference solid>:

This option controls whether the detail will be placed inside or outside (on top of) the base solids. When placed inside, the detail is subtracted from the base solids.

10 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:

11 **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 modeled 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.

12 Press **Enter** to complete the command.

Now, all four walls have this wall cap and the corner connections are created automatically.

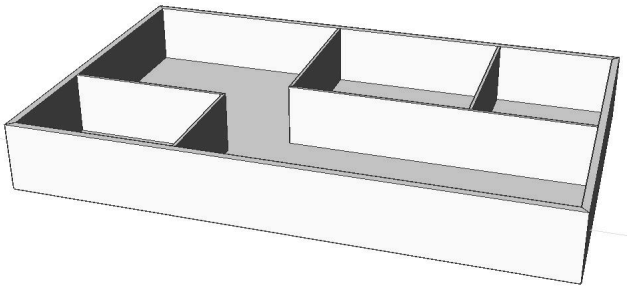
20.5.7 Propagate corner

20.5.7.1 Commands

BIMPROPAGATECORNER

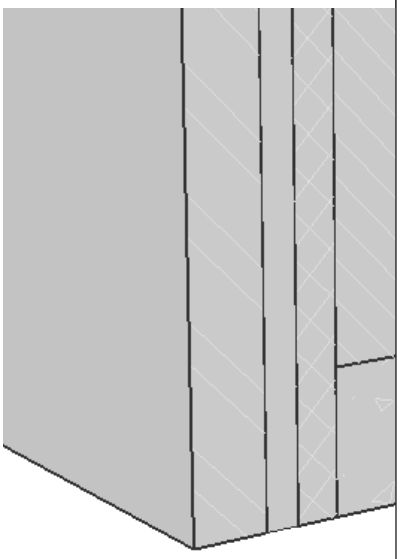
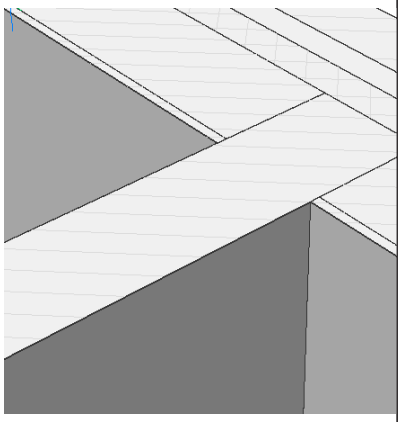
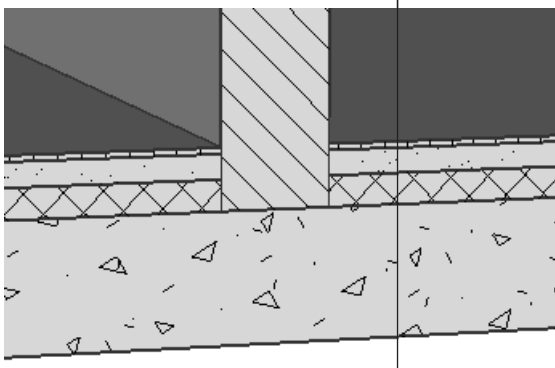
20.5.7.2 Setting the foundations

In this example the 4 exterior walls, 5 interior walls, and floor slab have already been assigned a composition.

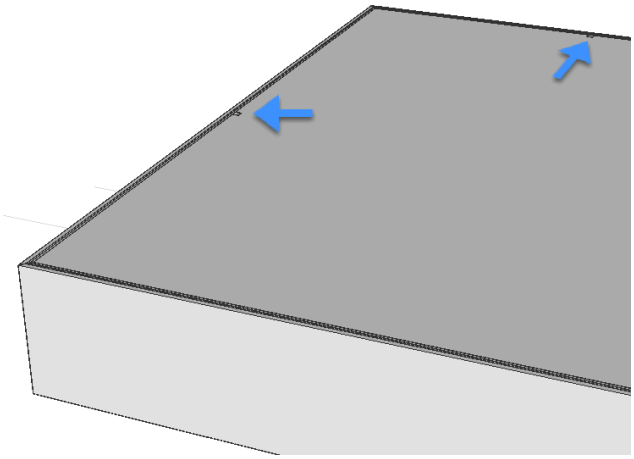


First, you must connect the 3 planar types of connections separately. For a detailed explanation of how to do this, read the article Propagate planar.

You should now have the following connections:

		
Exterior wall - Floor slab	Exterior wall - Interior wall	Interior wall - Floor slab

The nodes where the exterior wall, interior wall, and floor slab come together can't be correctly propagated using BIMPROPAGATEPLANAR. Instead, use BIMPROPAGATECORNER. BIMPROPAGATECORNER can propagate details connected to three planar base solids (i.e. corners).



20.5.7.3 Propagate corner

1 Launch BIMPROPAGATECORNER.

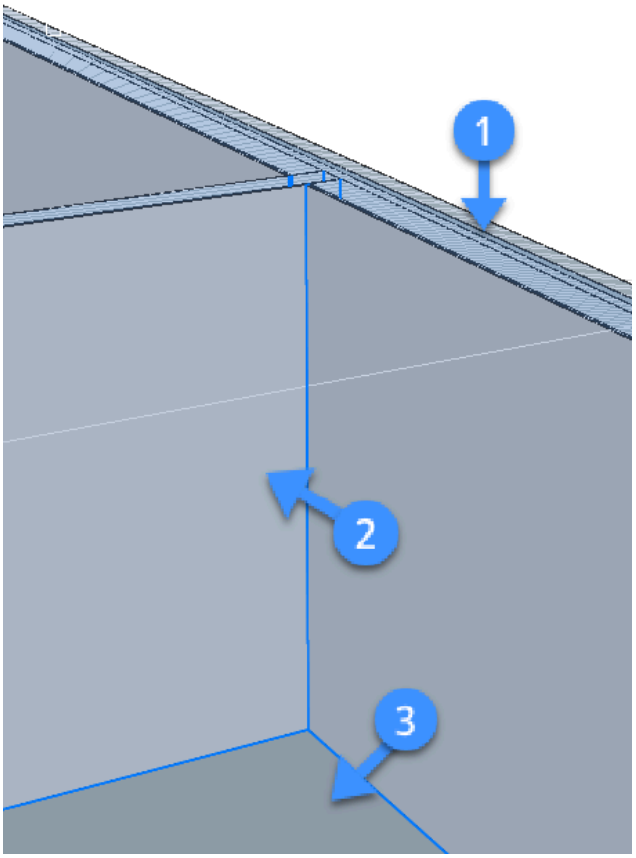
1 **You are prompted:**

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar

corners. [selection options (?)]:

2 Select the exterior wall (1), the interior wall (2), and the floor slab (3).

3 Press **Enter** to accept.



You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the corner detail. [selection options (?)]:

If you have extra solids in your detail, i.e. an insulation block, you can select this as an extra detail object and it will be copied to the other nodes when using BIMPROPAGATECORNER.

4 Press **Enter** when ready.

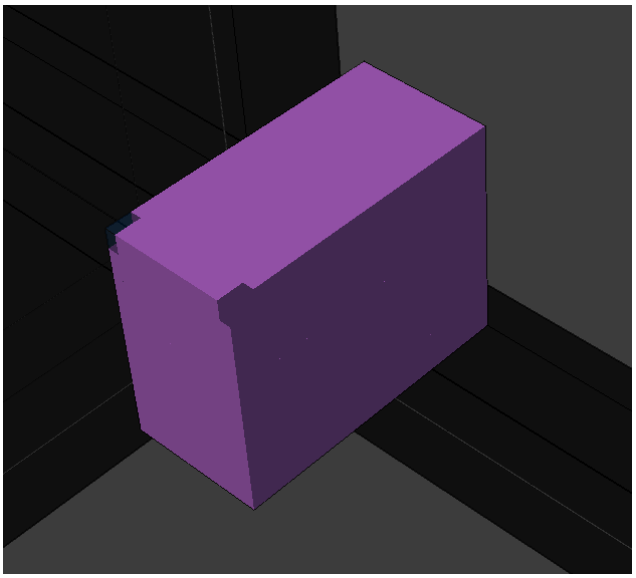
Nota: If you don't have any extra details, just press **Enter** to continue.

5 The volume that will be propagated is encased in a purple box.

6 **You are prompted:**

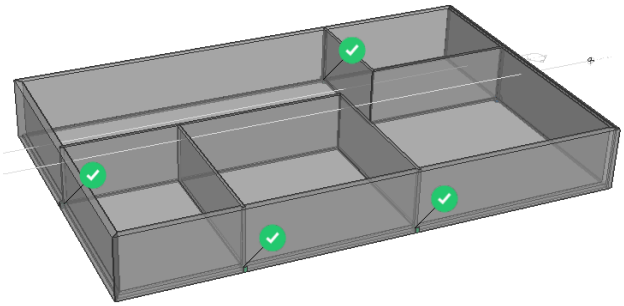
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

7 Press **Enter** to accept the detail.



Nota: BIMPROPAGATECORNER shows a 3D volume instead of a 2D section.

8 BIMPROPAGATECORNER 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. A green checkmark means that this suggestion will be applied when pressing **Enter**.



You are prompted: Choose next action [Apply all/Cancel] <Apply all>:

9 Press **Enter** to complete the command.

The nodes where the exterior wall, interior wall, and floor slab come together should now be propagated correctly.

20.5.8 Parametrize details

20.5.8.1 Commands

BIMPARAMETRIZEDETAIL

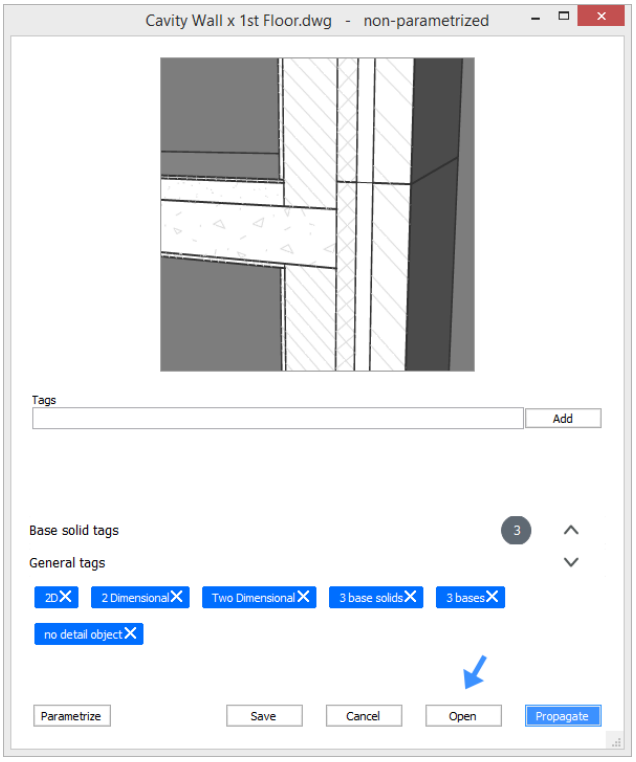
20.5.8.2 About BIMPARAMETRIZEDETAIL

Use the BIMPARAMETRIZEDETAIL command to automatically generate parameters for a saved detail. Standard parameters such as angles, offsets in different directions and tolerance parameters are created. These parameters mean you can propagate details to target situations similar to the example below.

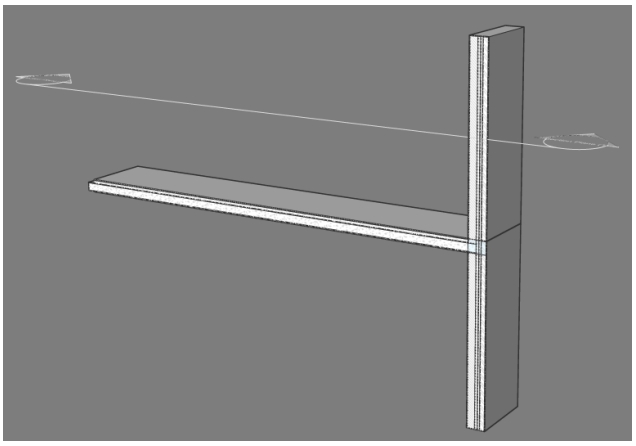
The automatically generated parameters form a framework that can control the position and shape of the reference solids. You can extend this framework adding custom constraints. You can evaluate the effect of these changes by animating the parameters from the **Mechanical Browser**. You can add custom constraints to link detail objects to the reference solids and to each other. You can also use the framework parameters in the expression of those constraints.

20.5.8.3 Parametrize and edit a wall-slab connection detail

1 To open the .dwg file of the detail you want to parametrize click on the detail in the **Details** panel. The **Detail** dialog box displays:



2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL. The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**.

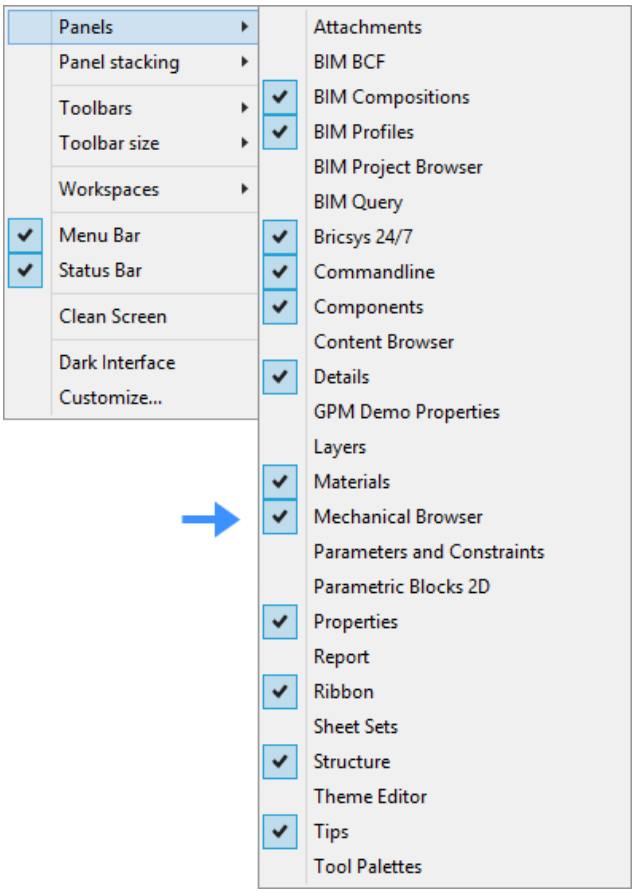
4 Click the **Mechanical Browser** button on the right of the screen.



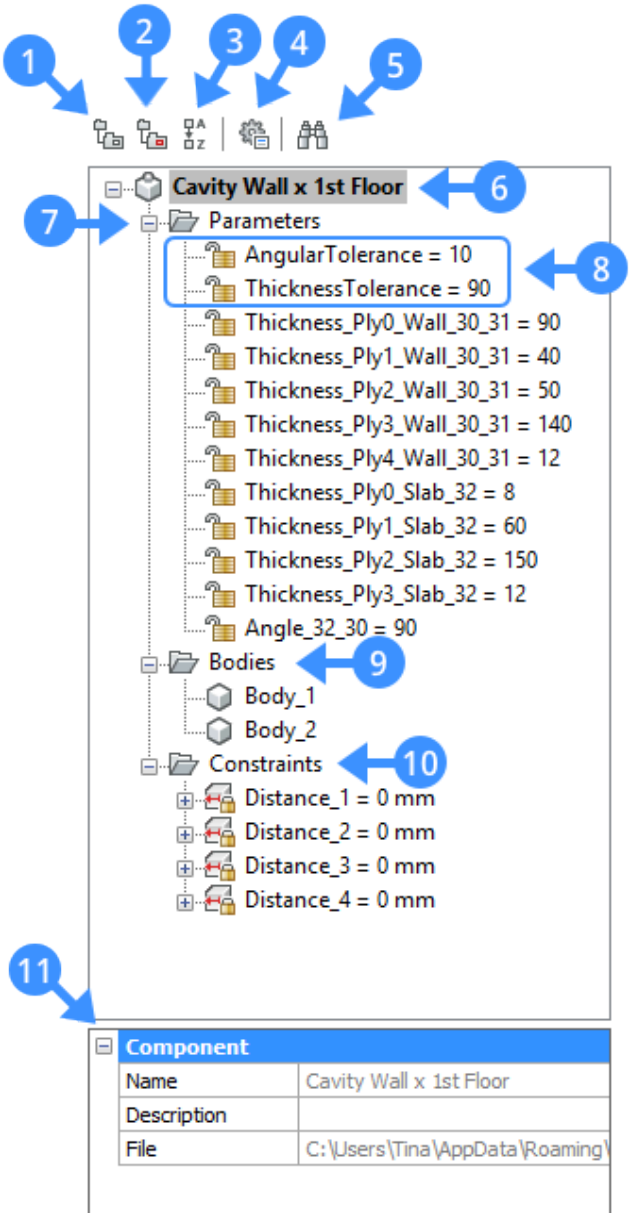
5 If the **Mechanical Browser** button isn't visible yet, move the cursor to the toolbar or ribbon panel and right-click. A context menu displays.

Expand the **Panels** option.

Select **Mechanical Browser** in the context menu.



6 Once you have clicked on the **Mechanical Browser** button, the **Mechanical Browser** displays:



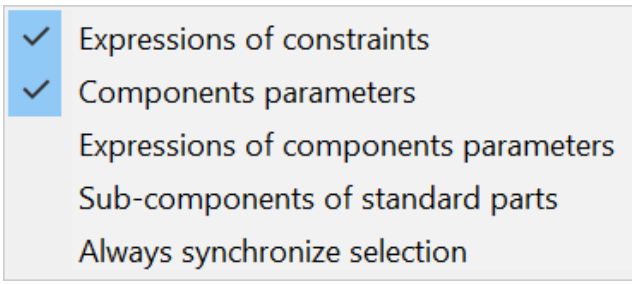
Nota: You can also use the MECHANICALBROWSEROPEN command.

(1) **Group by entity**: Group the parameters and constrains by entity.

(2) **Group by type**: Group the parameters and constrains by type.

(3) **Sort**: Sort alphabetically.

(4) **Settings**: An options menu will display:

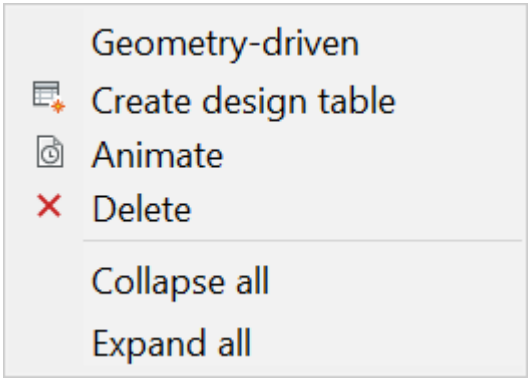


- **Expressions of constraints**:Controls whether the numeric value or the assigned parameter name displays.
- **Components parameters**:Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.
- **Expressions of components parameters**:Sub-component parameters control the visibility of sub-component parameters.
- **Sub-components of standard parts**:Parameters at properties add a parameters section for the selected instance to the **Mechanical Browser** properties.
- **Always synchronize selection**:When checked,theselectednodes in the browser mirror the selection in the document.

(5) **Show search** (Ctrl+F): Show or hides the search bar.

(6) The name of the parametrized detail.

(7) **Parameters**: Right-click on a parameter to open a context menu:



- **Geometry-driven**:If checked, it makes the parametergeometry-driven.
- **Create design table**: Creates a design table to drive parametric block parameters.
- **Animate**:Animates models by means of parameters.
- **Delete**: Deletes the selected parameter.
- **Collapse all**: Collapse the whole **Mechanical Browser**.
- **Expand all**: Expand the whole **Mechanical Browser**.

(8) **Tolerance Parameters**: The maximum and minimum variation of an angle, thickness or dimension from the original parameter value.

(9) **Bodies**: Display the bodies in the model.

(10) **Constraints**: The constraints you can delete, modify or add.

(11) **Component**: The properties of the detail file.

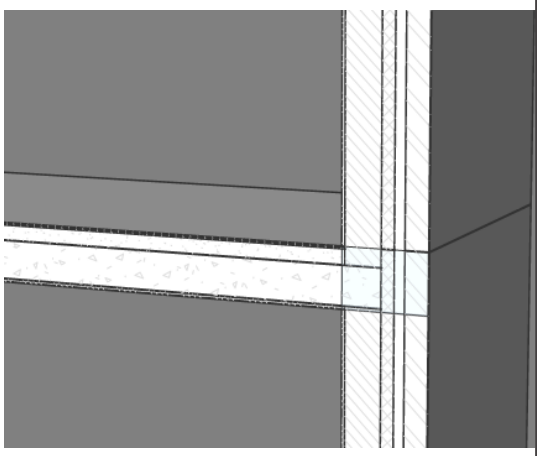
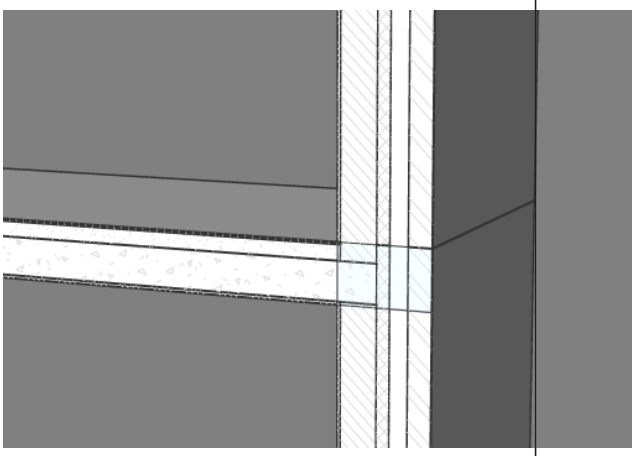
7 ***Nota:*** Some default constraints are generated that link the detail volume and the detail objects to the reference solids. You can delete and replace them with other constraints.

8 Edit the parameters.

9 The parameters that were generated have specific values to ensure the detail looks exactly as it was created. If you change these values, the detail will change.

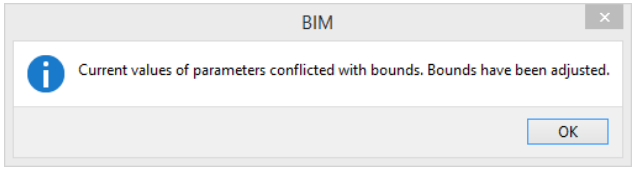
For example:

- Change the **Thickness_Ply1_Wall_30_31** parameter from 40 to 70 (in the **Expression** field). The new detail is visibly different in the model space:

Before	After
	

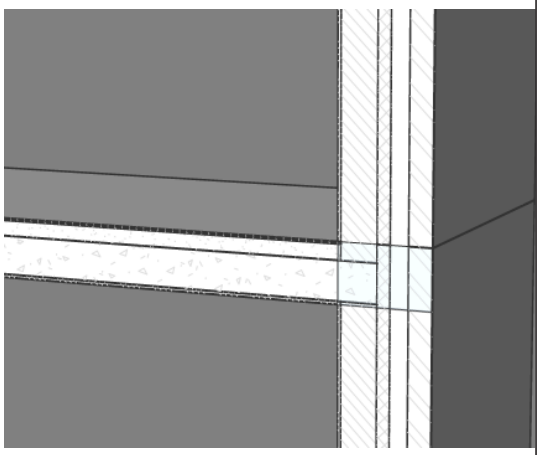
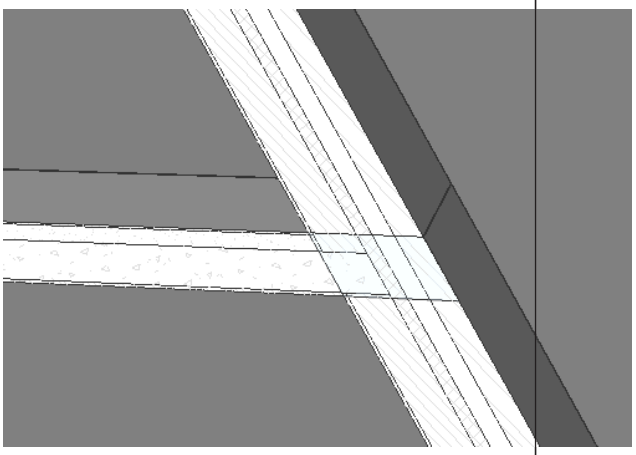
- Change the **Angle_32_30** parameter from 90 to 120. The new angle between the wall and the slab is different in the model space.

  A warning pops up:



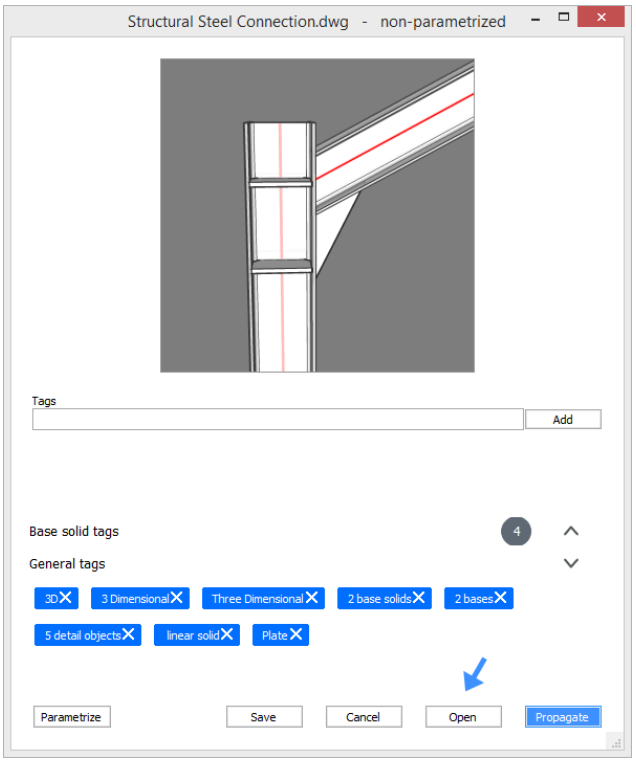
  This is because the **AngularTolerance** parameter was set to 10. The new **AngularTolerance** parameter automatically changes to 30 because you changed the **Angle_30_31** parameter by 30 degrees.

- Click **OK**

Before	After
	

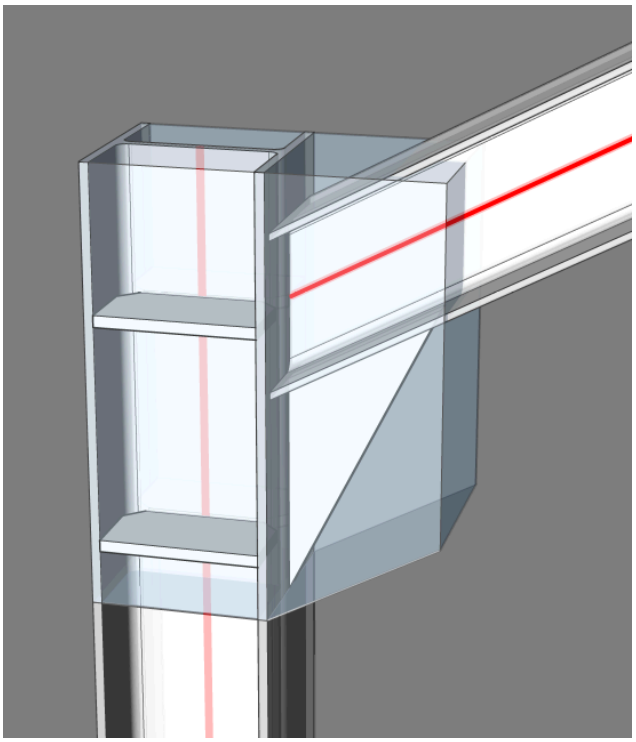
20.5.8.4 Parametrize and edit a structural steel connection

1 To open the .dwg file of the detail you want to parametrize, click on the detail in the **Details** panel. The **Detail** dialog box displays:

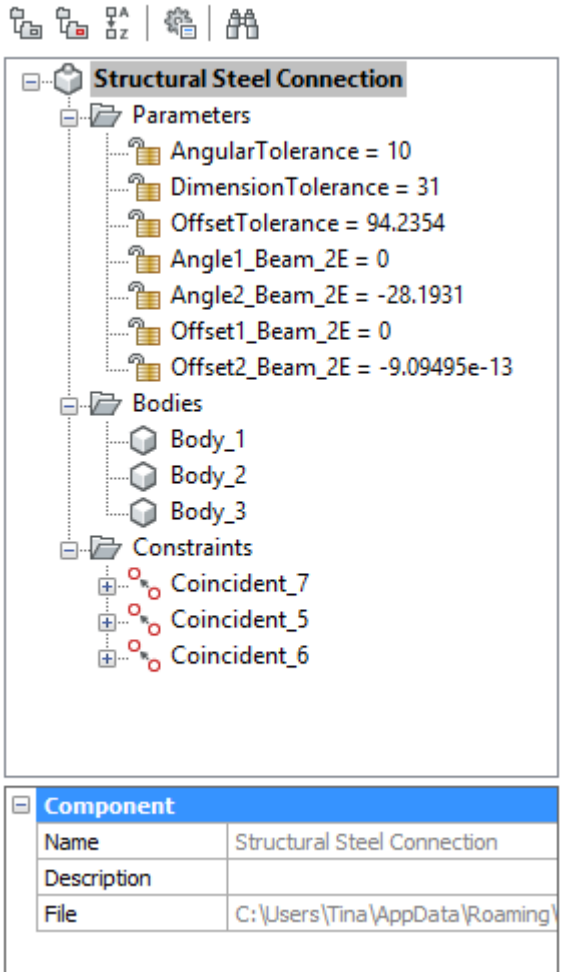


2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL.

The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**:



Nota: For connections between linear reference solids, only the framework parameters are automatically generated. You must manually add constraints to specify the behavior of detail objects in relation to the reference solids. In this example, three coincidence constraints have been manually added: two to make sure that the connection planes of the triangular steel plate maintain coincident with the flanges of the steel column and girder, and one to make sure that the end section of the girder maintains coincident with the column flange. You can add coincidence constraints before or after the parametrize tool. If you add the coincidence constraints after you can see how the reference solids change with new parameters. This makes it clear which constraints have to be added to make all detail objects move correctly.

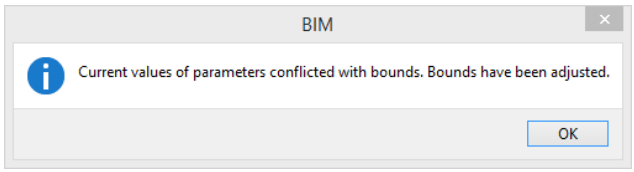
4 Edit parameters.

5 The parameters that were generated have specific values that make sure the detail looks exactly as it was created. If you change these values, the detail will change.

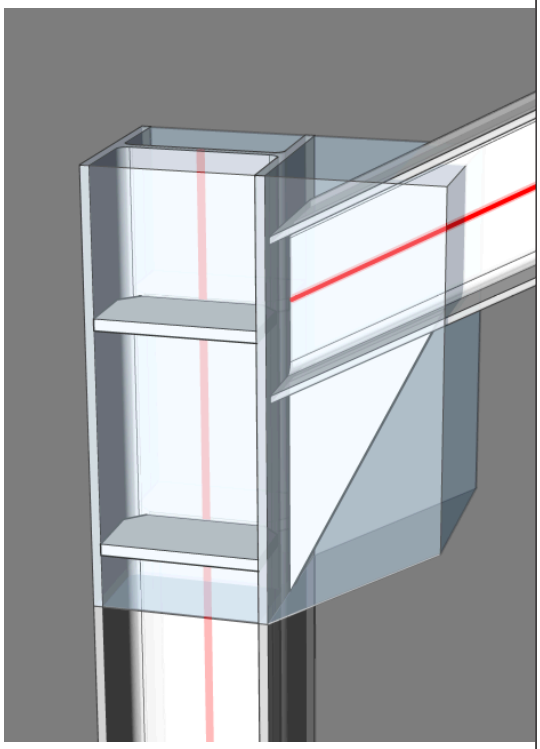
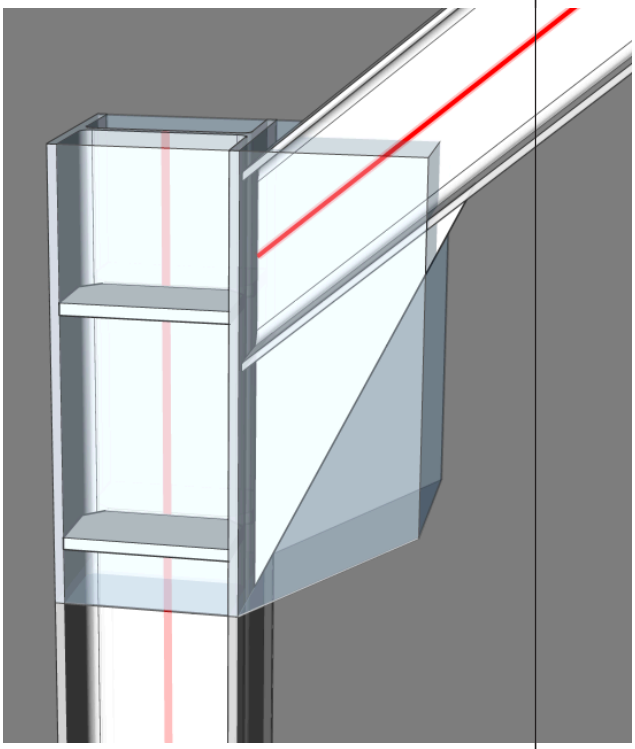
For example:

- Change the **Angle2_Beam_2E** from -28.1931013282 to -40. The column and the girder change in the model space.

  A warning pops up:

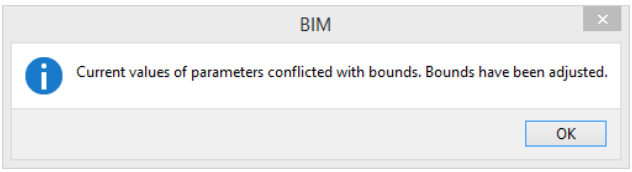


This is because the **AngularTolerance** parameter was set to 10. The **AngularTolerance** parameter automatically changed to 11.8069 because you have changed the **Angle2_Beam_2E** by 11.8069 degrees. Click **OK**.

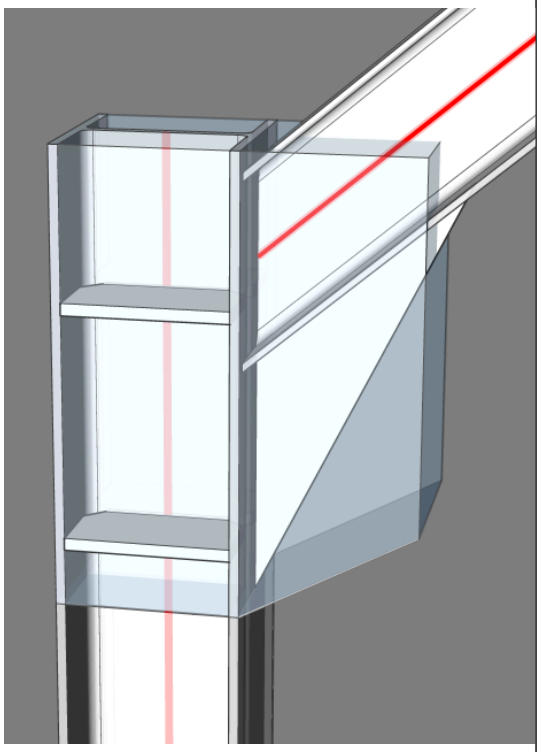
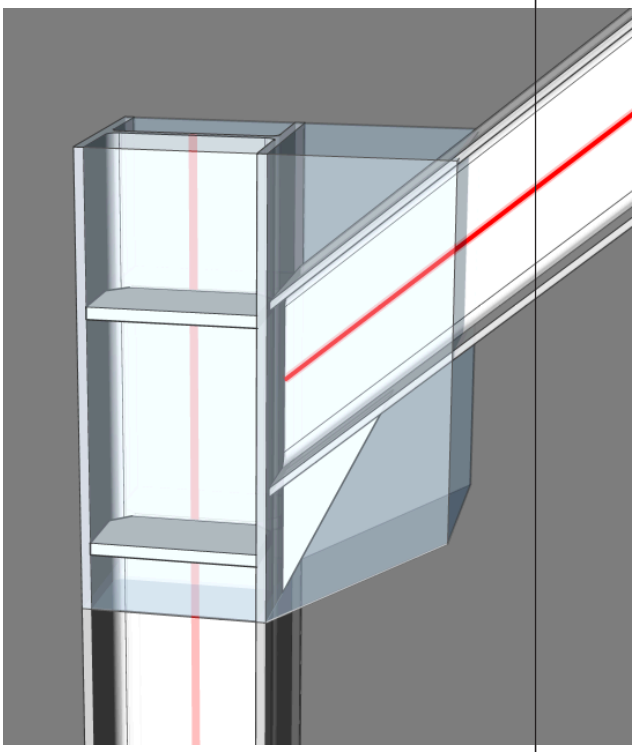
Before	After
	

- Change the **Offset1_Beam_2E** 0 to 200. You see the girder moves 200 mm downwards.

  A warning pops up:



This is because the **OffsetTolerance** parameter was set to 31. The **OffsetTolerance** parameter automatically changed to 200 because you changed the **Offset1_Beam_2E** parameter by 200 mm. Click **OK**.

Before	After
	

20.5.8.5 Propagate a detail in a project

There are different methods to propagate a detail:

- With the **Details** Panel: propagate a detail from the project on all similar connections in the project.
- With the BIMPROPAGATEFROMFILE command: propagate a detail from a file.

Fewer suggestions are given by the BIMPROPAGATE command for a detail that is not parametrized. This is because a detail that is not parametrized has fewer possible locations.

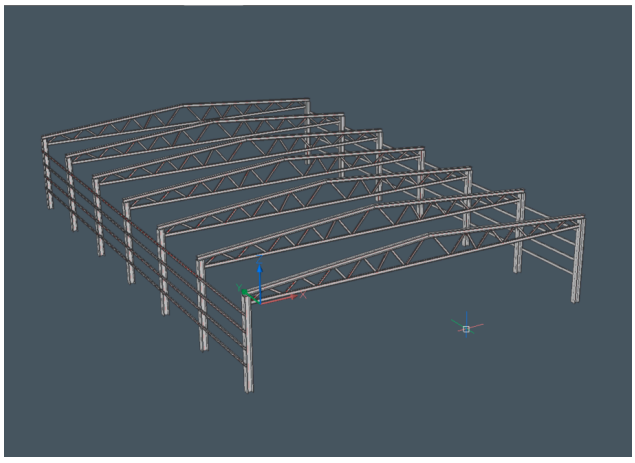
For example: if a non-parametrized detail of a wall-slab connection is propagated, BricsCAD BIM won't be able to propagate this detail to a wall-slab connection when the composition is different or a particular ply in the wall differs 10 mm from the corresponding ply in the wall of the original detail. To solve this problem parametrize the detail.

The BIMPROPAGATE command can adapt a detail with parameters to situations that differ from the original (eg: another corner, a smaller profile, etc.). You can decide how much the connections can differ from the original with the tolerance parameters.

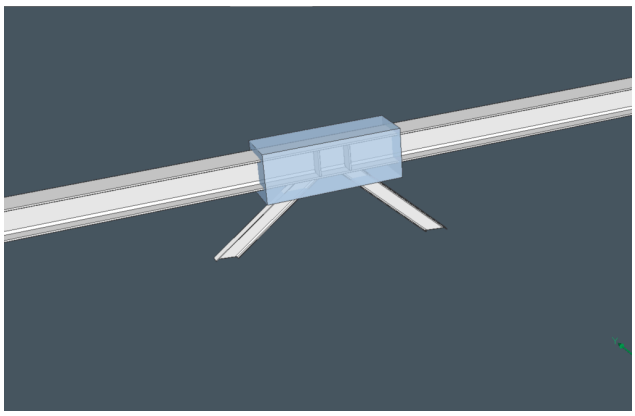
Different tolerance parameters:

- **Angular**: Sets the tolerance on angle parameters of the detail in degrees. Angles in the target situation cannot deviate more than this tolerance value from the original angles in the detail.
- **Dimension**: Sets the tolerance for the profile dimensions of linear reference solids in the detail. Profile widths and heights in the target situation cannot deviate more than this tolerance from the original widths and heights in the detail.
- **Offset**: Sets the tolerance for offset parameters of the detail. Offsets in the target situation cannot deviate more than this tolerance from the original offsets in the detail.
- **Thickness**: Sets the tolerance for the total thickness of planar reference solids in the detail. Thicknesses in the target situation cannot deviate more than this tolerance from the original thicknesses in the detail.

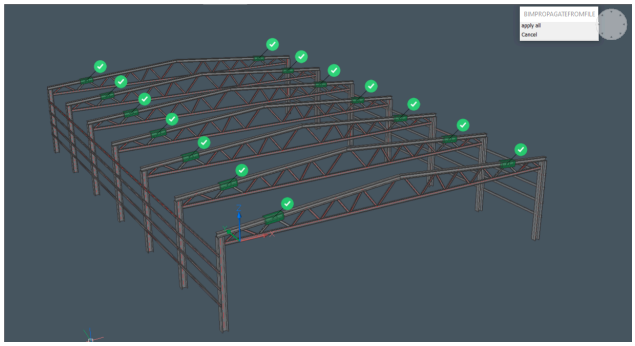
In the following project, there are a lot of similar connections present. However, the angles of the connections vary and the lower beam in the truss and has a different profile size than the upper beam.



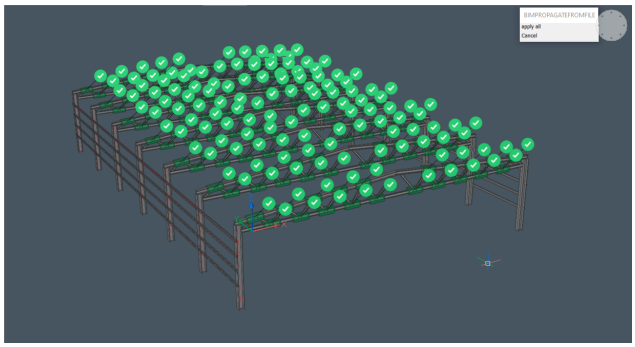
Propagate a detail in the project:

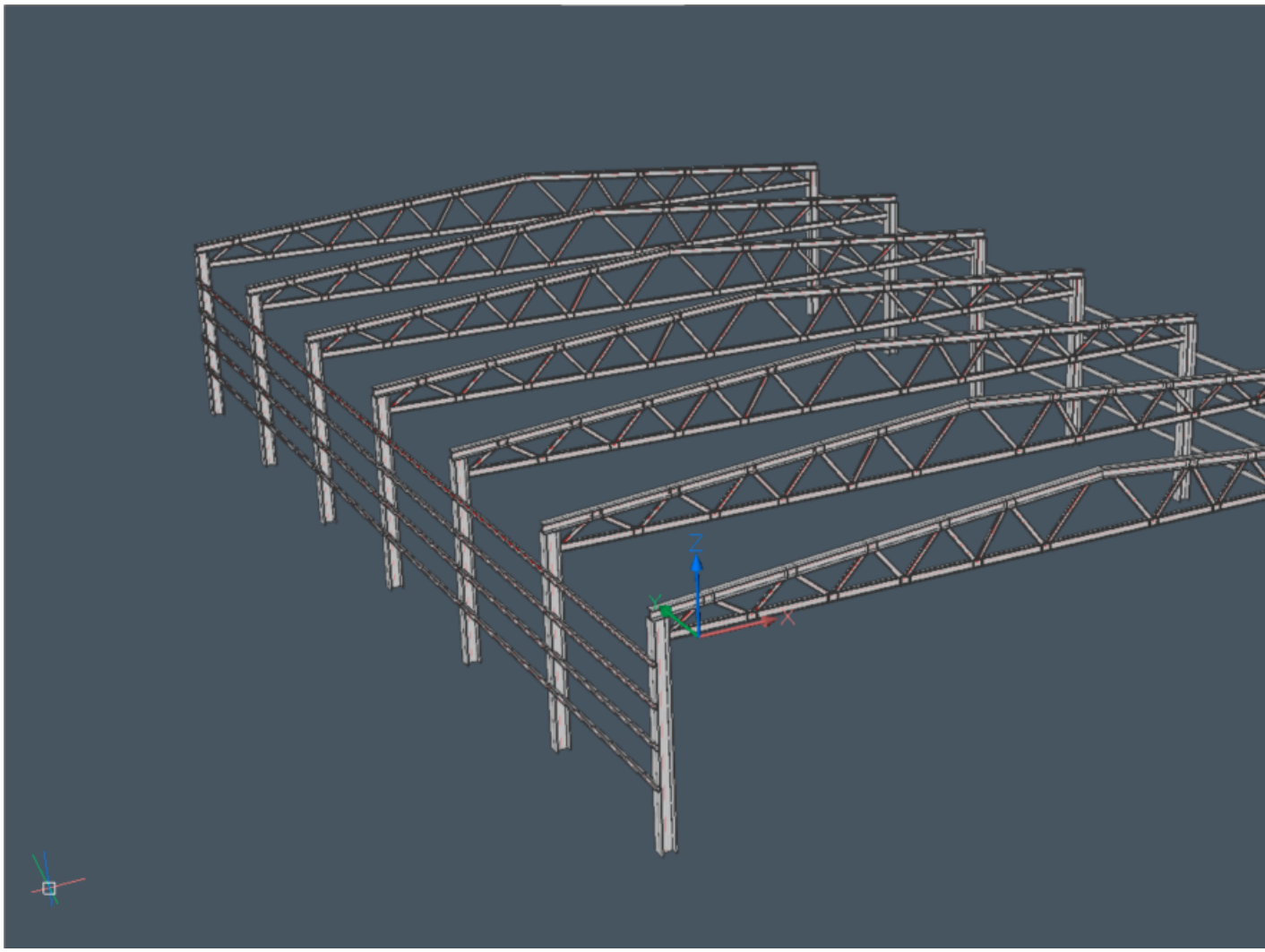


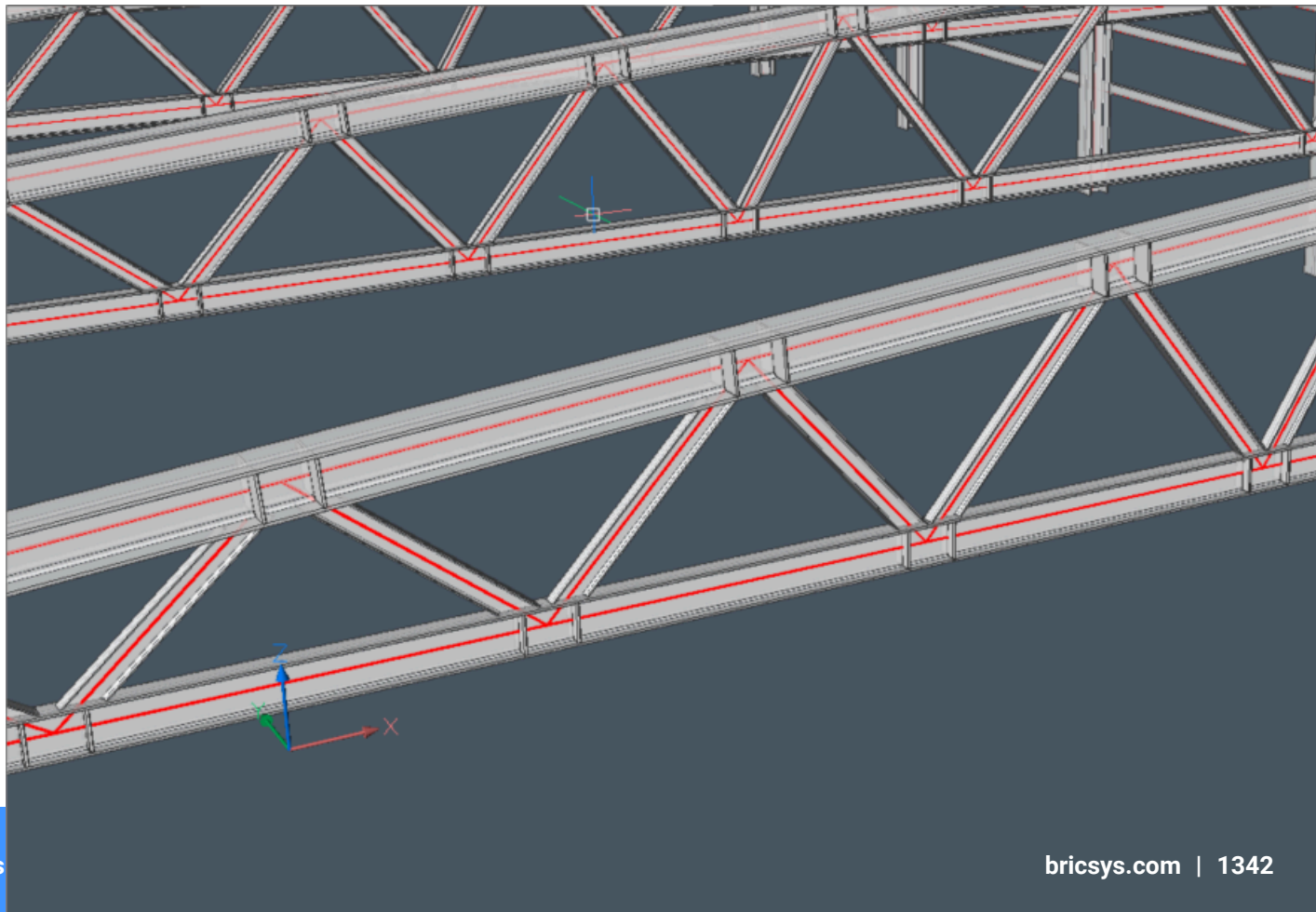
The **Mechanical Browser** displays the parameters. The **AngularTolerance** parameter is set to 0 degrees, the detail will only propagate to connections with exactly the same angle.



Change the **AngularTolerance** parameter to 45 degrees. The detail will propagate to all the similar connections. This is because none of the connections exceed the tolerance value.







20.5.9 Details panel

20.5.9.1 Commands

DETAILSPANELOPEN, DETAILSPANELCLOSE

20.5.9.2 Opening the details panel

Execute the DETAILSPANELOPEN command.

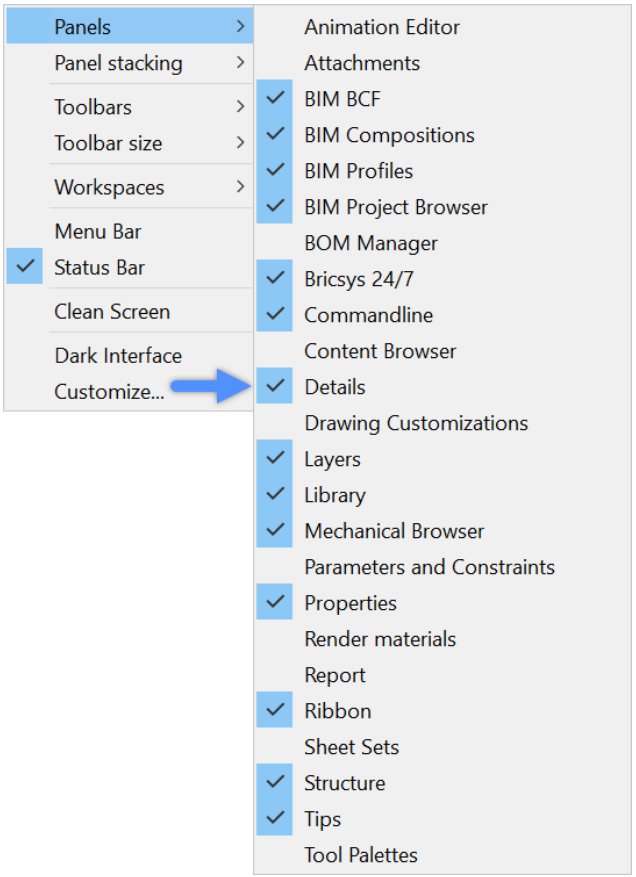
or

1 Place the cursor on a toolbar or ribbon panel and right-click.

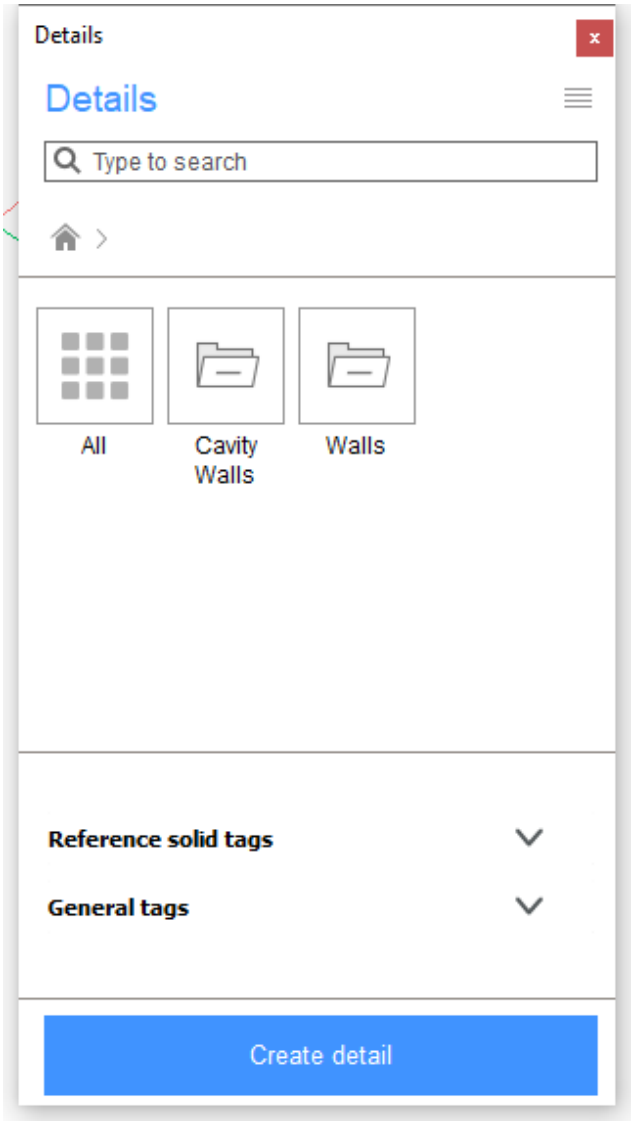
1 A context menu displays.

2 Expand the **Panels** option.

3 Select **Details** from the context menu.



The Details panel displays and the **Details** button ( ) is added to the collapsible panelset at the right-hand side of the BricsCAD application window.



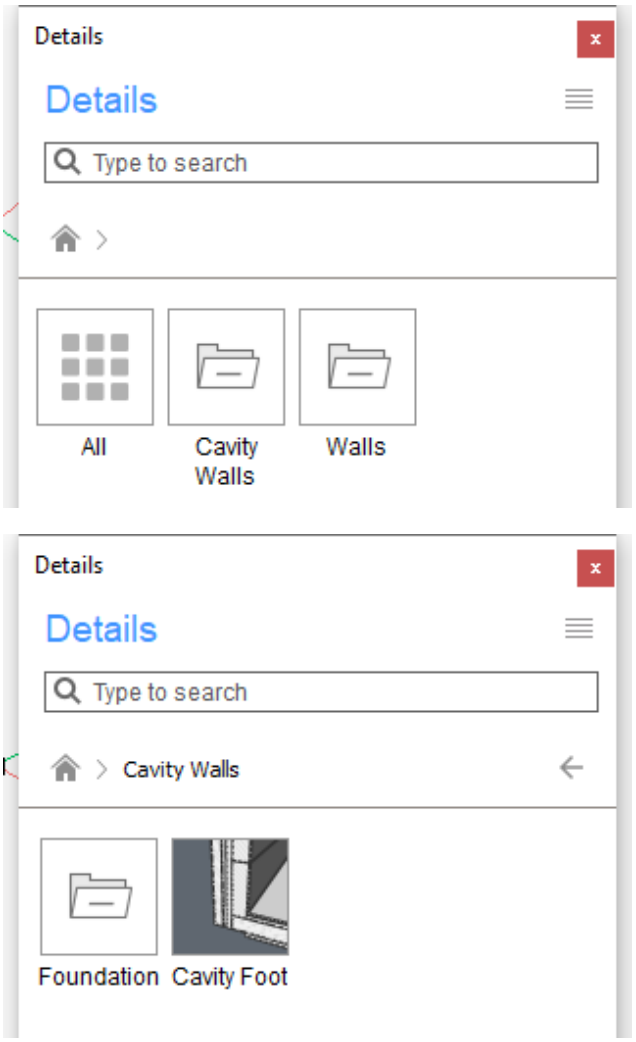
Click on the **Details** button ( ) to expand/collapse the **Details** panel.

20.5.9.3 Saving details to the details panel

After you created a new detail, it's possible to save it to the **Details** Panel. This way you can use the same detail in other files as well. Use the command BIMCREATEDETAIL.

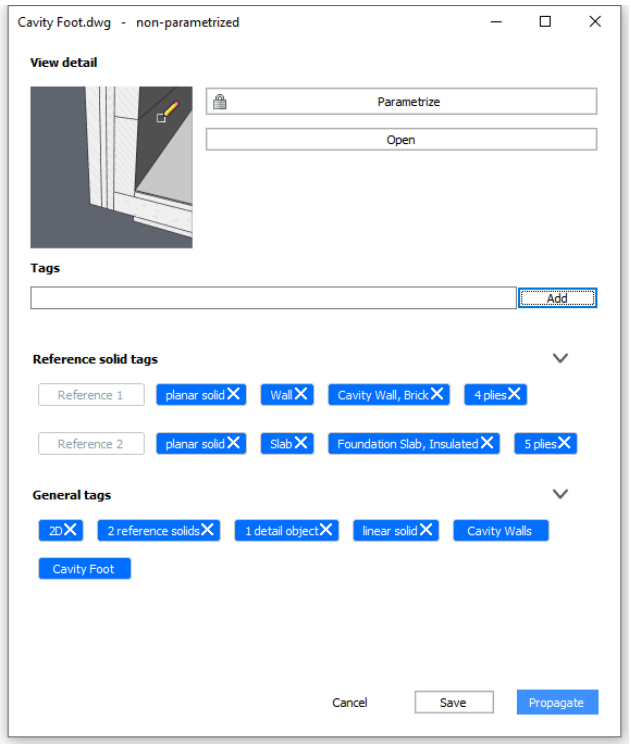
20.5.9.4 View a detail

1 Click on the category of the detail you want to view. The details assigned to that category display.



2 Click on the detail you want to view.

3 The **Detail** dialog box displays:



- Edit the thumbnail image:
  - Click the **Pencil** icon (✎).

    The detail dwg opens.

    The CREATETHUMBNAIL command is launched.
  - Use view commands (zoom, pan, rotate) to adjust the view.
  - Choose **OK** or press Enter to save the thumbnail.
- **Parametrize**: Opens the .dwg detail file and executes the BIMPARAMETRIZEDETAIL command.
- **Open**: Opens the .dwg file of the detail.
- **Tags**: Add general tags.
- **Reference solid tags**: Click the down arrow to display the tags. You cannot add reference solid tags; you can only delete the existing tags.
- **General tags**: Click the down arrow to display the tags.
- **Save**: Saves the changes and closes the **Detail** dialog box.
- **Propagate**: Propagates the detail. See the BIMPROPAGATE command.
- **Cancel**: Closes the detail dialog box without saving.

Tags can be added to each detail. These are used to search for a specific detail in the library. There are two different tag categories:

- **Reference solid tags**: Tags that identify an individual reference solid from the detail.
- **General tags**: Tags that generally identify the detail.

During the BIMCREATEDETAIL command, tags are automatically generated for the detail. You can delete unrequired tags and add new tags. New tags can only be **General tags**.

You can add tags during the BIMCREATEDETAIL command or at any moment after the detail has already been created.

To add tags during the BIMCREATEDETAIL refer to the BIMCREATEDETAIL article. To add tags at another moment, see below.

Managing tags

1 In the **Details** panel, click on the detail you want to adjust.

The **Detail** dialog box displays.

2 Expand the **Reference solid tags** category.

You can only delete these tags, you cannot add new tags.

3 Click in the **Tags** field and type the name for the new tag, then click the **Add** button to add **General tags**.

The new tag is added to the **General tags** category in an orange box.

Repeat this procedure to add more tags.

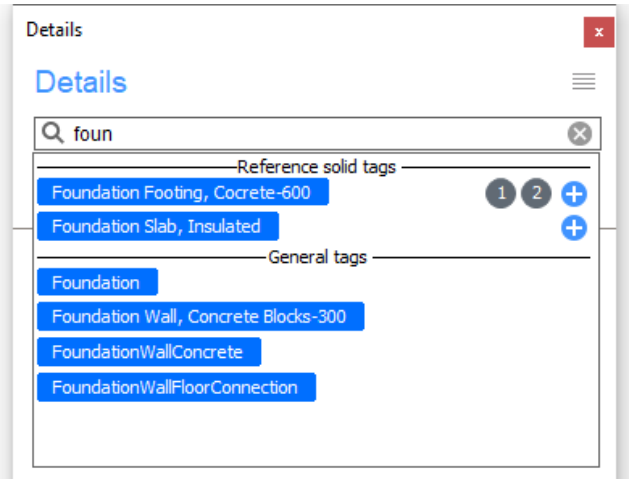
4 Click the **Save** button to save the changes and close the **Detail** dialog box.

Searching details with tags

In the **Details** panel you can search the library for specific details using tags. You can add one or more tags at once to your search query to select all matching details.

1 Click the search bar in the **Details** panel and type in a search string. The matching tags display:

Reference solid tags and the **General tags**.



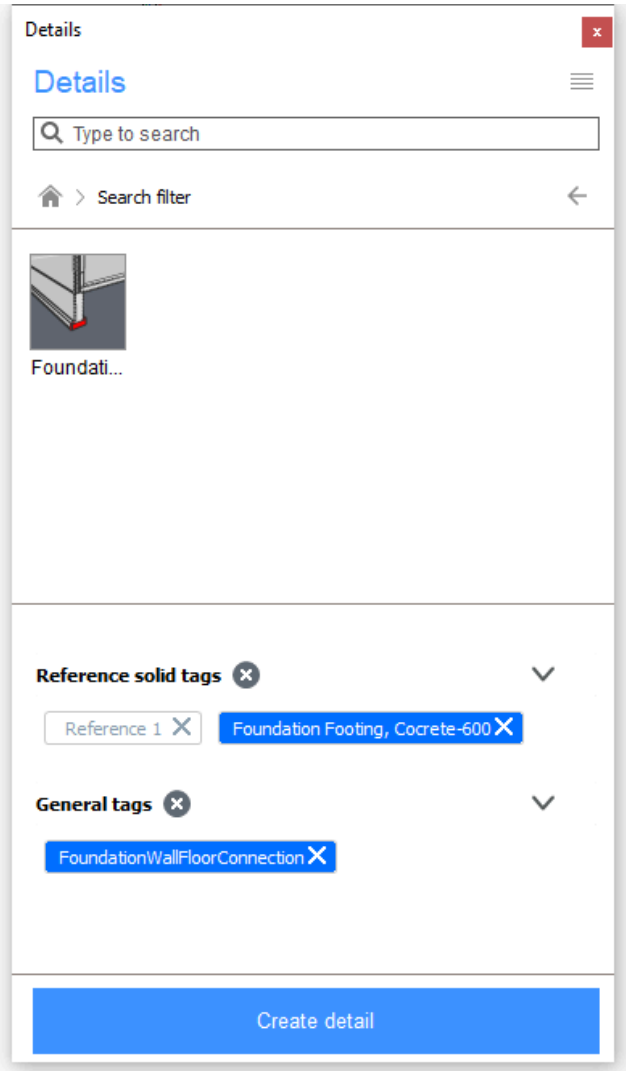
2 Do the following:

- **General tags**: Click the desired tag.
- **Reference solid tags**:
  - To add the tag to a reference solid that is not yet in your search query: click the plus sign next

to the required tag.

- Add the tag to a reference solid that is already in your search query: click the number of the required reference solids next to the tag.
- Once these tags have been added to your search query, they will appear at the bottom of the **Details** panel under the right tag category.

The filtered search results display.



20.5.9.5 Editing a detail

1 Select the detail you want to edit.

2 Click the **Open** button.

The .dwg file of the detail opens in a separate drawing tab.

3 Make the necessary adjustments to the detail.

4 Save the detail .dwg file.

5 The detail is updated in the **Details** panel.

6 The existing details won't be updated automatically. To replace the existing details by the new one, follow the steps below to propagate a detail.

20.5.9.6 To propagate a detail

You can propagate a detail to all of the similar connections in your project.

Do one of the following:

Drag and drop the detail from the library into the drawing.

or:

1 Select the detail in the library.

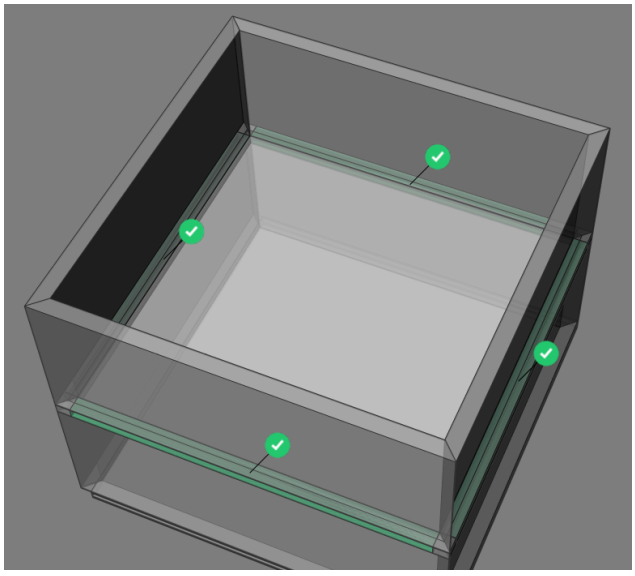
1 The **Detail** dialog box dispays.

2 Click the **Propagate** button.

To apply all suggestions, hit Enter.

To reject a suggestion click on the checkmark. A red cross icon replaces the checkmark. Click the icon again to accept the suggestion.

Press Enter to accept.



You can also propagate a detail in your project using the BIMPROPAGATEFROMFILE command. This command allows you to propagate detail files that are not in the library.

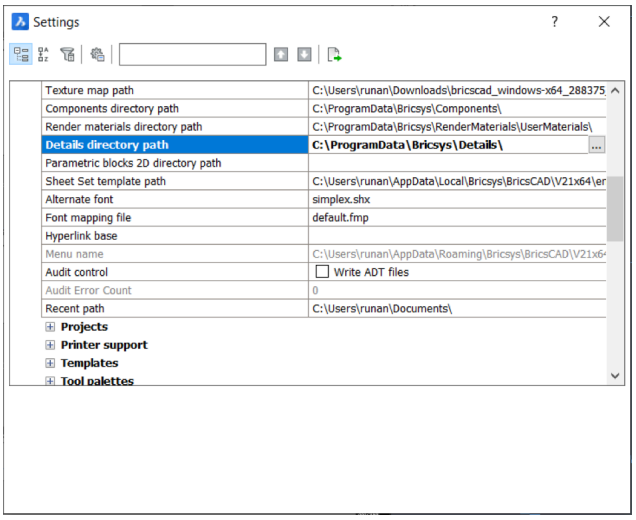
20.5.9.7 Sharing details

You can save details as a dwg-file and share the details folder/library. Save the detail drawings at the correct location which will then display in the details panel.

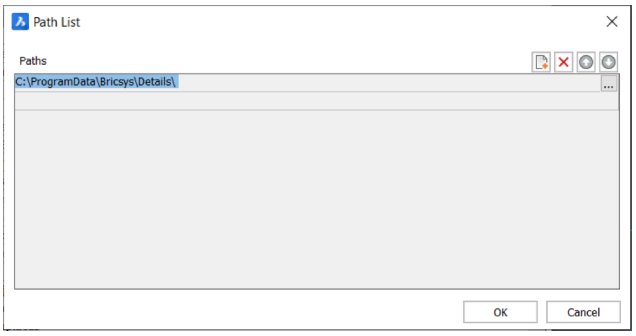
1 If you want to make a new detail to share, you can create a new detail first, as is explained above;

1 The following steps apply both to new details as well as existing details.

2 All the details you have created are automatically saved in the folder, and its subfolders, which is set through the DETAILSPATH system variable. By default, this is **C:\ProgramData\Bricsys\Details**.



Then click on the three dots next to the existing name in the **Path List** to change the location.

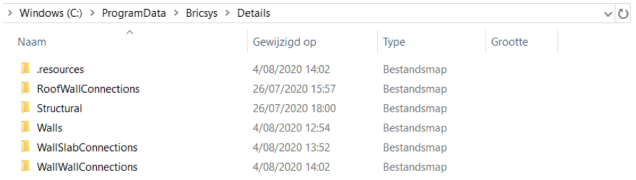


You can also create additional paths.

Click the **New** icon (), then click on the three dots of the new row to add a path for the **Details** library.

3 If you want to share a folder, go to the file explorer on your computer and follow the path to the details folder. If you haven't created any details yet, this folder only contains the folder '.resources'.

4 If you have created some details, you can find the same folders here as in your **Details** panel.



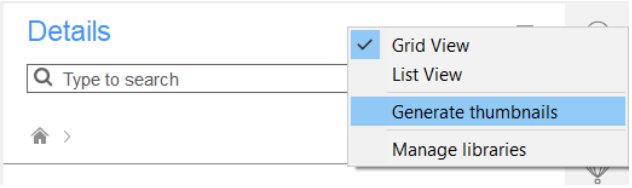
5 Copy the folder that you want to share and turn it into a zip file. This zip file can be shared with others.

6 If you receive a zip file you should unpack it and copy the folders containing the details to the folder *C:\ProgramData\Bricsys\Details*.

Nota: Make sure the details are saved in a folder that you place in the **Details** folder. If you just place the dwg files in your **Details** folder, they will not appear in your **Details** panel.

7 To load the new details in your library panel, go to the **Details** panel, open the menu and click **Generate thumbnails**.

The new details folders will appear in your **Details** panel.



20.5.10 Rhino/Grasshopper integration

20.5.10.1 Commands

RHINO, GRASSHOPPER, TOGRASSHOPPER

20.5.10.2 About

Rhinoceros3D is a 3D modeler developed by Robert McNeel & Associates of Seattle, Washington USA. Rhino is used to create, edit, analyze, document, render, animate and translate NURBS (curves, surfaces, solids, point clouds and polygon meshes).

Grasshopper is a visual programming language environment that runs within the Rhinoceros 3D computer-aided design application. It is tightly integrated with Rhino's 3D modeling tools.

20.5.10.3 Installing Rhino/Grasshopper connection

1 Download Rhino 7 from https://www.rhino3d.com/download.

Nota: Since the releases of BricsCAD V21 and Rhino 7, you can use the Rhino/Grasshopper Connection within BricsCAD with trial license keys of both BricsCAD BIM V21 and Rhino 7. Keys for older versions of Rhino will not work for this integration.

2 Download the Grasshopper-BricsCAD Connection from the BricsCAD Application Store: https://www.bricsys.com/applications/a/?rhino-grasshopper-connection-for-bricscad-a1353-al2360. The installer copies all required files to the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

Nota: The Grasshopper-BricsCAD Connection currently available from the Application Store will only work within BricsCAD BIM V21, not within older versions.

3 Restart BricsCAD V21.

4 To launch Grasshopper or Rhino, open and save a new drawing. Then use the buttons in the new tab that is displayed in the ribbon or launch with the Quad.

To launch a Grasshopper file from BricsCAD

1 Open an empty BricsCAD file.

2 Save it.

3 Open Grasshopper with the button in the new **Grasshopper** tab inside BricsCAD.

4 You can close the Rhino pop-up window. Rhino will continue to work in the background.

5 In the **File** tab of Grasshopper, choose **New document** to open a new and empty Grasshopper script.

6 Choose **Open document...** to open an existing Grasshopper script. You can use any script you made with Grasshopper in the past and open it in BricsCAD, as long as the input geometry from another program has been internalized. (To internalize data right-click the input geometry component and hit **Internalise data**.)

7 The geometry you created with the script should now pop-up in your BricsCAD model space, displayed in red. Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed.

8 If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

To link a Grasshopper file with a BricsCAD file

By default, the BricsCAD file that the Grasshopper document will link to is the one that was active when you launched Grasshopper from BricsCAD. If you want to link it to another BricsCAD file, you need to:

1 Make the target BricsCAD drawing the current drawing (i.e. the one you see in the model space window of your current session).

2 Link the Grasshopper script to this open drawing. You do that by clicking the Link icon  in the bar just above the Grasshopper canvas that contains the script that you are trying to link.

To launch a sample file

The sample files are provided with the Grasshopper-BricsCAD Connection. They can show you what is possible with the Connection. They are stored in the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

1 Open up a sample .dwg file by opening a .dwg from the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

2 You will get a warning that the file is read-only. This happens because the document is located in a secured folder. Click **Yes** to open as read-only.

Optional: If you don't want this warning, or if you want to edit the file, copy-paste the sample files into one of your own folders and open them from there instead.

3 From inside Grasshopper, choose **File** > **Open Document**. Go to the 'Program Files\Bricsys \Grasshopper-BricsCAD Connection' folder again.

4 Choose the .ghx-file with the same name as the .dwg file.

5 The sample file now opens and you should see red preview geometry in your BricsCAD model space.

6 Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed. If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

7 You can play around with the input sliders in the Grasshopper script.

8 If you are happy with your model and would like to convert the preview geometry into real BricsCAD geometry, you can 'bake' the geometry into BricsCAD. You do that by selecting all the **Bake Building Element** components (Shift-clicking them all) and pressing the **Bake** button .

9 A pop-up window appears, just click **OK** to accept.

10 The geometry is now converted into real BricsCAD geometry, even with BIM data attached to it.

11 You won't be able to save your changes to your files, as they were read-only. So, discard your changes or save the files as new ones in a different location.

20.5.10.4 Customizing the Rhino/Grasshopper connection

The code for the Rhino/Grasshopper Connection is 100% open-source and is available at https://github.com/Bricsys/rhino.inside-bricscad. This allows you to fully customize your Connector by coding to your heart's content (assuming, of course, that you know how to code).

20.5.10.5 Uninstall the Rhino/Grasshopper connection

1 Launch **Add or Remove Programs** in Windows.

2 Search for **Grasshopper-BricsCAD connection**.

3 Hit **Uninstall**.

20.5.10.6 How to draw a box in Rhino and bake into BricsCAD via Grasshopper

1 Open a new BricsCAD file.

2 Save the file.

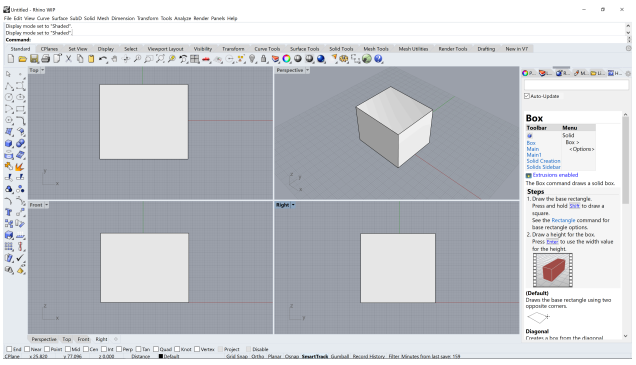
3 In BricsCAD, click on **Grasshopper** > **Rhino**.

4 Draw a Rhino box by typing BOX in the Rhino Command line. (This works in the same way as the BricsCAD BOX command).

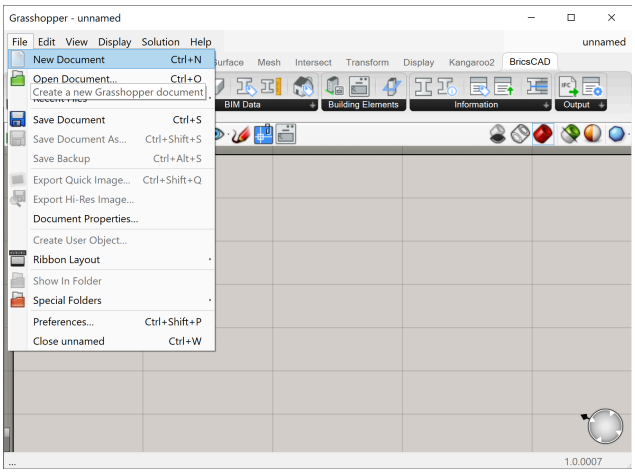
5 Click for the first corner of the base.

6 Click for the other corner of the base.

7 Click to define the height of the box.



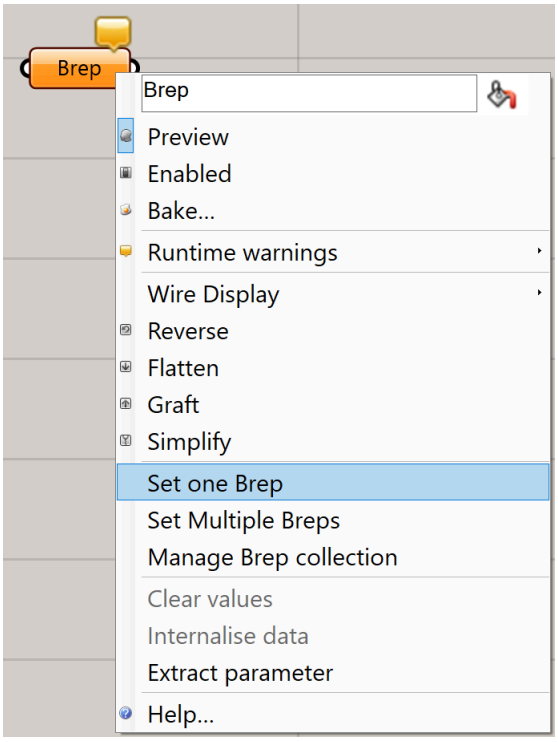
8 To open a new Grasshopper script, inside the Grasshopper environment, click on **GrasshopperGrasshopper** then click on **File** > **New Document**.



9 Save the file.

10 Drag and drop the **Params** > **Geometry** > **Brep component** onto the Grasshopper canvas.

11 Right-click on the component and select **Set one Brep**.



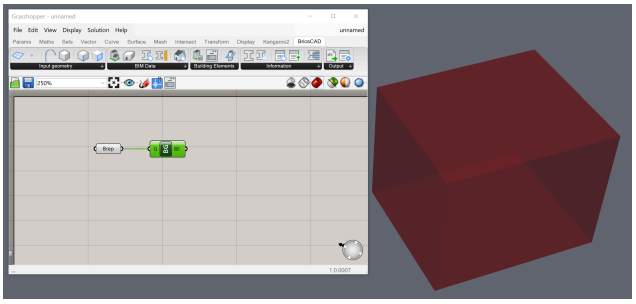
12 Go to the Rhino canvas and select your cube.

13 The Brep component will change color from orange to grey. If you close the Rhino window and return to the **Grasshopper** window, a preview of the cube will display in the BricsCAD model space.

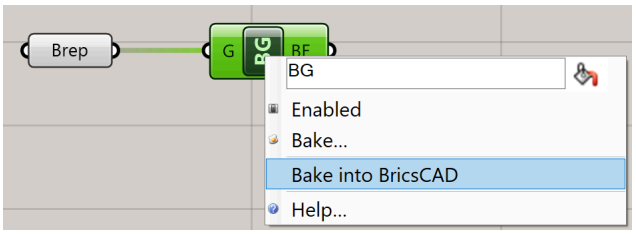
14 Drag and drop the **BricsCAD** > **Building Element** > **Bake Geometry component** onto the Grasshopper canvas.

15 To link the two components, click and drag from the Brep component's right dot to the BricsCAD Bake

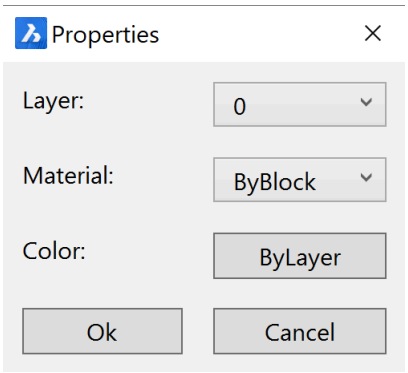
component's left dot.



16 Right-click onto the **Bake Geometry component** and choose **Bake into BricsCAD**.



The following dialog displays. Here you can choose the destination **Layer**, **Material** and **Color**:

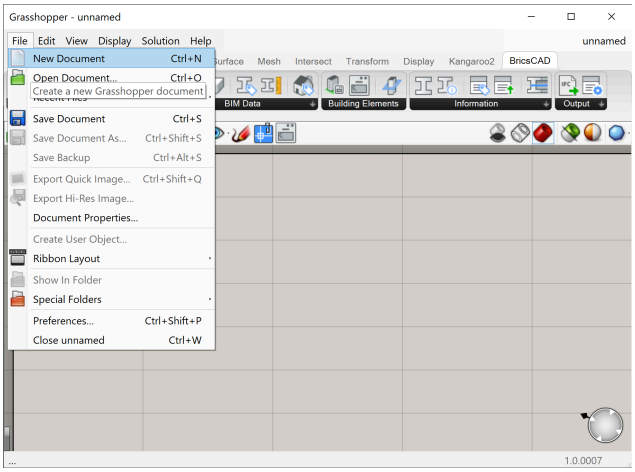


17 Click **OK**. The box is baked as a solid into BricsCAD.

20.5.10.7 How to draw a box in Grasshopper from a BricsCAD rectangle

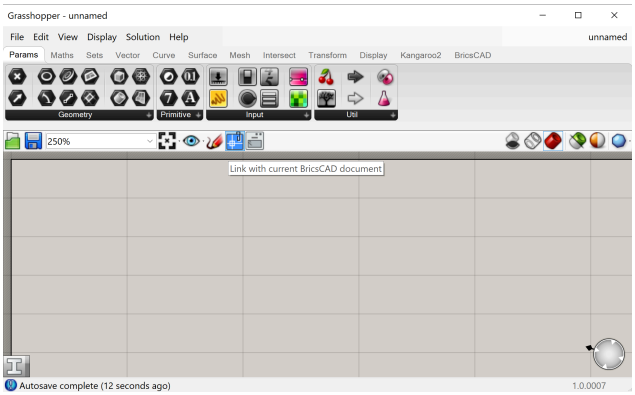
1 To open a new Grasshopper script, click on **Grasshopper** > **Grasshopper** and clicking on **File** > **New**

Document inside the Grasshopper environment. Save the file.



2 Open a new BricsCAD file in a **meter template** and save it.

3 To link the new script to the open .dwg file hit the **Link** button in the **Canvas** Toolbar.



4 Go to the BricsCAD model space and draw a rectangle.

Nota: Use a polyline to create irregular geometry.

5 Drag and drop the **BricsCAD** > **Input Geometry** > **Curve component** onto the Grasshopper canvas.

6 Right-click onto the **Curve component** and choose **Set One BricsCAD Curve**.

7 Go to the BricsCAD model space and select the rectangle.

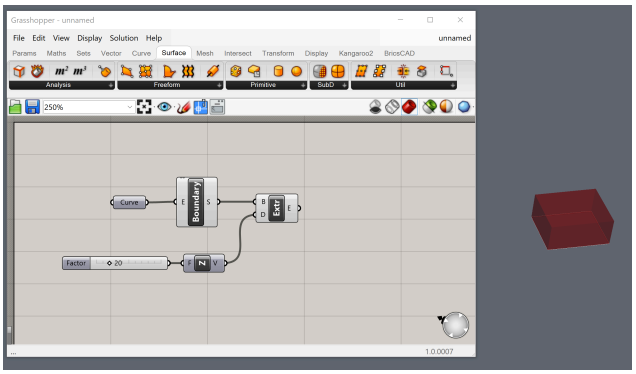
Nota: Save time with the Quad: select the curve in BricsCAD and then select **ToGrasshopper** under the **Grasshopper** tab of the Quad. A Grasshopper component appears in the upper left corner of your Grasshopper canvas.

8 Link the Curve component to a **Surface** > **Freeform** > **Boundary Surfaces component**.

9 Link this to a **Surface** > **Freeform** > **Extrude component**.

10 In the Direction input of the Extrude component link a **Vector** > **Vector** > **Unit Z component** with **Params** > **Input** > **Number Slider component**. Set this to the height of your box, 17 for example.

Nota: You can edit the range and preciseness of the slider by right-clicking the component and selecting **Edit...**. A dialog box pops up where you can set the digits and min and max value of the slider.



11 You now have drawn a box in Grasshopper, using BricsCAD geometry as an input.

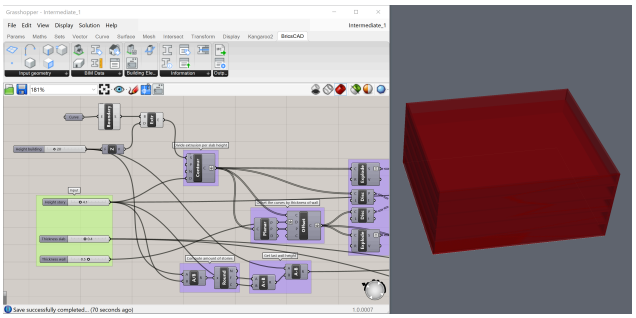
20.5.10.8 How to split up a box into different floors using Grasshopper

1 Add the 'Start_1.ghx'-code to your previous Grasshopper script.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_1.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
   - Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_1).
   - You then get a drop-down of all the active Grasshopper scripts, so open the one you saved in 'How to draw a box in Grasshopper from a BricsCAD rectangle'.
   - Now click on the canvas and hit **Ctrl+V** to paste the 'Start_1.ghx'-code.

2 Right-click to rename the number slider from the previous procedure to **Height building**.

3 Link this number slider to input A from the Division component in the **Compute amount of stories** group and to input A from the Subtraction component in the **Get last wall height** group.

4 Link your extrusion to the Shape input of the Contour component.



5 Do some cleaning up of the file:
   - Drag the green **Input** group to the front of your canvas.
   - Add the **Height building** slider to the group by selecting the component and then right-clicking on the group and choosing **Add to group**.
   - Put off the preview for your Boundary and Extrusion components: select the components (Shift-click them) and right-click on an empty space of your canvas and choose **Preview Off**.

6 **(Optional)** For intermediate files that already have these changes, use 'Intermediate_1.ghx' and 'Intermediate_1.dwg'. (Located in the zip file at the bottom of this page.)

7 The script you just made allows you to:
   - Divide the extrusion per slab height, using the Contour component.
   - **Offset** the curves by the thickness of the walls.
   - Make the bottom surfaces of the walls (the surfaces are split into lower floor walls and roof walls).
   - Compute the last wall height.
   - **Extrude** the surfaces into roof, slabs, and walls.
   - Adapt the height of the building, the height of the stories, and the thickness of the slabs and walls, by using the sliders in the green **Input** group.

8 Add 3 **BricsCAD** > **Building Element** > **Bake Building Element components** to the end of your Grasshopper script.

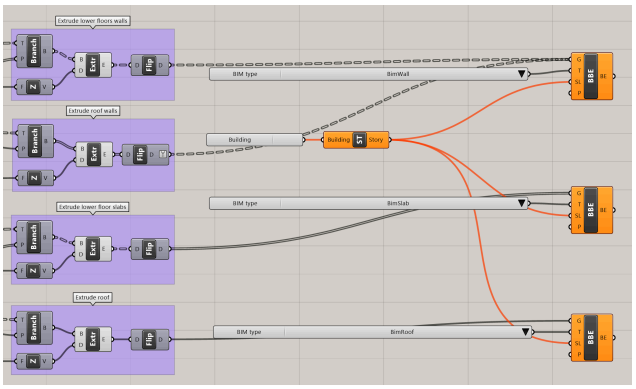
9 Attach the extruded elements (the Flip components) to the Geometry input nodes of the **Bake Building Element** components:
   - Connect the extruded lower floor walls and extruded roof walls to one **Bake Building Element** component. (Hold Shift while connecting multiple nodes to one input node).
   - Connect the extruded lower floor slabs to another **Bake Building Element** component.
   - Connect the extruded roof to the remaining **Bake Building Element** component.

10 Add the following components to your canvas from the **BricsCAD** > **BIM Data** category: 1 **Buildings**, 1 **Stories**, and 3 **BIM Types components**.

11 Attach Buildings to the Building input of the Stories component and attach this to the Spatial Location input of all the **Bake Building Element** components.

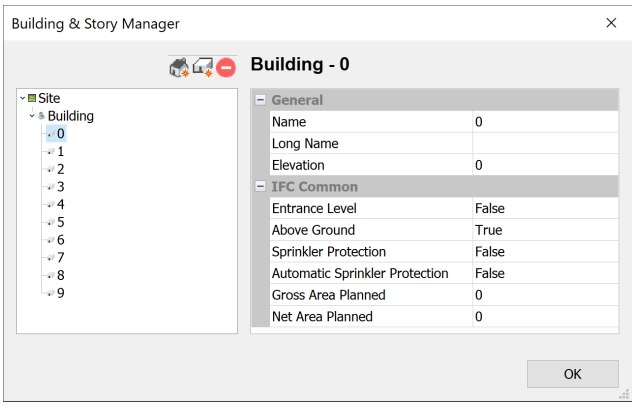
12 Attach BIM Types (set respectively to **BimWall**, **BimSlab**, and **BimRoof** in the selection menu) to the Element Type input of the **Bake Building Element** components of the extruded walls, the slabs, and the roof respectively.

13 The following warning will display: **Input parameter Building failed to collect data**. This is because, when no buildings are defined in the BricsCAD drawing, Grasshopper fails to collect input.

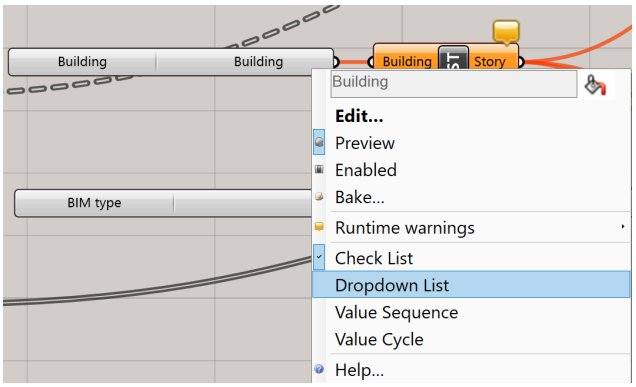


14 To make spatial locations, for the component to detect, go to BricsCAD and click on **Home** > **Classify** > **Spatial Locations**.

15 Add a building and a few stories (more stories than you want to divide your building into).



16 Go to Grasshopper and right-click the Buildings component and put it onto **Drop-down List** instead of **Check List**.

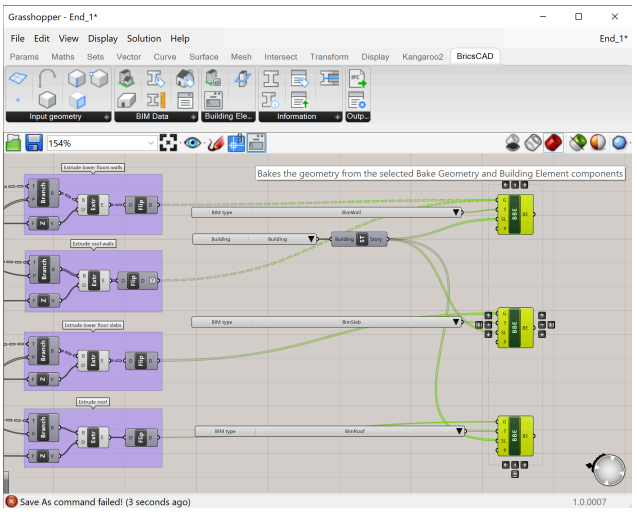


17 Select the newly made building in the drop-down list.

18 Shift-select all the **Bake Building Element** components and choose the **Bake** option from the **Canvas** Toolbar. Click **OK** to accept the **Layer**, **Material**, and **Color**.

19 The drawing will now have different floors.

Nota: Check the properties to see that the geometry is correctly classified as **Wall**, **Slab** or **Roof**. These elements will also have a spatial location assigned to them.

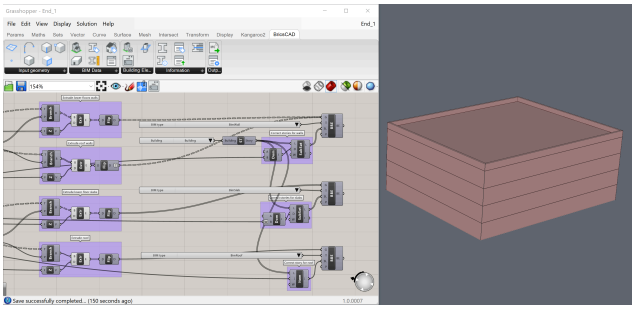


20 The last input geometry will be baked onto all remaining stories as there are now more stories than geometry. To correct this, select a correct subset of the stories to assign. Bake the geometry in BricsCAD and the stories will be correct. No 'duplicate' geometry will be baked.

21 You can see how it's done by opening the 'End_1.dwg' and then opening the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

22 You have now successfully created a building.

23 If you want to see the result of the baked building, check the file 'End_1_Baked.dwg' or bake it yourself.



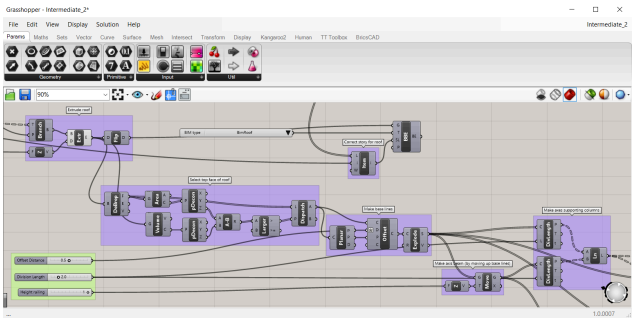
20.5.10.9 How to add a railing to the top of the building

We will continue with the building made in the previous procedure. So open the 'End_1.dwg' and then open the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

1 Add the 'Start_2.ghx'-code to the Grasshopper canvas.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_2.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.

- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_2).
- You then get a drop-down of all the active Grasshopper scripts, so open 'End_1.ghx'.
- Now click on the canvas and hit **Ctrl+V** to paste the 'Start_2.ghx'-code.

2 Link the **Extrude Roof** group's end component to the Brep input of the Deconstruct Brep component of the **Select top face of roof** group.



3 The script does the following:
- The **Select top face of roof** group will select the top face of the roof.
- The remaining script will take that face and use it to make the baselines for the railing.
- It will then make the axes of the supporting columns and beams.
- It will also calculate half of the height of the beam profile.
- There is still a profile required, however, to complete this step and the following one. We will, therefore, create a profile from step 6 onward.
- With that profile and the axes, the beams and supporting columns are made as extrusions.

4 The intermediate result is found in the 'Intermediate_2.ghx' and 'Intermediate_2.dwg' files. (Located in the zip file at the bottom of this page.)

5 **(Optional)** Skip the **Select top face of roof** group and replace it with a **BricsCAD** > **Input Geometry** > **Face component**. Set it to the top face of the roof slab in your BricsCAD baked building. (Do this by right-clicking on the component, click on **Set one BricsCAD face** and select the face in the BricsCAD drawing). Connect this Face component with a **Params** > **Geometry** > **Surface component** and connect this surface to the correct inputs.

6 Go to **BricsCAD** > **BIM Data**.

Drag the **Profile Names** and the **Profile Sizes** components onto the canvas.

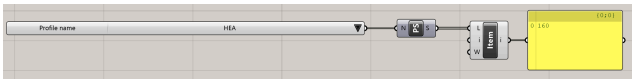
7 Select **HEA** from the drop-down list in **Profile Names**. This drop-down list displays all the names of the profiles in the **Profile Library** of BricsCAD.

8 Link the component to the input **ProfileName** of the **Profile Sizes** component.

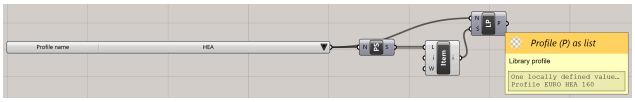


9 Link the **Profile Sizes** component to the input node **List** in a **Sets** > **List** > **List Item component**, where i is set to '4' (right-click on the **i** input and **Set integer** to **4**). This **List Item** lets you pick one size from the list with all available profile sizes for the **Profile Name** linked to it.

Nota: If you link a **Params** > **Input** > **Panel component** to the Profile Sizes output, you will see that the index 4 stands for size 160.

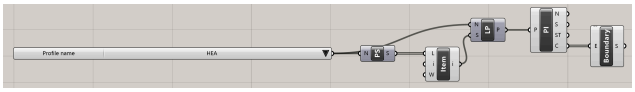


Link the **Profile Names** and **List Item** components to the **BricsCAD** > **Information** > **Library Profile component**, to get an HEA 160 profile as an output.



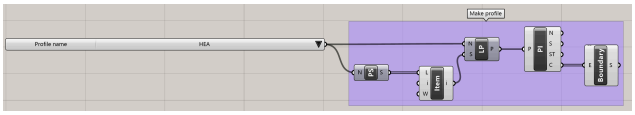
10 To get the curves that make up this profile, attach a **BricsCAD** > **Information** > **Profile Info component** to the **Library Profile**. One of the outputs created is the **Profile Curves** as a tree.

11 Attach a **Surface** > **Freeform** > **Boundary Surfaces component** to the **Profile Curves** output node. This is to later extrude the surface as a solid, instead of the lines as a surface.

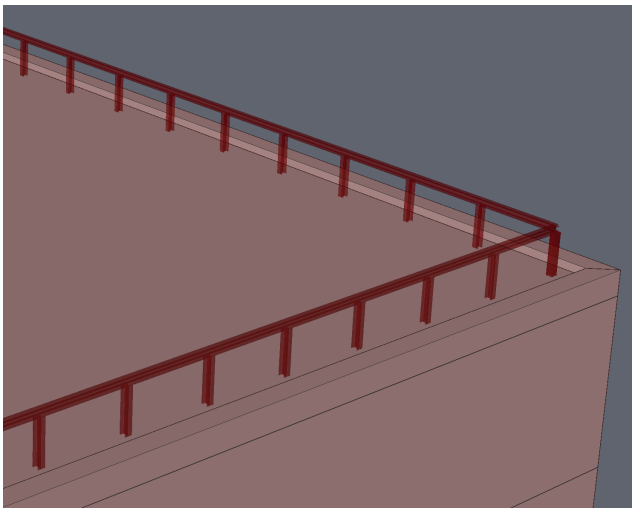


12 To group together the components just created, select them (with a selection box) and right-click on **Group**.

13 **(Optional)** To rename the group, right-click on the purple rectangle. Rename the group in the upper text box, e.g. to **Make profile**.



14 The Boundary Surfaces component should now be linked to the Content input of the Bounding Box component of the **Calculate half of height of beam** group. The Boundary Surfaces component should also be linked to the two Profile inputs of the Extrude components in the groups **Make supporting columns** and **Make beams**. The extrusions should now show you a railing with HEA 160 beams and columns.



15 To bake those beams and columns into BricsCAD attach two **BricsCAD** > **Building Elements** > **Bake Building Element components** to the Grasshopper canvas.

16 Do the following:
- Set the **Geometry** input to the extrusions.
- Set the **Element Type** input to a **BricsCAD** > **BIM DataBricsCAD > BIM Data > BIM Types component**, which is set to **BimColumn** for the column extrusions and to **BimBeam** for the beam extrusions.
- Set the **Spatial Location** input to the same as the roof.
- Set the **Profiles** input to the **Library Profiles** component from the **Make profile** group.

17 You now successfully created a railing on top of your building. (The full script to do this is in the 'End_2.ghx' file at the bottom of this page, which is linked to the 'End_2.dwg'.)

18 Bake into BricsCAD. The profiles are now set to the one in the Grasshopper script. You can still modify the connections of the beams in the corners by using L-Connect in BricsCAD. (To see the finished building with the railing on top of it, check out the 'End_2_baked.dwg' file at the bottom of this page.)

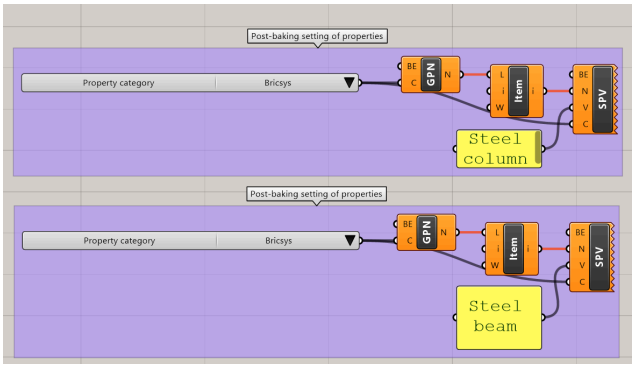
20.5.10.1 Procedure: add information to a baked model 0

We will continue with the building made in the previous procedure. So, open the 'End_2.dwg' and then open the Grasshopper file 'End_2.ghx' that is linked to the 'End_2.dwg'.

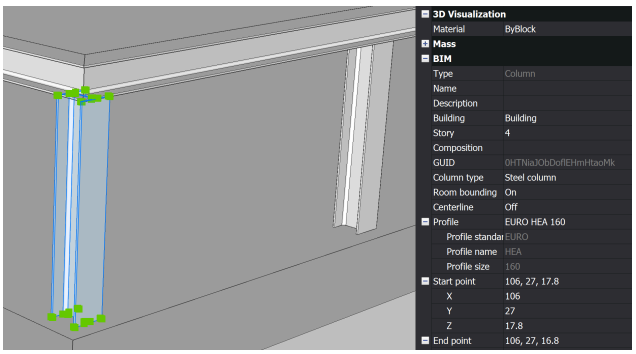
1 Add the 'Start_3.ghx' code to the canvas.
- You do that by downloading the zip file at the bottom of this page and extracting its content.
- Then open the 'Start_3.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
- You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current

document (Start_3).

- You then get a drop-down of all the active Grasshopper scripts, so open 'End_2.ghx'.
- Now click on the canvas and hit **Ctrl+V** to paste the 'Start_3.ghx'-code.



2 This code has some post-baking settings of classified and baked beams and columns:
   - The script uses the **Get Properties Names component** to display the BIM properties in the Property category **Bricsys**.
   - In those properties, it selects item 1 for the columns and item 0 for the beams, **ColumnType** and **BeamType** respectively.
   - To conclude, it sets the properties to the value specified in the **Panel components**, using the **Set Property Value components**.
3 To make the script work, bake the two **Bake Building Element components** that make up the railing.
4 Now link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Get Properties Names components**.
5 Also, link the Building Element output of the **Bake Building Element components** to the respective Building Element input of the **Set Property Value components**.
6 If you now select a column in BricsCAD and open its Properties, the **ColumnType** will be set to **Steel column**. When you change the value of the **Panel component**, the Properties in BricsCAD change in real-time.
7 The resulting script: 'End_3.ghx' can be found in the zip file, located at the bottom of this page. It works with the 'End_3.dwg' file in which nothing is yet baked. Bake all the Building Elements, to get the post-baking settings.



20.5.10.1 The BricsCAD toolset in Grasshopper

1

Icon	Name	Description
	Plane	Represents a plane in BricsCAD.
	Point	Represents a point in BricsCAD.
	Curve	Represents a BricsCAD curve.
	Entity	Represents a BricsCAD entity.
	Edge	Represents a BricsCAD edge.
	Face	Represents a BricsCAD face.
	Vertex	Represents a BricsCAD vertex.
	Buildings	Provides a name picker for all the buildings present in **Spatial Locations** in BricsCAD.
	Stories	Returns all the stories attached to the input building.
	Profile Names	Provides a name picker for all the profiles present in **Profiles** in BricsCAD.
	Profile Sizes	Returns all the sizes attached to the input profile.

Icon	Name	Description
	BIM Types	Provides a type picker for all the **BIM Types** available in BricsCAD.
	Property Categories	Provides a category picker for all the property categories available in BricsCAD.
	Bake Geometry	Bake the Grasshopper geometry into the current BricsCAD drawing, while disregarding the BIM data attached to it. The output of Bake Geometry is a reference to the baked building element without BIM data.
	Bake Building Element	Bake the Grasshopper geometry into the current BricsCAD drawing, while adding BIM data to it. The output of Bake Building Element is a reference to the baked building element with BIM data.
	Elements on Spatial Location	By default, returns all the building elements present in BricsCAD. When using input parameters, returns the building elements filtered by element type and/or spatial location.
	Library Profile	Returns a profile from the library, according to the given name and size.
	Profile Info	Returns the information (name, size, standard and curves) of the specified profile.
	Property Names	Returns the property names, attached to a building element, in the specified property category.

Icon	Name	Description
	Property Value	Returns the property value, attached to a building element, for the specified property name and category.
	Linear Solid Info	Returns information (axis, extrusion path and profile curves) about a linear solid present in the BricsCAD drawing.
	Set Property	Sets the property value of the building element according to the specified name, category and value.
	IFC Export	Exports the specified building elements to IFC.
	Link	Links the Grasshopper script to the open BricsCAD .dwg file.
	Bake	Bakes the geometry from the selected Bake Geometry and Bake Building Element components.

Starter files.zip

20.6 Colaboração no projeto

20.6.1 BIM collaboration format (BCF) panel

20.6.1.1 About BCF

The BIM Collaboration Format (BCF) is an open file format used to exchange textual comments, images, and more on top of an IFC model. This allows better communication between different parties in the BIM process.

BCF makes it possible to link information such as a comment, a screenshot, a list of involved objects, and a camera viewpoint to an issue. An issue can be anything ranging from a small change in the design to clashes between different disciplines in the model. These issues can then either be saved as .bcf files or they can be managed on the cloud, using any sort of BCF cloud management system.

The advantage of BCF is that it allows communication over different software packages (e.g. a BCF issue created in Solibri can be opened in BricsCAD BIM), and that it enables easy navigation through a model based on issues.

20.6.1.2 BCF files

Issues can be saved in standalone files. This is generally a more cumbersome workflow than working with a cloud system but it can be useful in case you are working offline. A .bcf file can then be imported in any BCF manager (e.g. BricsCAD BIM) to read the issues and see where in the model it applies.

Note that a .bcf file cannot be created in BricsCAD BIM. For this, you will need access to another service e.g. Solibri Model Checker or BIMcollab ZOOM. This software allows you to create a .bcf file based on one or more .ifc files. You can then create issues containing:

- Comments
- Screenshots or other images
- A list of involved objects, based on GUID (Global Unique IDs) to distinguish them unambiguously
- A camera position

20.6.1.3 Using the cloud

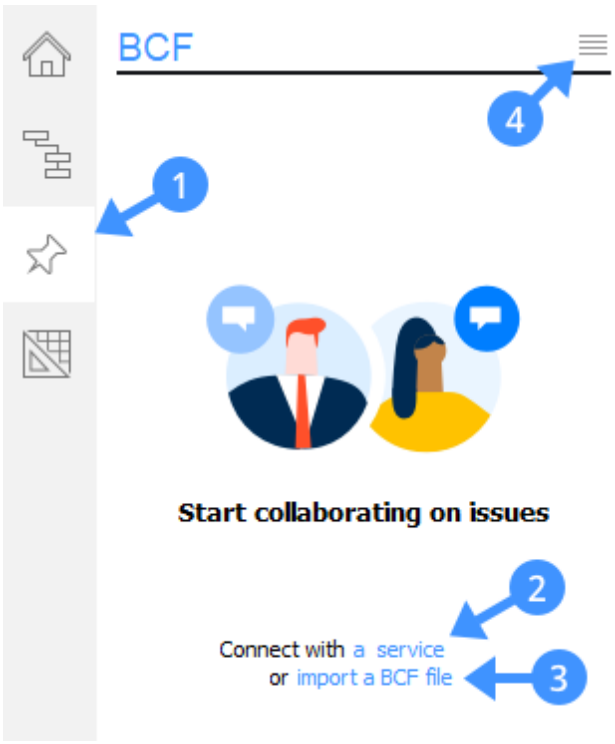
It is generally easier to use cloud services to manage BCF issues. Currently, BricsCAD BIM supports connection to three such services: **BIMcollab**, **BimSync**and **BIMtrack**. If you wish to connect to these services from within BricsCAD BIM you will need an account of these services first. These allow you to create issues and manage them in real-time from anywhere.

Note that it is not possible to **create** issues within BricsCAD BIM. The BCF Panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

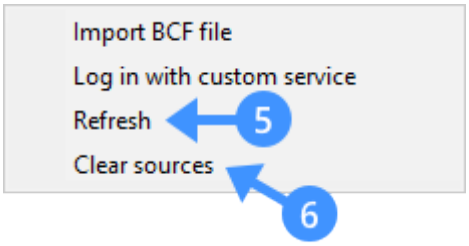
20.6.1.4 The BCF panel

The **BCF** panel can be opened by clicking the board pin icon on the left of your screen (1). If the icon is not there, right-click an empty space in the ribbon and under **Panels**, enable the BIM **BCF** panel.

Using the panel, you can either **Connect with a service** (2) or **import a BCF file** (3), as stated above. The hamburger menu at the top (4) gives you the same options, as well as refreshing the panel and clearing the sources.



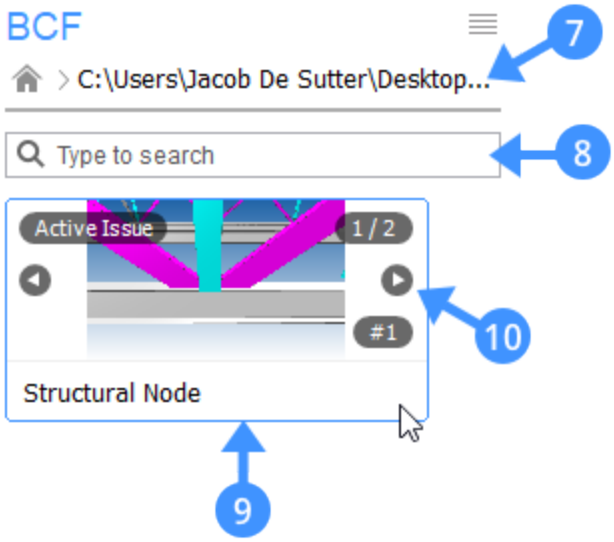
Refreshing the panel (5) will make sure you see the latest updates in case you are working on the cloud. **Clearing the sources** (6) will disconnect you with the cloud service in case you are online or close the .bcf file in case you are working on an imported .bcf file.



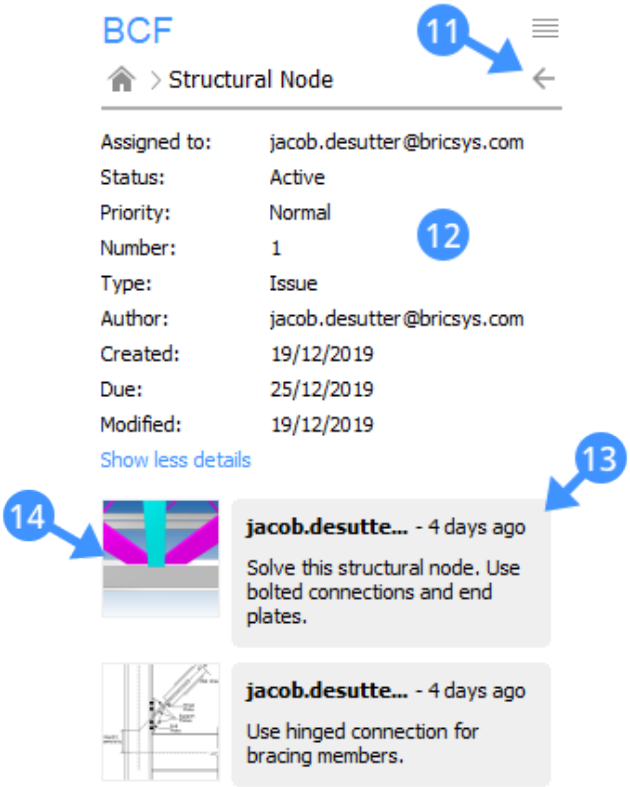
20.6.1.5 Procedure: working with the BCF panel

The following example will be shown with the files attached at the end of this article. If you want to follow along, open **West_str.dwg** and import **Structural_node.bcf**.

When importing a .bcf file you will see the path of the currently opened file (7), a search bar that can be used to filter on issues (8) and a list of issues. In the case of this imported .bcf file, there is only one issue (9) with two associated views (10).



Click on the issue to open it. When opening an issue, you will see a button to return to the issue overview (11), some more detailed information about the issue such as status, creation date and so on (12) and different comments on this issue (13). If this comment has an associated camera position, then clicking the thumbnail image (14) will take the camera in the current drawing to this position. This is useful if you want to know where exactly the issue is and you don't want to manually search through the model.

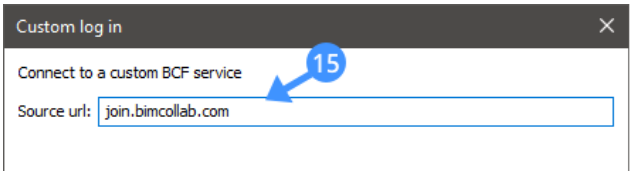


When using a cloud service, you are presented with some more options, such as setting statuses and adding comments and screenshots.

To connect to a cloud service, click the **Connect with a service** button (2). You are prompted to enter a **Source URL** (15). The correct Source URL's for the respective services can be found in the table below.

Cloud service	Source URL
BIMcollab	*<space name>*.bimcollab.com *
BimSync	bcf.bimsync.com
BIMtrack	bcfrestapi.bimtrackapp.co

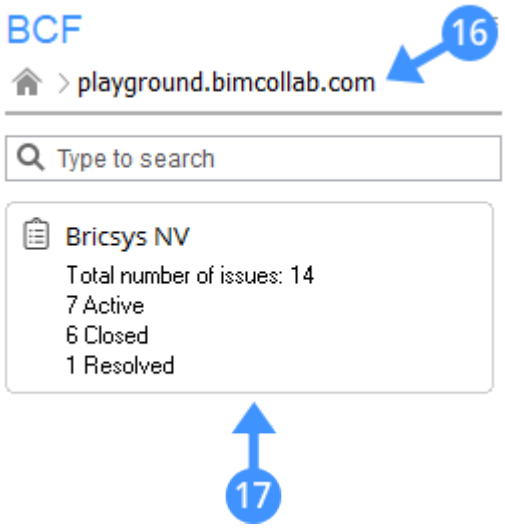
* Note that the Source URL for BIMcollab includes the name of the space you bought. If you are using a free account, use *join.bimcollab.com* instead.



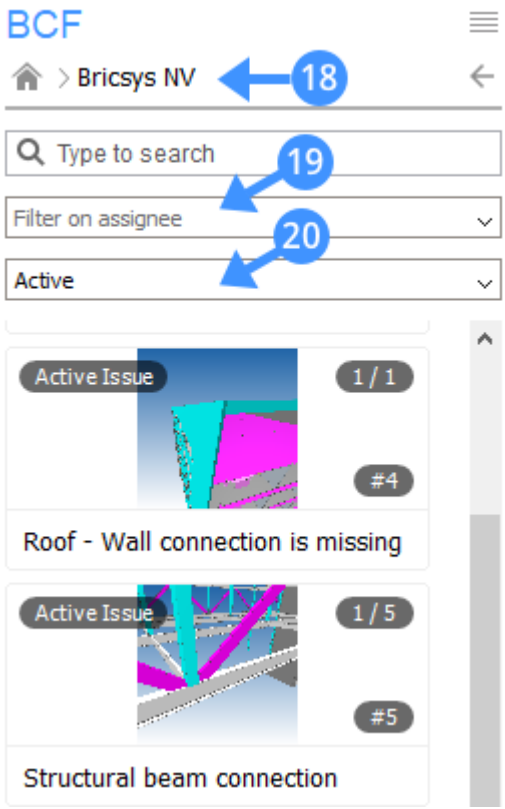
The following examples will be shown on an internal cloud space so it will not be possible to follow along with these steps exactly. It is possible however to create a free account on, for example, BIMcollab.

Clicking the **Log In** button should take you to a login page of the chosen service.

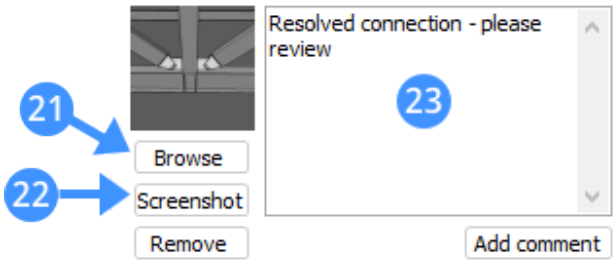
After logging in, the source URL should be shown (16), along with a list of projects that you are involved in (17). In this example, only one project is shown with 14 issues. Left-clicking the project will open it and show a list of the issues, similar to what we saw when importing the .bcf file.



At the top of the panel, the current project is displayed (18). Note that now it is also possible to filter the issues on assignee (19) and status (20). This makes it easy to quickly find the issues that are relevant to you.



Left-clicking an issue will open it and you will see similar options as before: detailed information about the issue (12) and comments on this issue (13). However, note that at the bottom it is also possible to add bitmap images (21) or a screenshot from model space (22). You can add some textual comments as well (23).



Structural_node.bcf

West_Str.dwg

20.6.2 Bricsys 24/7 panel

20.6.2.1 About Bricsys 24/7

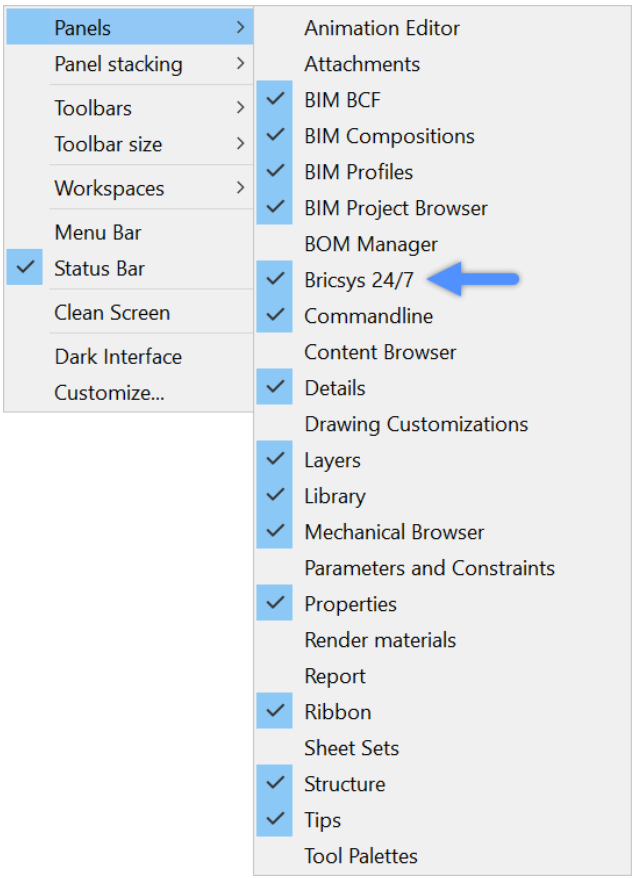
Bricsys 24/7 helps team members to find and share their drawings. Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

20.6.2.2 Use the Bricsys 24/7 panel in BricsCAD BIM

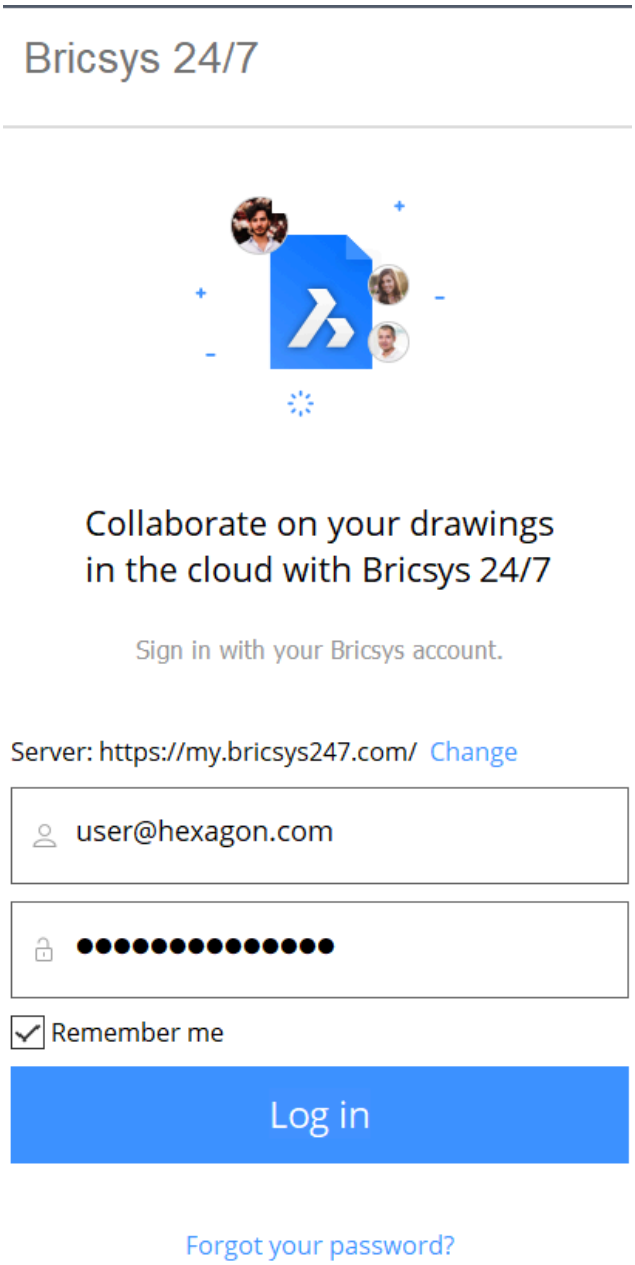
1 Open the **Bricsys 24/7** panel by clicking on the Bricsys 24/7 icon in the **Tool** panel on the right-hand side of the workspace.



Nota: If you do not find this icon in the **Tool** panel, right-click in the **Tool** panel and check **Bricsys 24/7** in the drop-down menu.



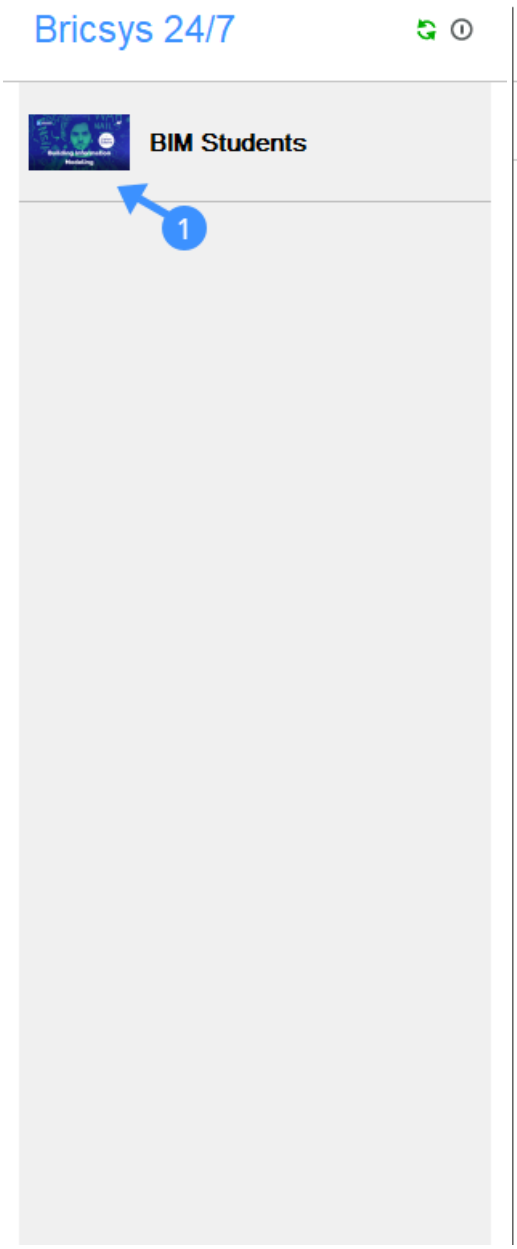
If you are logged out of your 24/7 account a screen will appear:

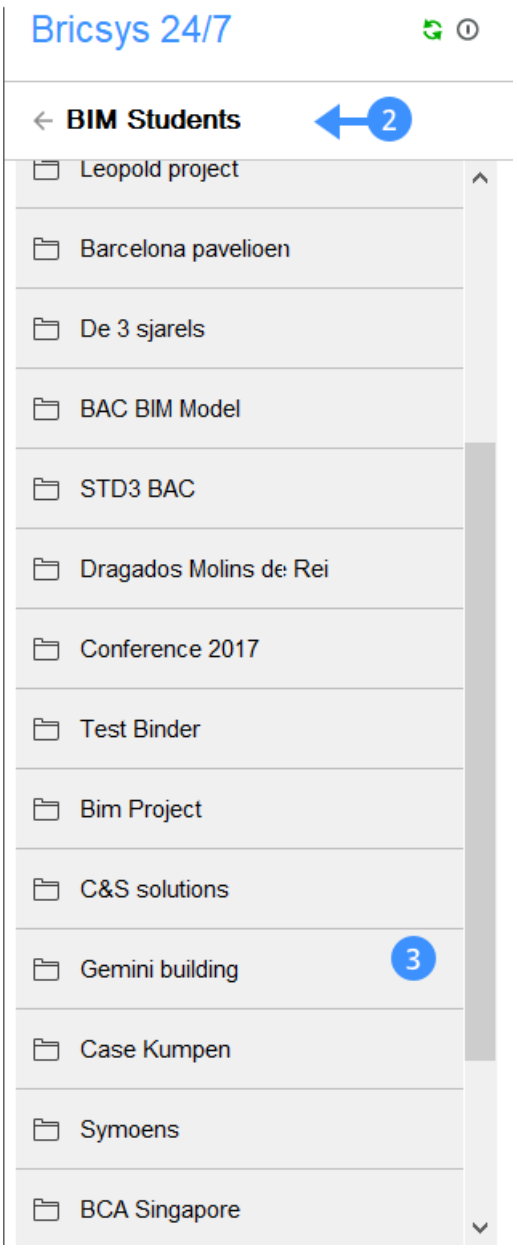


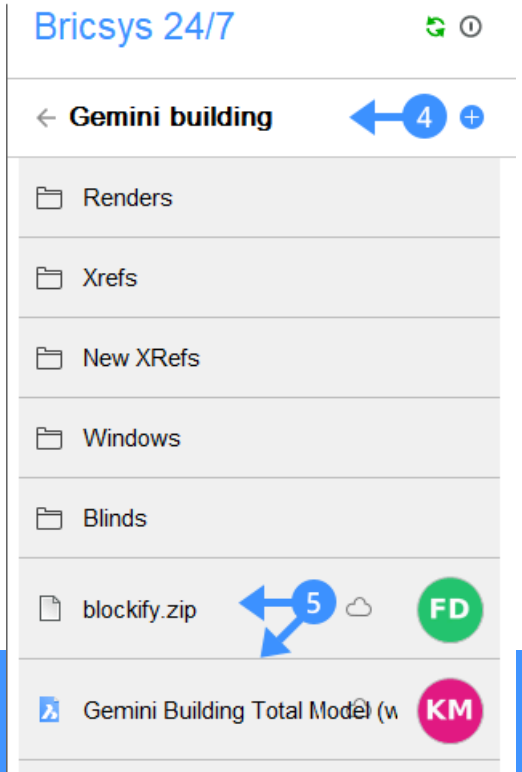
2 Enter your email address and password and press the **Log in** button.

3 (Option) Tick the **Remember Me** checkbox to store username and password.

4 Once logged on you will see an overview of your 'in the cloud' Projects, Folders, and Documents.

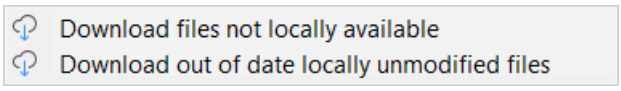

Bricsys 24/7
BIM Students
1


Bricsys 24/7
BIM Students
2
Leopold project
Barcelona pavelioen
De 3 sjarels
BAC BIM Model
STD3 BAC
Dragados Molins de Rei
Conference 2017
Test Binder
Bim Project
C&S solutions
Gemini building
3
Case Kumpen
Symoens
BCA Singapore


Bricsys 24/7
Gemini building
4
Renders
Xrefs
New XRefs
Windows
Blinds
blockify.zip
5
FD
Gemini Building Total Model (w
KM

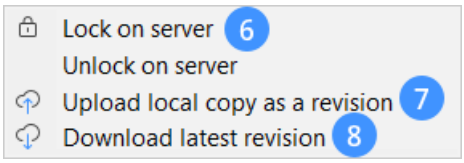
(1)	**project**	The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.
(2)		At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.
(3)	**folder**	When you open a project, a list of folders is shown.
(4)		At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it.
(5)	**documents**	In a folder you find documents. On the right-hand icons are displayed: The profile picture or the initials from the author of the document. The document is in the cloud. Someone is currently working on the document. The document is currently uploading. The document is locked by someone, another person cannot change it.

Right-click on a folder to get a menu with 2 possible actions:



You can download a BIM model from your Bricsys 24/7 project to a local folder.

5 Right-click on a document to get a menu with possible document actions:



(6)	Lock the document so no changes can be made.
(7)	Upload your drawing as a revision of the original drawing.
(8)	Download the latest revision.

20.6.3 IFC import and export in BricsCAD

20.6.3.1 Commands

XREF, XATTACH, XOPEN, XCLIP

20.6.3.2 About

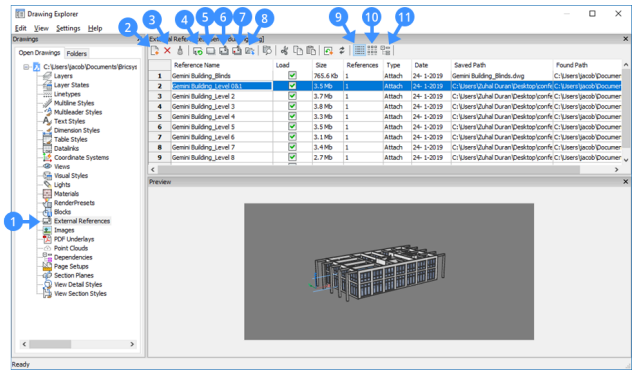
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. 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.

Nota: 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.

20.6.3.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.

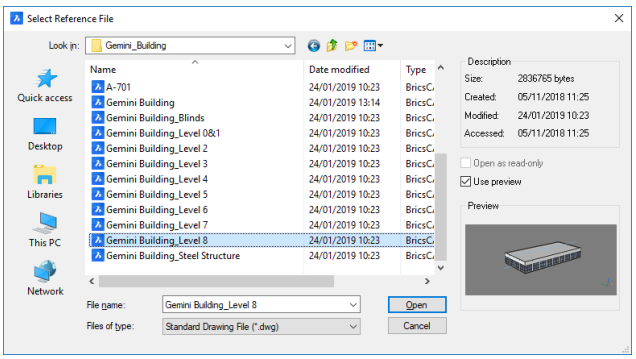


External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and 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 the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

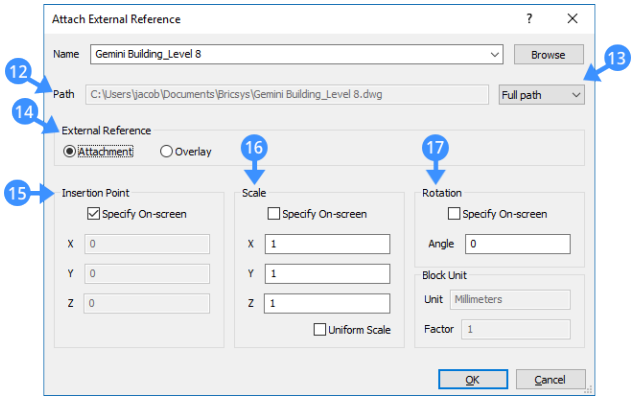
The **Select Reference File** dialog box displays.

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**.



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**.



Path field (12)	Displays the full path of the selected drawing.
Path type (13)	Includes **Full path**, **Relative path**, 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)	Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields.
Scale (16)	Allows the user 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 the user to specify the Rotation angle on the screen or in the **Angle** field.

Note that the **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.



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.

Nota: 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.

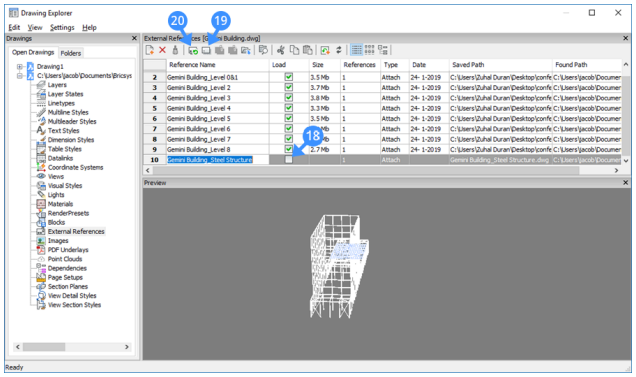
- Click the **Load** (18) column of a loaded XRef.

  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 (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.



5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **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 appear after you close the **Drawing Explorer**.

20.6.4 Import and underlay Autodesk® Revit® files in BricsCAD

20.6.4.1 Commands

IMPORT

20.6.4.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

20.6.4.3 About Revit import

The Revit import function allows you to import .RVT files into BricsCAD as 3D solids or as a geometry underlay for referencing purposes. However, any project data and model metadata, i.e. Constraints, Wall Type structure assembly, materials etc., from the Revit model are not imported.

You can also choose to import selected geometries by 3D Views, by Family Categories, by Levels or by View Worksets, depending on how the Revit project file has been set out.

If you import the RVT file as a solid, only the geometry will be imported with no BIM metadata attached to it. The solids can later be classified and edited with direct modelling tools in BricsCAD.

If you import the RVT file as an underlay, almost like how an XREF is used, the BricsCAD model performs faster and smoother generally. This process also takes a shorter time than importing as a solid. However, the underlay model becomes an OdRvtEntity, like a non-editable, singular block. This method is particularly

useful if you are looking to import the Revit model only for reference and coordination purposes, i.e. importing a Revit structural model to look for clashes with your architectural / MEP model within BricsCAD.

BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the IFC Import and Export in BricsCAD article.

20.6.4.4 Procedure: importing a Revit file as a solid in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **I** and Press Enter to select Import.

5 **You are prompted:** Choose elements [**A**ll/by **V**iews/by **C**ategories/by **L**evels/by **W**ork**S**ets]:

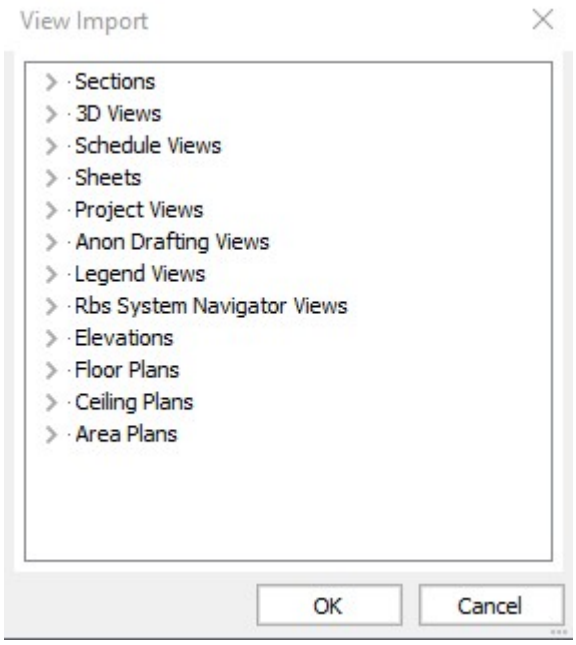
There are different filters to import a file. Select the filter you want to use by clicking into your desired option in the command-line or by hitting their respective letters in bold.

- **All**

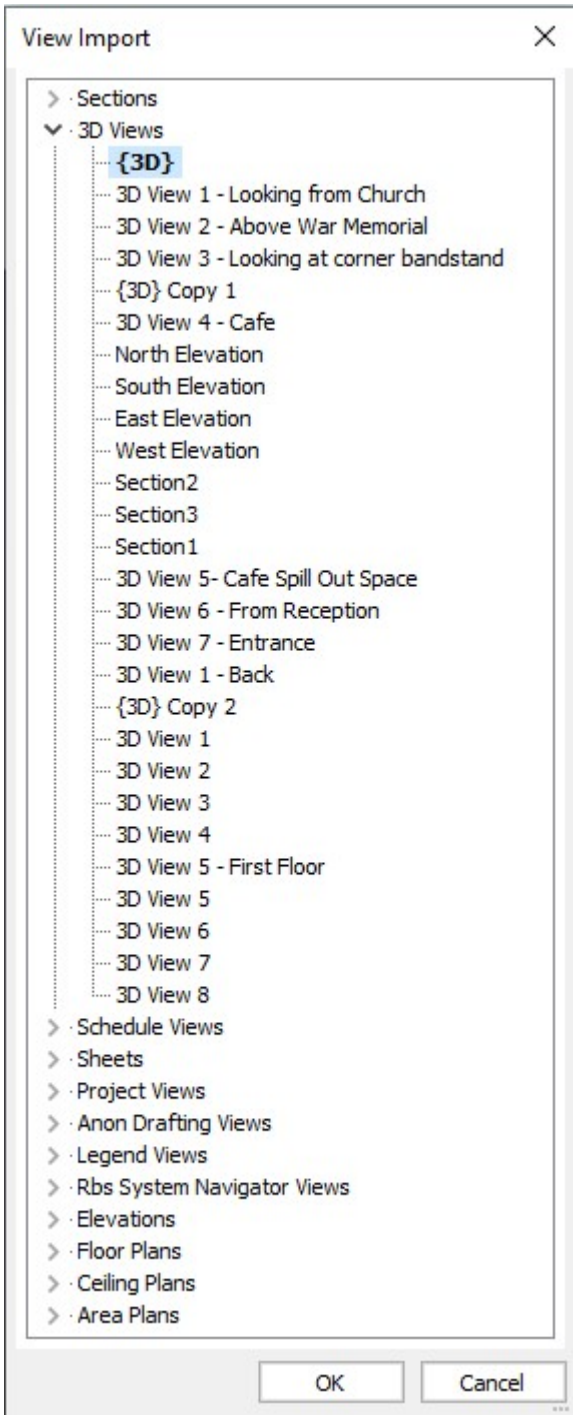
  All elements present within the RVT file will be imported.

- **By Views**

  Select a view from the **View Import** dialog box and the elements will be imported based on the view's Visibility/Graphics Overrides settings in Revit. The views are structured based on their View Types.



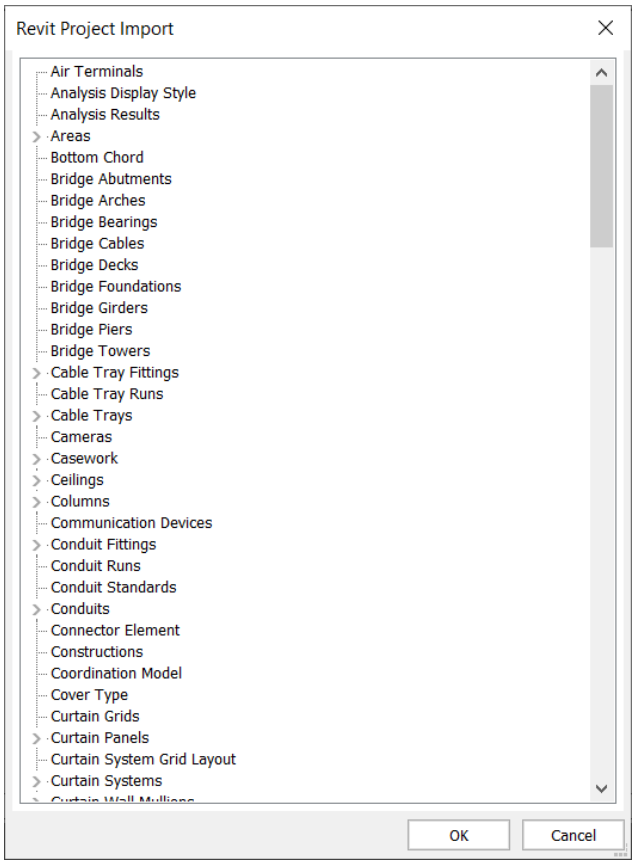
You can expand any relevant or applicable View Type category for the specific views. It could look like this, if the **3D Views** category is expanded:



Nota: Importing elements via a 2D view, ie. Floor Plans etc., only brings elements that are made visible in the Visibility/Graphic Overrides in the Revit model settings for that particular view. It does not necessarily bring elements hosted onto its corresponding level, if they are made hidden on the plan view.

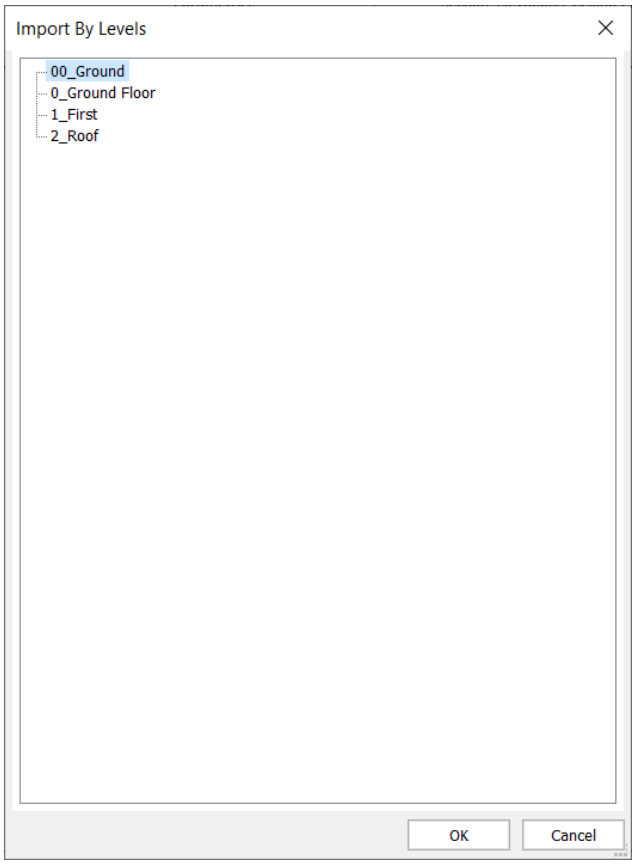
- **By Categories**

  You can select one or multiple built-in Categories and Element Types from the **Revit Project Import** dialog box. Elements of those chosen categories and element types will be imported.



- **By Levels**

  You can select one or multiple levels from the **Import By Levels** dialog box. Elements hosted to the chosen levels will be imported.

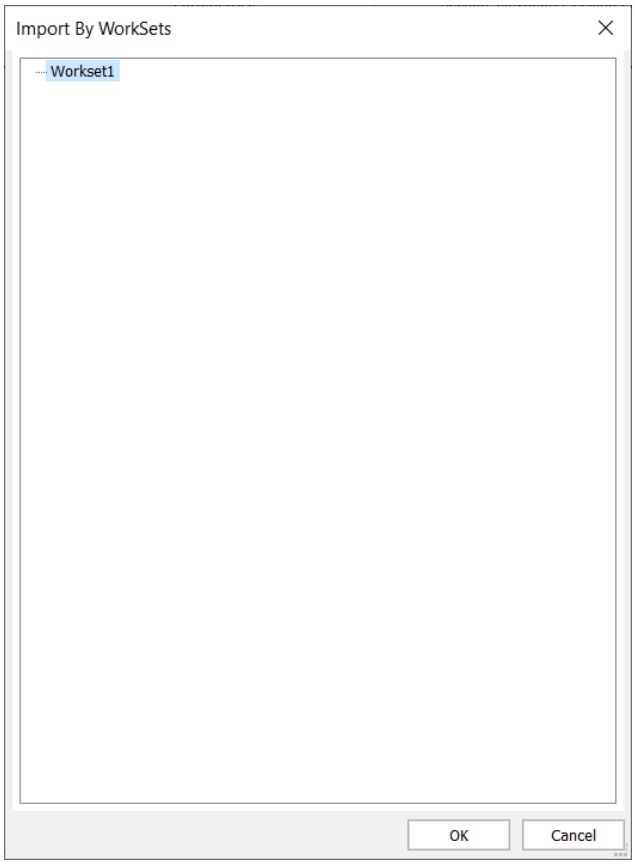


- **By WorkSets**

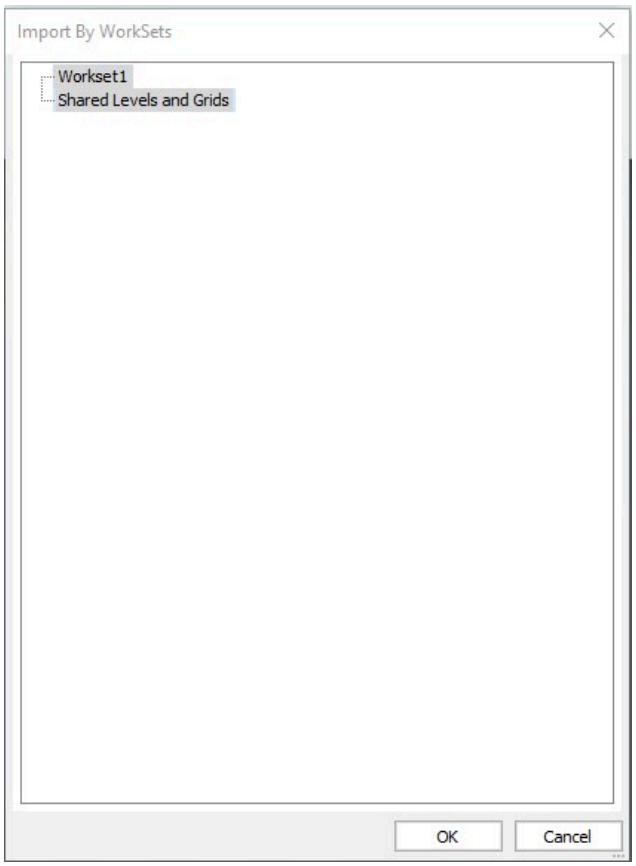
  You can select one or multiple worksets from the **Import By Worksets** dialog box.

  This method of importing works best on a workshared Revit model. It is recommended you import a Central Model to ensure you are importing the latest information in the worksets. Detaching the Central Model is optional.

  If you import a non-workshared Revit model, only **Workset1** is available.



If you import a workshared Revit model, either Central or Local, the User-Created worksets within the model will be available in the dialog. The elements which belong to this workset will be imported.



Nota: The **Shared Levels** and Grids and **Workset1** worksets are created in Revit by default when Worksharing is enabled and can be managed by the Revit user.

6 If you have chosen to import by view, you do not have to specify the level of detail. The Detail Level of the selected view set in Revit will be used.

7 For the rest of the other options,

8 **you are prompted:** Import Elements with Detail Level [Fine/Medium/Coarse]:

Select the level of detail you want. Type **F** in the command-line for fine, **M** for medium and **C** for coarse and press Enter.

Nota: When selecting the level of detail, make sure you are aware of how the incoming geometries have been set out in terms of Detail Levels within the Family Elements Visibility Settings in Revit.

9 Select the View, Categories, Levels or WorkSets you want to import and click **OK**. The RVT file will be imported as 3D solid geometries.

10 Now, you can classify the solid manually or automatically and/or further edit it with direct modeling tools in BricsCAD.

Nota: Some of Revit's system families, such as walls and floors etc., are imported into BricsCAD as 3D solids, where you can further manipulate their geometries directly. This also applies to Model-In-Place components and Element Types, such as Fascias, that can be defined geometrically by a Profile.

However, other system families designed with in-built constraints, i.e. stairs and curtain walls etc.

are imported into BricsCAD as separate Block References. For eg., Curtain Panecl and Curtain Wall Mullion Types, which make up a Curtain Wall Type in Revit, will exist as separate Block References. This also applies to all other loadable families, like furniture, placed within the Revit model. For more information, please visit the article on Working with Blocks.

20.6.4.5 Procedure: importing a Revit file as an underlay in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops-up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **U** and Press Enter to select **Underlay**.

5 Follow Steps 5-7 of the 'Procedure: importing a Revit file as a solid in BricsCAD' above.

6 **You are prompted**: Deviation value:

By default, the deviation value is 0.001. Press Enter to accept **or** type a new value and press Enter.

Nota: The deviation value defines the accuracy of the imported model. A lower value will result in a finer geometry, but it will take longer time to load.

7 The RVT file is now imported as a non-editable OdRvtEntity.

20.6.4.6 Procedure: importing a Revit family in BricsCAD

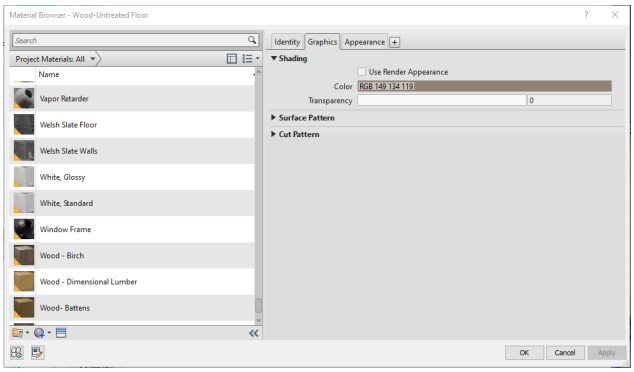
Please refer to the article **Importing Revit RFA Families in BricsCAD** for more information.

20.6.4.7 Texture and graphics of the imported geometries

Although project information and element metadata in a Revit model cannot be brought into BricsCAD (just yet), there are some aspects of the imported 3D solids, i.e. Material Color, Transparency, which can be brought over and therefore correspond to the original graphics set in the Revit.

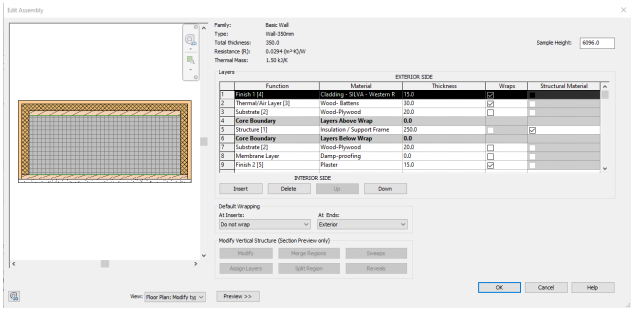
To retain the consistency of the graphical information during the import process, ensure your RGB color and transparency settings set in Revit's Material Browser are set to your desired values.

In the example below, the values taken will be **RGB 149 134 119** and **0** (%) for the colors and transparency values respectively.



If the imported element is a Revit System Family with the ability to define its structure assembly, i.e. Wall Type, and is made up of various layers of materials or functions, BricsCAD applies the color template values from the outermost materials (on both the Exterior and Interior sides) accordingly on to the corresponding geometry's faces.

Let's take an example from the illustration below, the values will be derived from the materials assigned on Layers 1 and 9, which are **Cladding...** and **Plaster** respectively.



Otherwise, BricsCAD applies the color value, and the transparency settings if applicable, to the 3D solid replacing the respective parts of the Revit Family. This also applies to those Loadable Families and Components, having loaded into the imported Revit project.

20.6.5 Importing Autodesk® Revit® families in BricsCAD

20.6.5.1 Commands

IMPORT, BMINSERT

20.6.5.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

20.6.5.3 Important notification before getting started

RFA import is a work-in-progress. See the BMINSERT command.

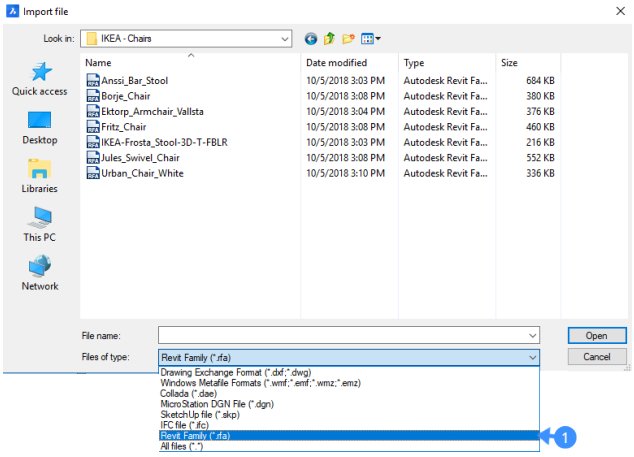
BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the **IFC import and export in BricsCAD** topic.

20.6.5.4 Importing RFA family files

1 Launch the Import command.

1 The **Import file** dialog box displays.

2 Select **Revit Family (*.rfa file)** (1) in the **Files of type** list.



3 Select the .rfa file you want to import.

4 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.

20.6.5.5 Procedure: inserting RFA family files

Nota:

- When DUCS = ON (Dynamic UCS) the RFA entity aligns with the 3D solid face under the cursor.
- When DYN = ON (Dynamic Dimensions) the distances from the nearest parallel edge on the face of the 3D solid display.

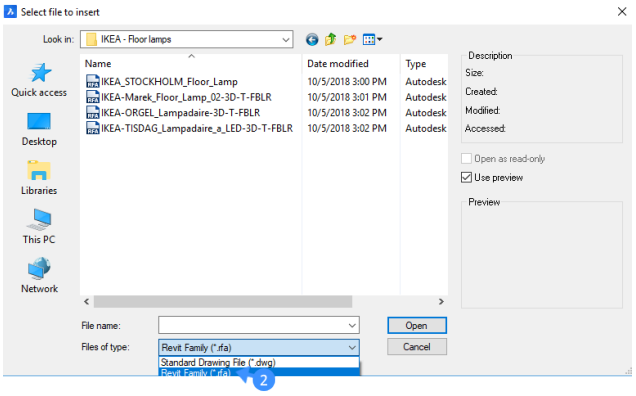
Using BmInsert

1 Launch the BMINSERT command.

1 A **File** dialog box appears.

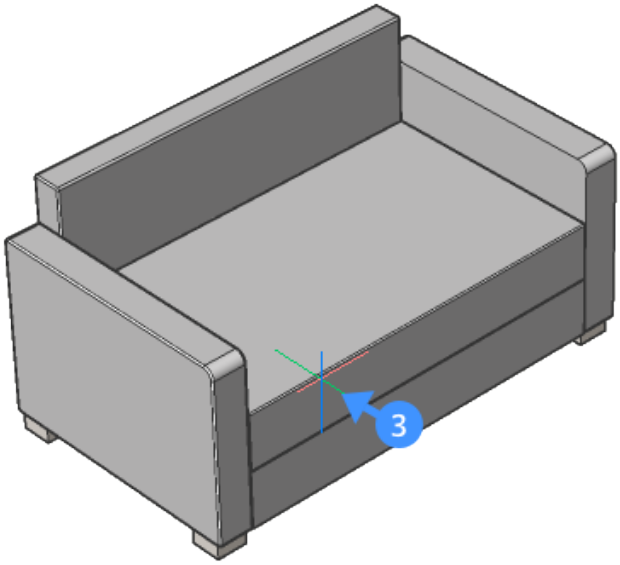
2 Browse to the folder where your .rfa family files are saved.

3 Select **Revit Family (*.rfa file)** (2) in the **Files of type** list.

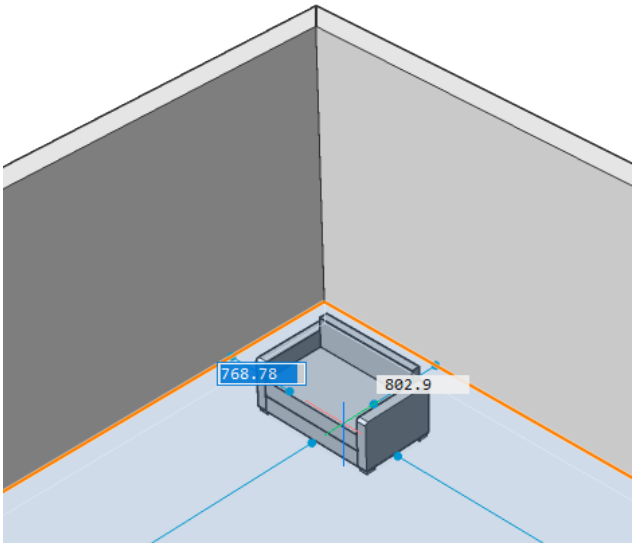


4 Select the .rfa file you want to import.

5 The selected file is attached to the cursor (3).



The following illustration shows inserting a Revit family sofa when both DYN and DUCS are on.



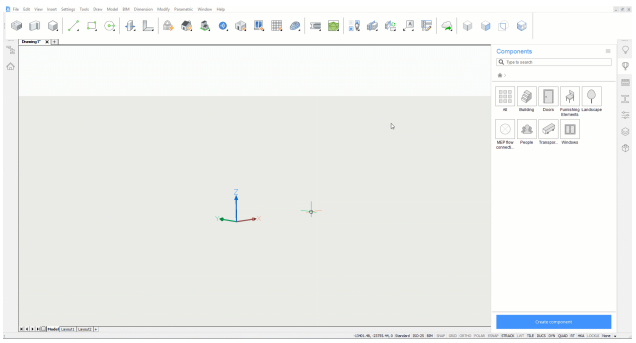
6 Do the following to place the selected RFA family file using the dynamic dimension fields:

- 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.
- Repeat the previous step to lock another dynamic dimension field.
- Optionally hit the Tab key to adjust the dynamic dimensions.
- Press Enter to accept the current position.

Using the Library panel

1 To insert a RFA family folder in the **Library** panel, add the RFA root folder path to the Library directory path in the **Settings** dialog box. The content of the root folder and all subfolders will be available in the **Library** panel.

The following animated gif shows how to add the RFA root folder path to the components directory path.



The RFA family files are ready to insert in the model space.

If necessary, choose to **Generate Thumbnails** in the hamburger menu on the **Library** panel.

2 Drag and drop the RFA file from the **Library** panel into your model space.

20.6.6 Interference checking

20.6.6.1 Commands

INTERFERE

20.6.6.2 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 **InterfereLayer** variable. Running an interference check on existing models allows you to prevent the conflicts in your entire model.



For more information about this command, visit the Command Reference article INTERFERE.

20.6.6.3 Procedure: performing an interference check

1 Select the desired 3D solids to be checked for interference, then launch the INTERFERE command.

Nota: 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 the 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.

3 The entities in the first set are compared with the entities in the second set.

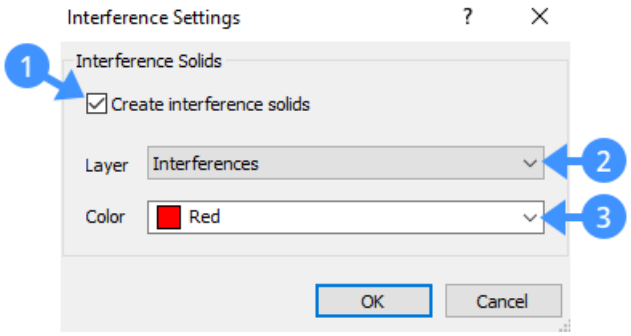
Nota: If you press the **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

4 Press **Enter** to run the interference check.

5 The interference volume is created for each collision and it is shown in a solid model.

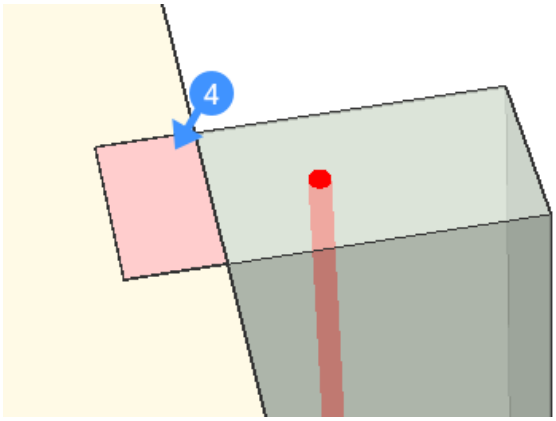
Nota: The solids of the interference volume are displayed in red since they are placed on layer **Interferences**. The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the Command line. 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.



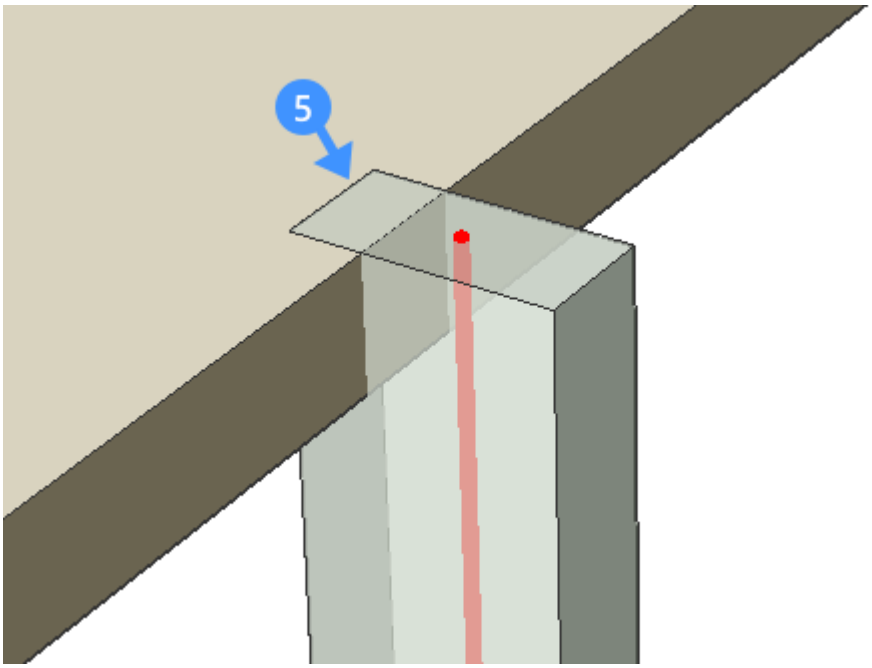
Create interference solids box (1)	Determines whether a solid or region of the interference is created. Check to create a solid of the interference volume or region of the interference area.
Layer (2)	Shows the current layer for interferences.
Color (3)	Shows the current layer color.

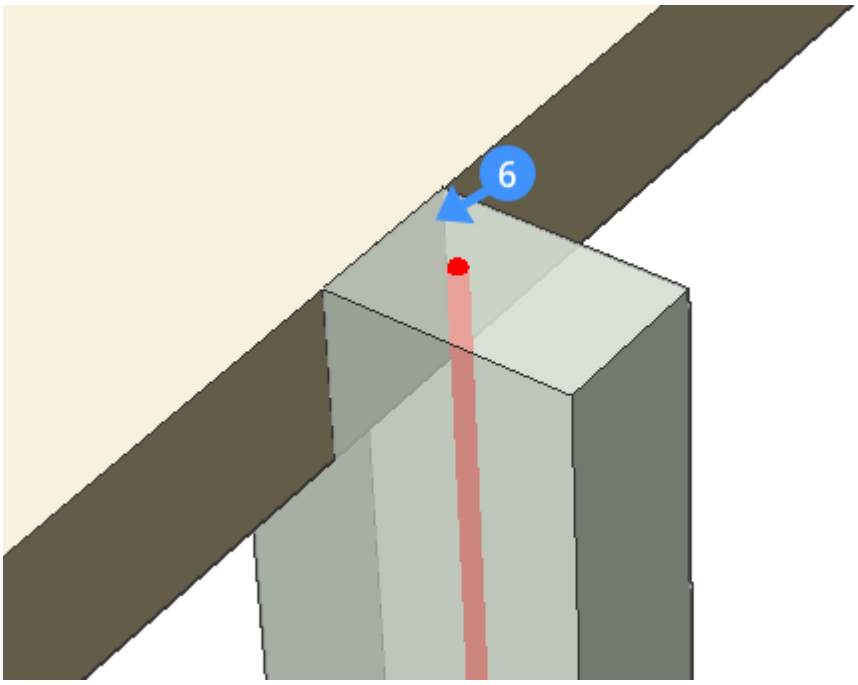
The following illustration shows the detected collision (4) between the floor slab and column.



6 To remove the undesired intersections, turn off the **Interferences** layer, and use 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.

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.





7 Run the INTERFERE command a second time to check whether all interferences are fixed.

8 If there are no collisions left, the Command line reports that "**Solids do not interfere**".

20.6.7 Using Xrefs

20.6.7.1 Commands

XREF, XATTACH, XOPEN, XCLIP

20.6.7.2 About

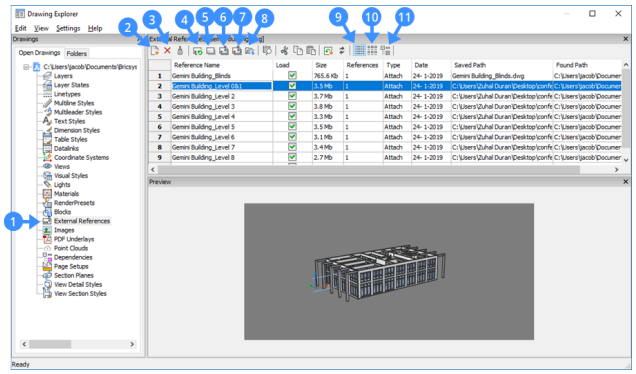
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. 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.

Nota: 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.

20.6.7.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.



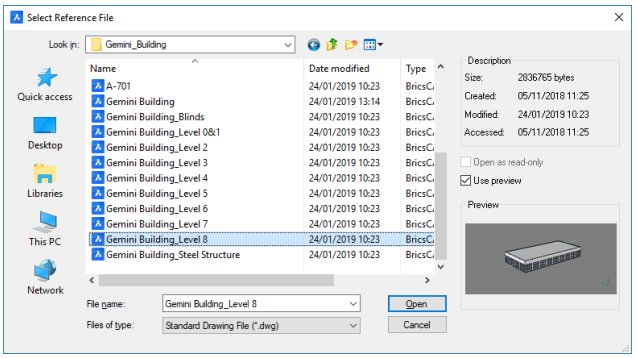
External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and 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 the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

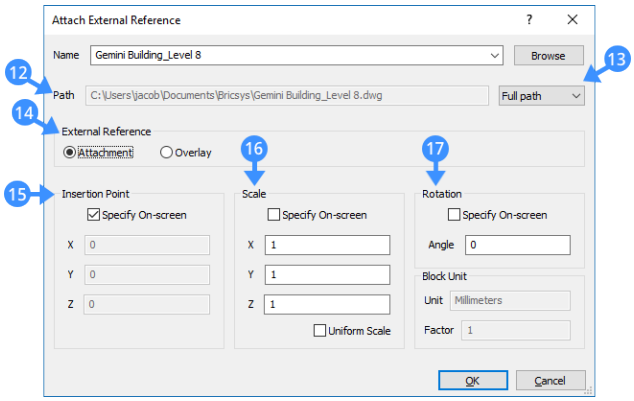
The **Select Reference File** dialog box displays.

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**.



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**.



Path field (12)	Displays the full path of the selected drawing.
Path type (13)	Includes **Full path**, **Relative path**, 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)	Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields.
Scale (16)	Allows the user 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 the user to specify the Rotation angle on the screen or in the **Angle** field.

Note that the **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.



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.

Nota: Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

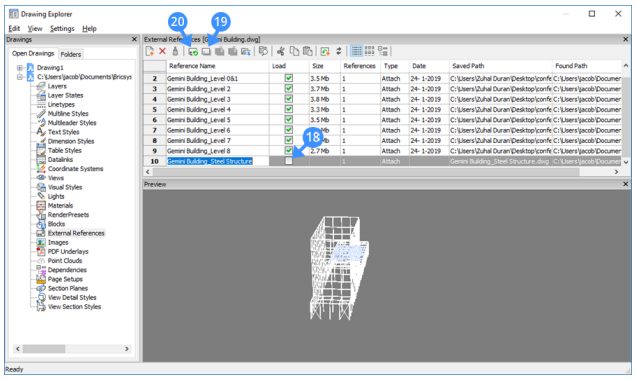
4 Do one of the following to unload and reload an existing XRef.

- Click the **Load** (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 (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.



5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **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 appear after you close the **Drawing Explorer**.

20.7 Documentação do projeto

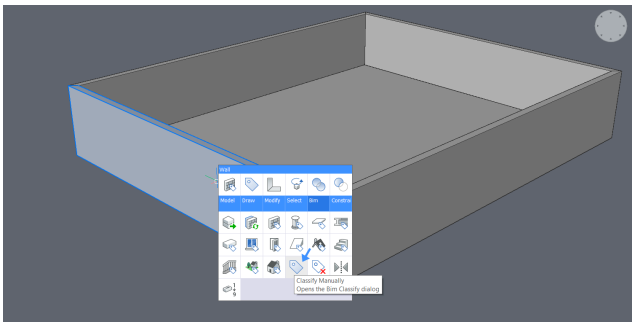
20.7.1 Annotations in 3D

20.7.1.1 Commands

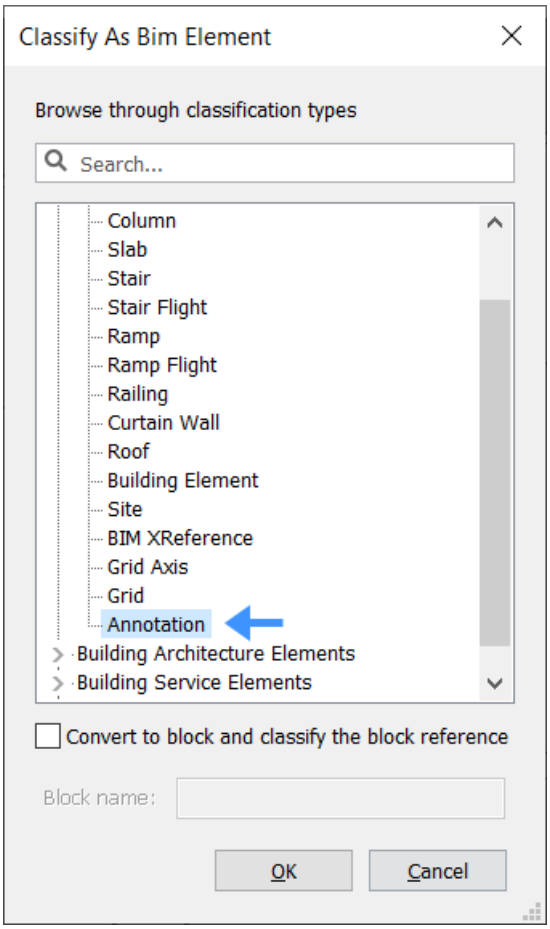
BIMTAG, BIMCLASSIFY

20.7.1.2 BIM classify as an annotation

1 Select a solid and select **Classify Manually** in the Quad or classify as other after launch the BIMCLASSIFY in the Command line:

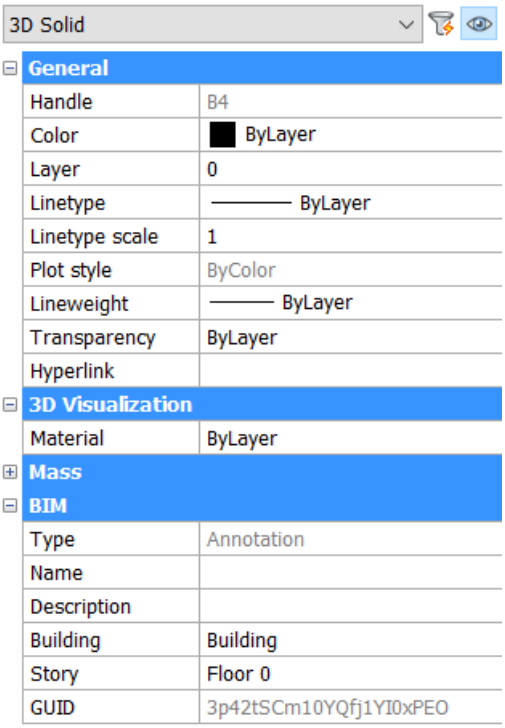


2 Choose to classify as **Annotation** in the folder of **Building Core Elements** and click **OK**:



3 When you open the Properties panel of the selected solid you can change the **Name**, **Description**, **Building** and **Story**.

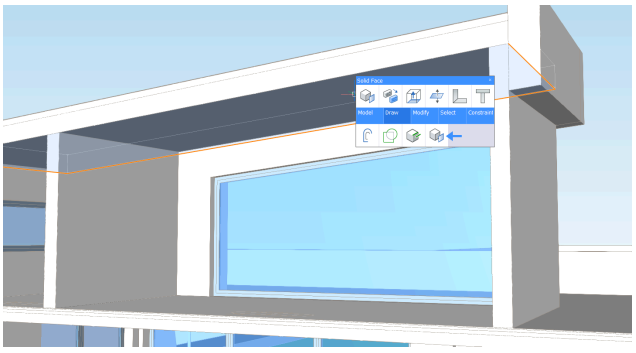
Nota: The property GUID cannot be changed because it has a unique, automatically generated string.



20.7.1.3 Show added information of a finishing material

In this procedure, you will add design information to a concrete roof structure.

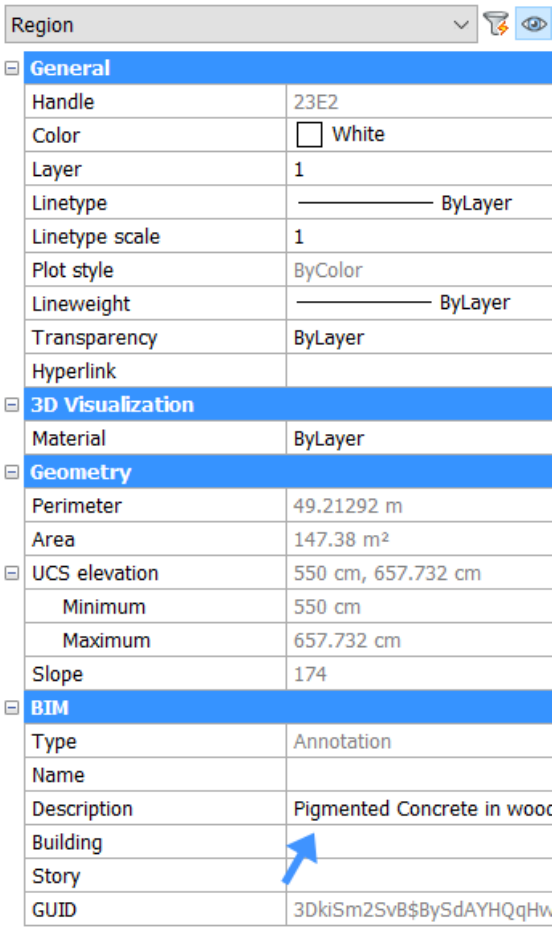
1 Use the EXTRACTFACES command under **Draw** in the quad, to extract the face of the **Roof** entity. A region is now created.



2 Classify the region as an **Annotation** as seen above.

3 Enter the desired text in the **Description** field (found in the Properties panel):

Nota: By default, the BIMTag references the **Description** property.



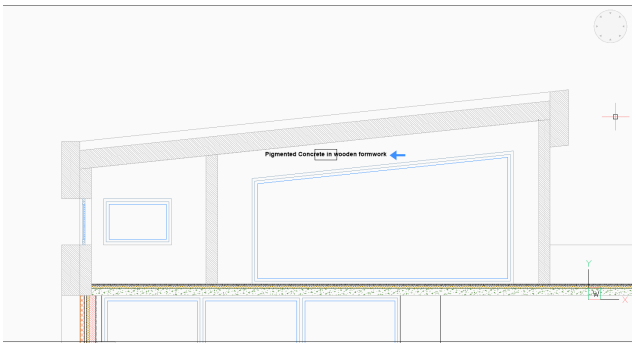
4 Make a section through the region (with BIMSECTION command) and select **Generate Section**( ) from the Quad:



5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

The text that you added is now in the 2D drawing:



20.7.1.4 Show the surface area of an extracted surface

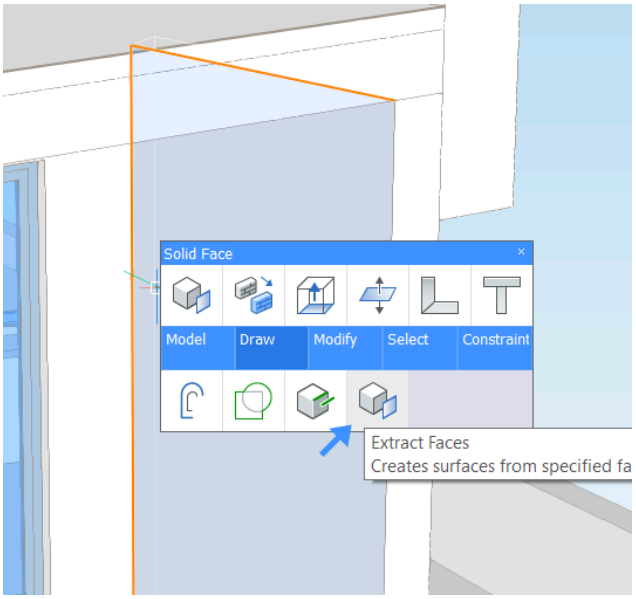
This procedure will teach you how to display other properties of an entity classified as an **Annotation** object.

Nota: The annotation tag pulls information from the **Description** property by default.

For more information about tagging, see the article BIM Tags.

1 Select the face you want to extract the information from.

2 Select **Extract Faces** from the Quad. A region is now created.



3 Classify the region as an **Annotation** as above.

4 Make a section through the solid (with BIMSECTION command) and select **Generate Section** ( ) from the Quad.

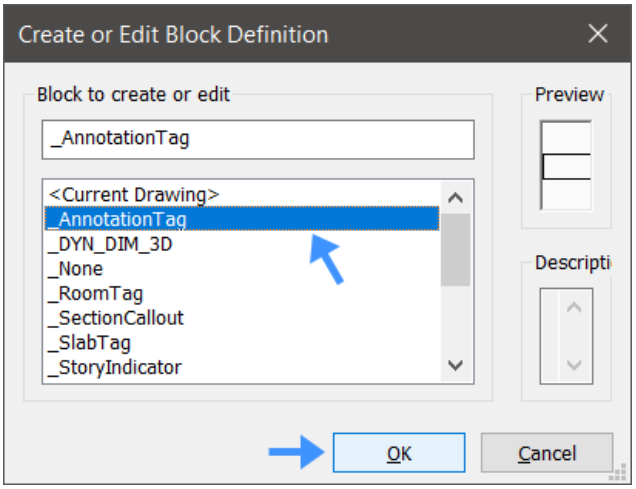
5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

Nota: The tag appears empty because it refers to the **Description** property by default.

7 Launch the BEDIT command.

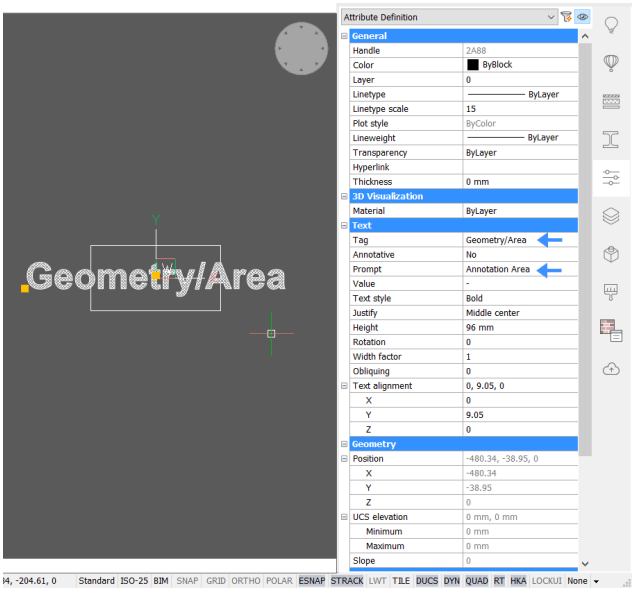
8 Select **_AnnotationTag** and click **OK**.



9 Select the attribute definition, in the properties panel.

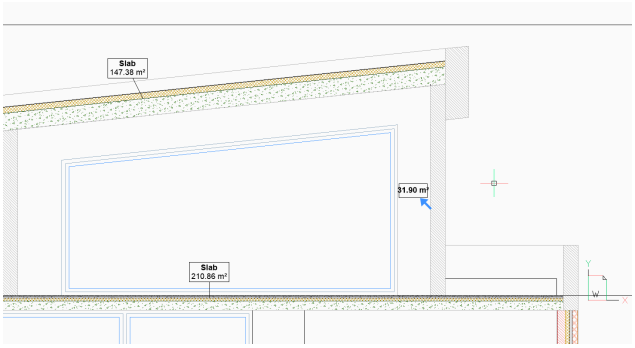
10 Edit the **Tag** property, change it to **Geometry/Area**.

11 **(Optional)** You can also change the **Prompt** property.



12 Enter BCLOSE in the Command line and click **Save**.

13 Select the viewport and pick **Generate Section** ( ) from the Quad. The area of the region is now displayed in the tag.



20.7.2 BIM sections

20.7.2.1 Commands

BIMSECTION

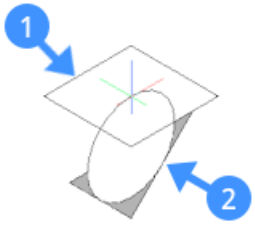
20.7.2.2 Defining a section

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.



For more information about this command, visit the Command Reference article BIMSECTION.

When defining a BIM section, the section tracker is attached to the cursor.



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 direction of the current coordinate system.

Nota:

- 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 a 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).

General procedure to create a BIM section

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

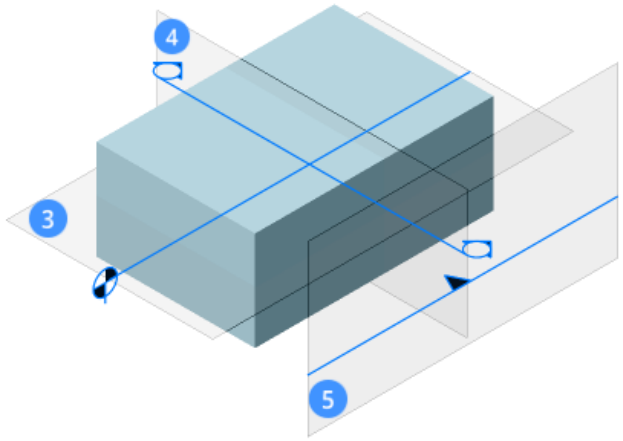
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

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, 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.
   - **(Optional)** Hit the Shift key to lock the highlighted plane. This allows you to start from a point outside the selected 3D solid face. Next, left-click.
3 The initial section plane displays dynamically and the 3D model is clipped accordingly.
4 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 depends on the direction of the section plane (horizontal or vertical) and whether it intersects with the model.



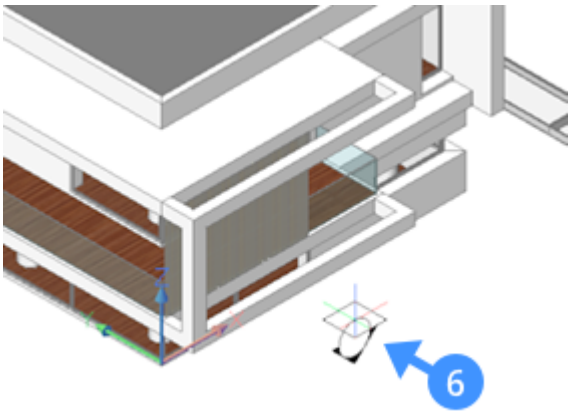
Plan (3)	Displays horizontal section plane.
Section (4)	Displays a vertical section that cuts through the model.
Elevation (5)	Displays the exterior elevation view.

Creating a plan section

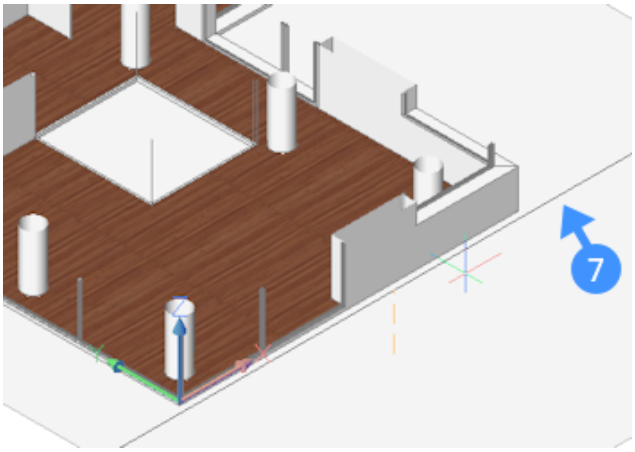
Nota: A plan section is best defined in the WCS (World Coordinate System).

1 Choose **Define Section** in the **Model** tab of the **No Selection 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 (6).



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 (7).



You are prompted: Specify distance:

3 If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

4 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.

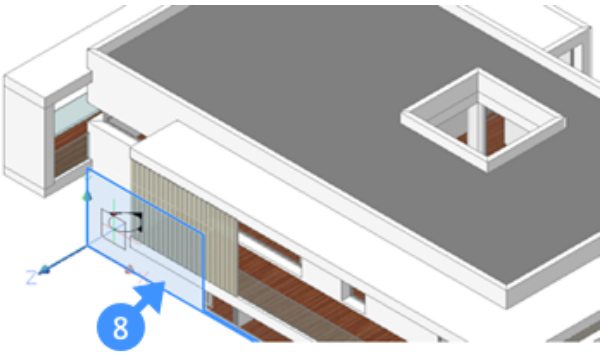
Creating a cross section

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

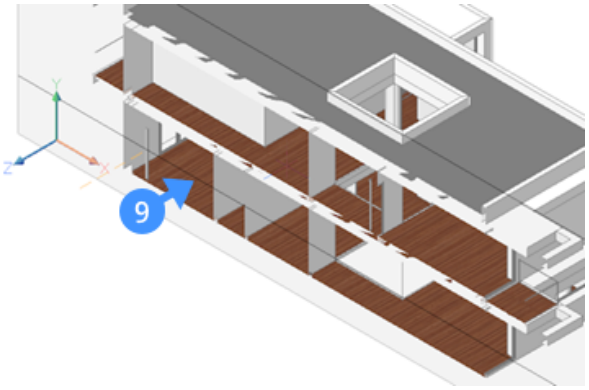
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker over a 3D solid face (8) parallel to the cross-section to be defined.



3 Click when the face highlights and move the section tracker inside the 3D model.

4 The section plane displays dynamically and the 3D model is clipped accordingly (9).



You are prompted: Specify distance:

If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

5 Click when the section plane is at the desired location or specify a distance.

6 The cross-section is defined.

7 **(Optional)** Click the section entity and hold down the left mouse key a little longer to display the **Manipulator**. Drag an axis of the manipulator to move the section.

8 **(Optional)** Click the Arrow grip to flip the view direction.

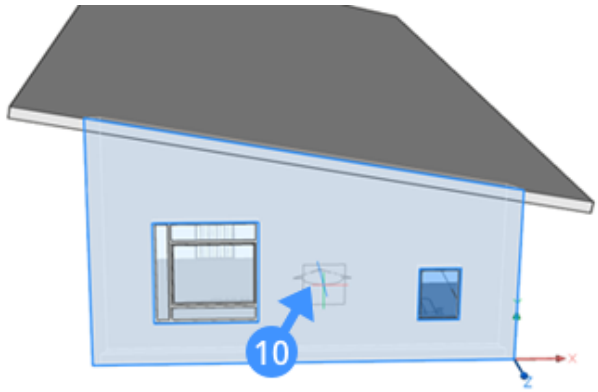
Creating an elevation

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

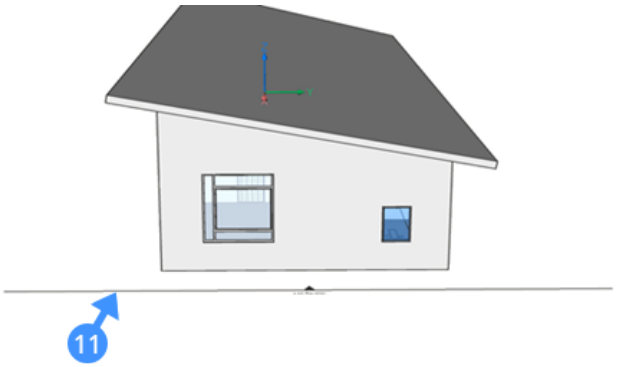
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

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.



3 You are prompted: Specify distance:

4 Move the section tracker outside the 3D model and click to define the elevation (11).



Vertical sections outside the model (no geometry is sectioned) get the Elevation section type automatically.

20.7.2.3 Editing the default properties of a BIM section

The default properties of a BIM section entity are copied from the section entities in _SectionSettings.dwg in the **BIM/Sections** subfolder of the **Support** folder.

1 Select the BIM section in the drawing or the **Structure panel**.

The section properties display in the Properties panel.

2 Define the properties of the BIM section entity.

General	
Handle	14F2
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Material	ByLayer
Section Entity	
Name	Floor 0
State	Plane
Live section	No
Clip display	No
Use lids	Yes
Plane transparency	70
Plane color	9
Destination file	Plans
Geometry	
Elevation	120 cm
Top plane	15.86 m
Bottom plane	15.86 m
Vertex	1
Position	-4122.5, 7000, 1200
X	-4122.5
Y	7000
Z	1200
UCS Elevation	899.57 mm, 139.957 cm
Minimum	899.57 mm
Maximum	139.957 cm
BIM	
Name	
Description	
Section type	Plan
Callout position	Start
Initial scale	1:50
Layer state	
Apply layer state	Off
Project section	On
Generate tags	On
Entity property sets	

Section Entity:

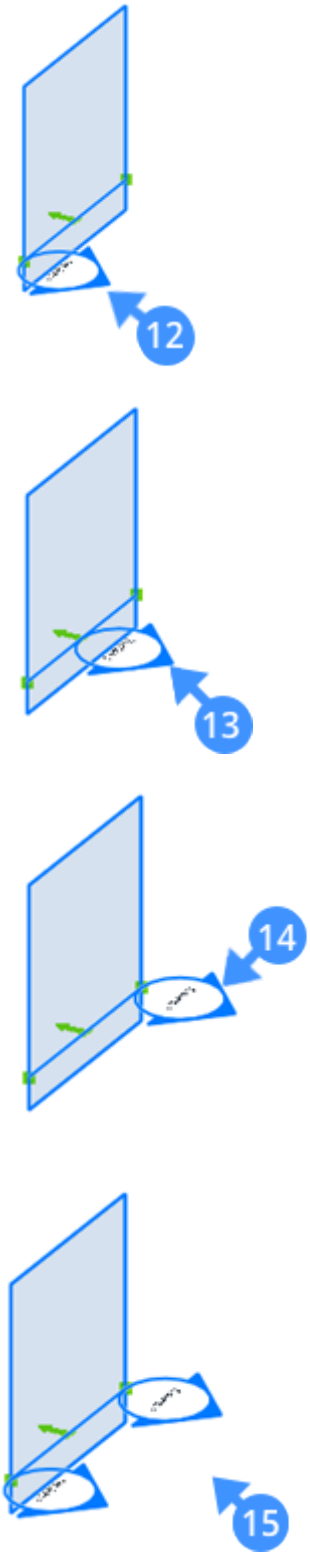
Name	Type a name.
State	Click the field and select a state. The selected state determines how the section result will be clipped. The options are: **Plane** (default), **Boundary** and **Volume**.
Live section	Sets the live section property of the section plane. Only one section entity can have the **Live section** property **ON**.

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 control points of the section plane. The number of vertices depends on the current section plane state. Plane: 2 vertices Boundary and Volume: 4 vertices. Select the field, then click the arrow buttons to select a vertex. An X indicates the selected vertex in the model.

BIM:

Name	Type a BIM name.
Description	Type a description.
Section type	Sets the BIM section type.

Callout position	Defines the position of the callout(s). The callout block and default position depend on the section type. The options are: **Start** (12), **Mid** (13), **End** (14) and **StartEnd** (15).  Click the green arrow ( ) to flip the view direction of the section.

Initial scale	Sets the initial scale of the papers pace viewport in the generated drawing.
Layer state	Sets 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. Applying a layer state allows to control which layers are ON/OFF when calculating the section result.
Project section	Controls the display of the section callouts and section line in section drawings which are perpendicular to this section.
Generate tags	Controls whether BIM tags are generated automatically.
Entity property sets	Controls whether 'per instance' entity properties can be used to generate tags. Click the **Browse** (…) button, then select the property sets in the **Enable per-instance property set** dialog box.

20.7.2.4 How to toggle the clipped display property of a BIM section entity

The **Clip display** property of the **BIM** section temporarily clips 3D models. The **Clip display** can be used in all modeling and editing operations on all drawing content.

The following illustration shows the 3D model of a villa with a plan section (16) for the ground floor plan.

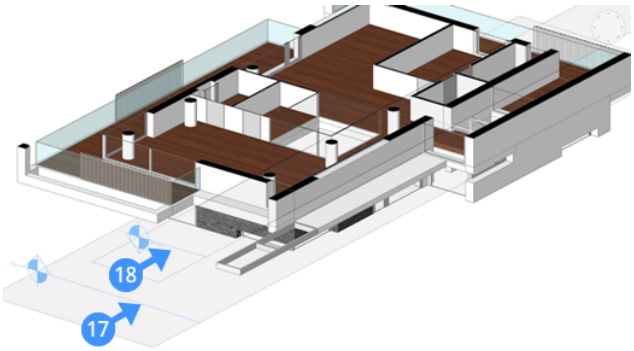


To clip the 3D model, do one of the following:

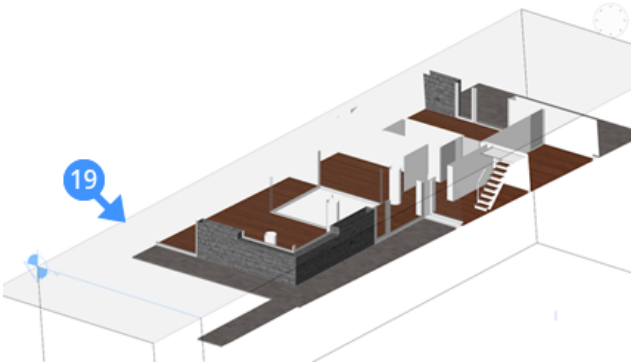
- Double click the section entity.
- Select **Clip display** from the Quad.
- Select the section plane and set the **Clip display** property to **Yes** in the properties panel.
- When **RT** (Rollover Tips) is selected in the **Status** Bar, place the cursor over a section entity to set the **Clip display** property.

Nota: **Clip display** can be on for multiple section entities simultaneously.

In the following illustration, **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.



Option: Set the **State** property for the current section to **Volume** in the properties panel.



A clipped section shown in the **Volume** state (19).

20.7.2.5 Converting a section entity to a BIM section

The BIMIFY command converts a section entity to a BIM section.

1 Launch the BIMIFY command.

1 You are prompted: Run Bimify on entire model or [Advanced] <Model>:

2 Choose the **Advanced** option.

The **Bimify** dialog box appears.

3 Choose **Select entities**, then click the **Select** button.

The **Bimify** dialog box closes.

4 Select the section entity, then right click.

5 The **Bimify** dialog box reopens.

6 Click the **OK** button.

20.7.3 BIM tags

20.7.3.1 Commands

BIMTAG

20.7.3.2 Source files

The following source files are used to create BimTags in a generated BIM section drawing:

- **_SectionTag.dwg**: contains the tag blocks and multileader styles.
- **_TagTypeToStyle.xml**: links a BIM entity type to a multileader style.

Both files sit in the folder:

C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\Bim\Sections

Nota: when assigning tags, BricsCAD needs to access the 3D model. For this reason it may take several seconds to place the first tag, if the 3D model is not currently opened, especially in large models. If you want to avoid this waiting time, consider having the 3D model open while placing tags.

20.7.3.3 Assign BIM tags manually

The BIMTAG Manual command allows you to select the BIM element that you wish to create a BIM tag for.

For more information about this command, visit the Command Reference article BIMTAG.



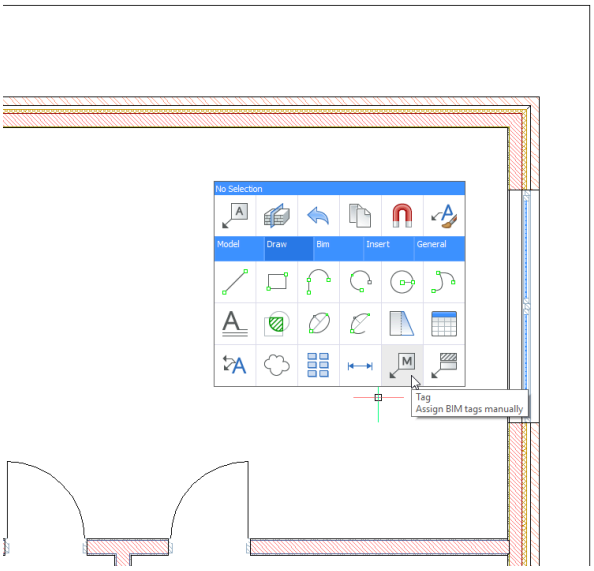
Manually create a BIM tag

1 Open the section drawing for which you wish to create BIM tags.

Nota: For more information about generating section drawings, please read the BIM article **Generating Drawings** first.

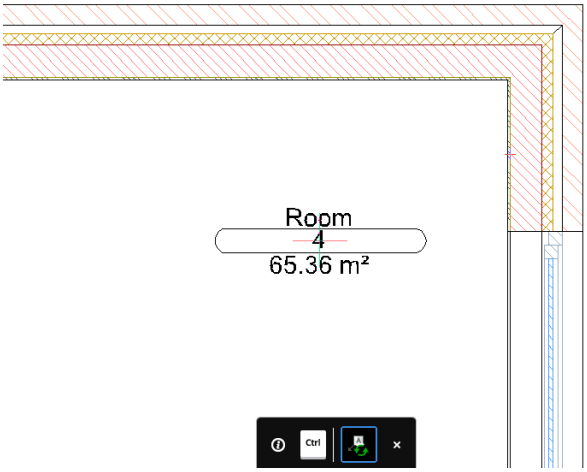
2 With no selection, select **Tag Assign BIM tags** manually from the Quad or enter BIMTAG followed by M

into the Command line to choose the manual option.



You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

3 Select a point on a line of a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible tags.



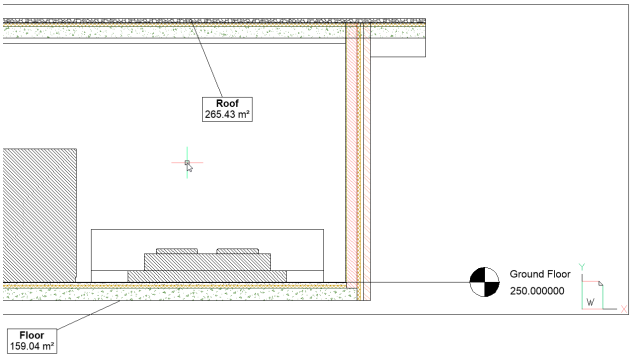
You are prompted: Place the tag [Next element]:

4 Move your cursor in order to adjust the position of the tag and left-click to place it.

5 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

6 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require more tags, press the ESC or ENTER key to complete the command. If you want all elements to be tagged, you can use the BimTag Select all entities command which we will explore next.

7 The same steps above can be used to tag a floor slab or a roof.



20.7.3.4 Assign BIM tags automatically

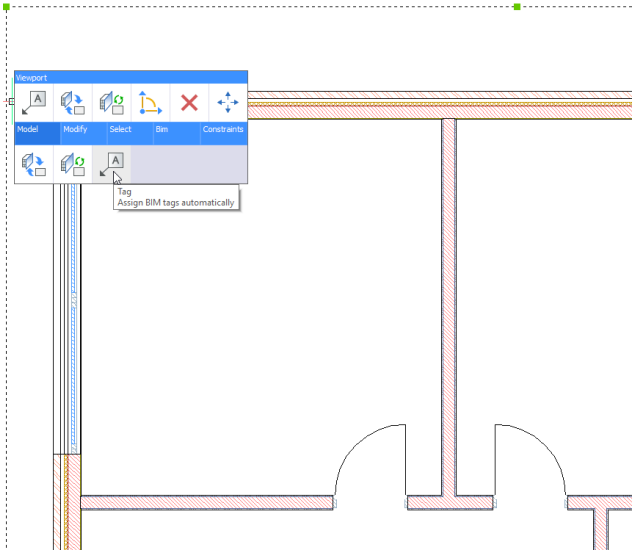
If you want all BIM elements in a section drawing to be tagged it is easiest to use the BIMTAG Select all entities command. This command automatically assigns BIM tags to all BIM entities inside a viewport. If the model is altered or new elements are added, the BIM tags can be updated and new tags added.

For more information about this command, visit the Command Reference article BIMTAG.

Automatically create BIM tags

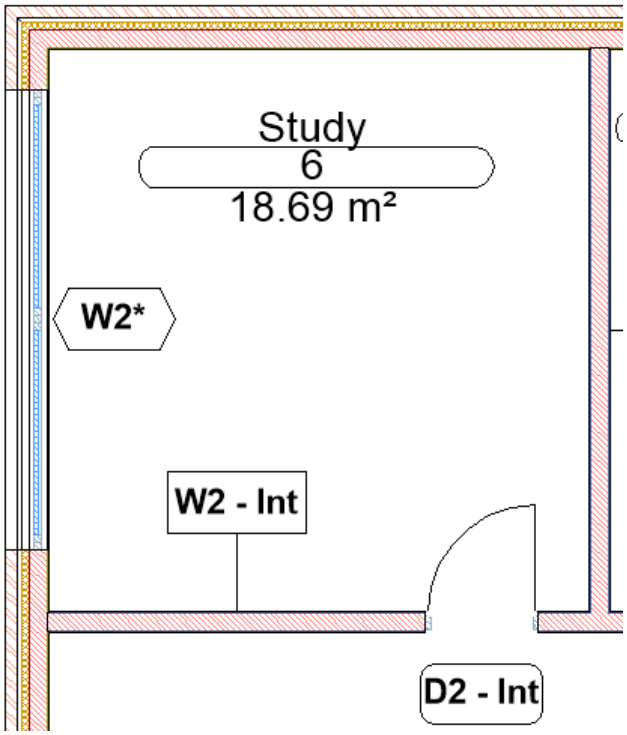
1 Open the section drawing on which you wish to create BIM tags.

2 Select a viewport containing the section which you will add BIM tags to.



3 Select **Tag Assign BIM tags automatically** from the Quad or enter BIMTAG into the Command line.

The BIM tags have now automatically been generated for each of the BIM entities in the viewport.



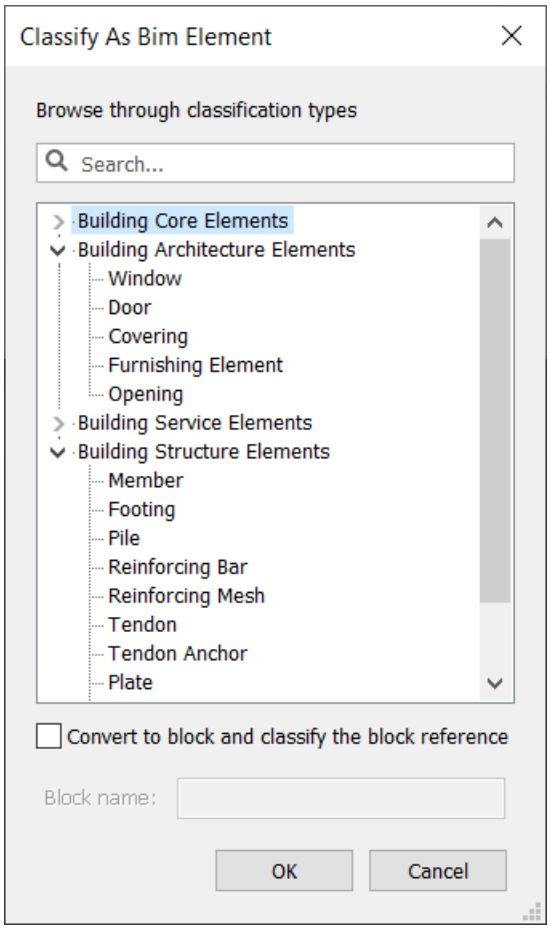
You are prompted: Select BIM section viewport [Manual]:

4 If you wish to automatically create BIM tags in another viewport on the sheet, select the viewport and these will be generated. If you do not require any further BIM tags, press Esc to complete the command.

20.7.3.5 Create the source data for a BIM entity type

By default the source data for the following BIM entity types already exist; **Annotation**, **Beam**, **Column**, **Curtain Wall**, **Door**, **Plies**, **Railing**, **Ramp**, **Roof**, **Room**, **Slab**, **Stair**, **Wall**, and **Window**.

Available BIM entity types are listed in the **Classify As Bim Element** dialog box. To open this dialog box, choose the **Other command** option of the BIMCLASSIFY command.



Create the tag block and multileader style

To create the source data for a Bim Member entity type:

1 Type SUPPORTFOLDER in the Command line and press Enter.

2 Once the folder has opened, navigate to the **BIM** > **Sections** folder and open the _SectionTag.dwg file.

3 Launch the EXPBLOCKS command.

4 The **Drawing Explorer / Blocks** dialog box will display.

5 Select the **_AnnotationTag** block (or any other existing block that you want to use as a template).

6 Right-click and choose **Copy** from the context menu.

7 Right-click and choose **Paste** from the context menu.

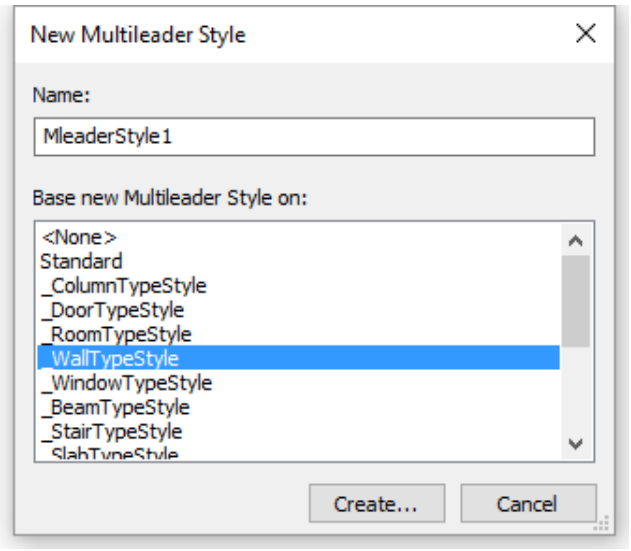
8 Click **Copy**, but keep both on the **Copy/Paste** dialog box.

9 Rename the copied block to **_MemberTag**.

10 Select **Multileader Styles** in the **Drawing Explorer** dialog box.

11 Click the New ( ) tool button.

The **New Multileader Style** dialog box will display:



12 In the Name field type _MemberTypeStyle.

13 Select a source style in the **Base new Multileader Style on** list.

14 Click the **Create...** button.

15 **(Optional)** Edit the **Leader** properties.

16 Click the **Content** tab.

17 Under **Block options**, click the **Source Block list** button and select **_MemberTag** from the list.

18 Close the **Drawing Explorer** dialog box.

19 Save _SectionTag.dwg.

Link the entity type to its multileader style

1 Type SUPPORTFOLDER in the Command line, then go to the **BIM** > **Section** folder and open _TagTypeToStyle.xml.

Use an ASCII text editor, e.g. Notepad.

2 Add the following right before </TagTypeToStyle>:

```
<tag type="BIM_MEMBER">
<style
name="_MemberTypeStyle"
autoPlacement="true"
autoRotation="true"
offset="1"
/>
</tag>
```

3 Save the file.

4 Note that this style has three options:

- **autoPlacement = "true"** will make sure tags are automatically generated when using the automatic BIMTAG option. If this option is set to false, then this tag type will not be generated automatically (you can still generate these tags manually).
- **autoRotation = "true"** will rotate the tag along the main axis of the tagged entity. If set to false,

the tag will be placed horizontally.

- **offset** will offset the tags perpendicular to the main axis of the tagged entity.

20.7.3.6 Customize tag blocks

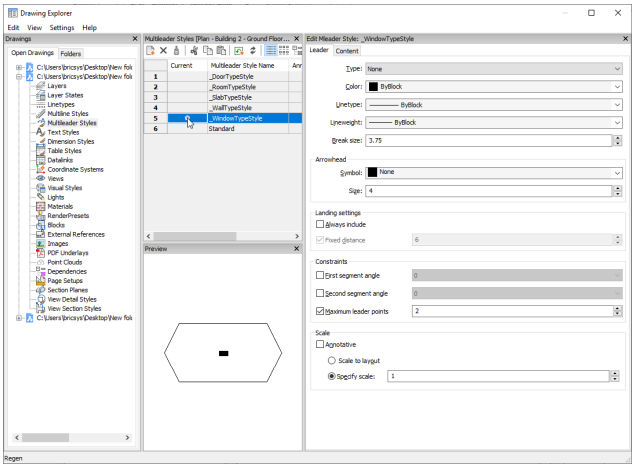
The content of a Bim Tag block consists of:

- Layout entities Linework, fixed texts, etc.
- Attributes Bim entity properties.

Nota: When you execute the BIMTAG command, BricsCAD checks whether the source data (tag blocks and their multileader styles) already exist in the Bim section drawing. If not, the source data is copied from the _SectionTag.dwg file. Otherwise, the tag block definitions and multileader styles in the section drawing are used. Therefore, editing tag blocks in the _SectionTag.dwg file only affects the creation of new Bim tags the first time BIMTAG is executed in a section drawing. If you edit tag blocks in a section drawing it affects Bim tags in that drawing only.

20.7.3.7 Customize tag blocks for the current drawing only

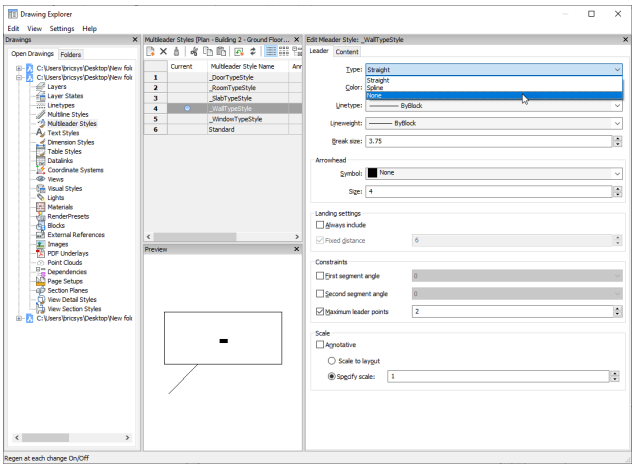
1 In the section drawing for which you wish to customize the BIM Tags, type MLEADERSTYLE in the Command line. The drawing explorer will open in the **Multileader Styles** tab.



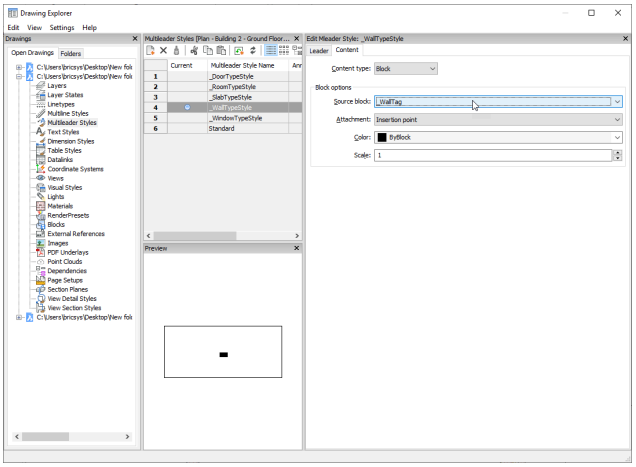
Each BIM tag is a multileader. The multileader style depends on the type of BIM element being tagged. All the multileader styles in the drawing are listed in the drawing explorer.

2 Select **_WallTypeStyle** from the list to see a preview of what the tag will look like. You can change the tag style from here.

3 Change the **Leader Type** from **Straight** to **None**. Now there will be no line connecting the tag to the wall element.



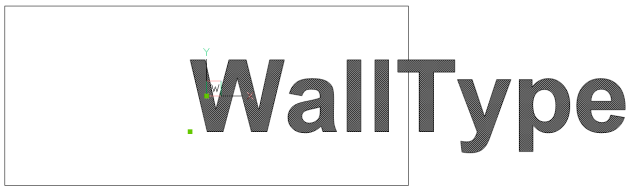
4 In the **Content** tab, the **Source block** is called **_WallTag**. Remember this and now close the drawing explorer.

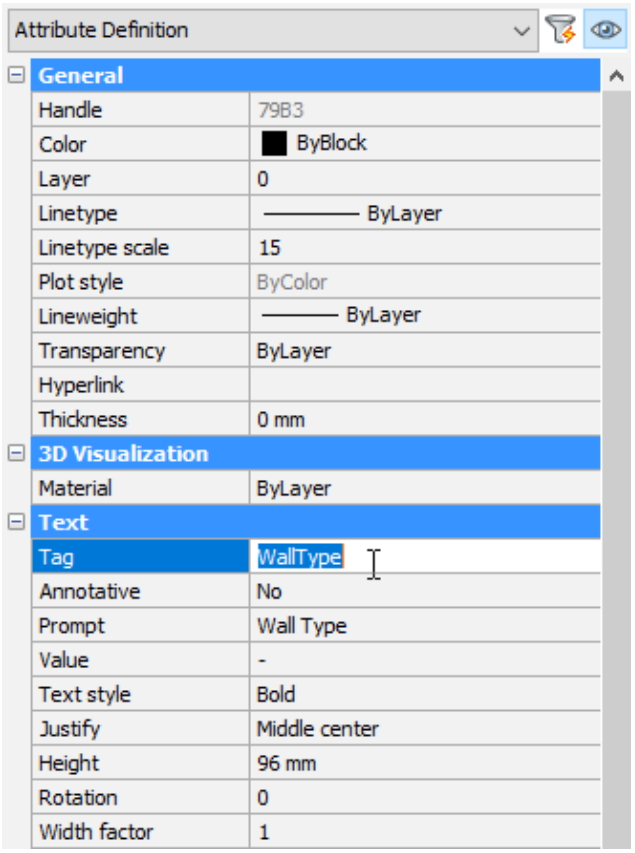


5 In the Command line type BEDIT and select **_WallTag** from the list. Click **OK**.

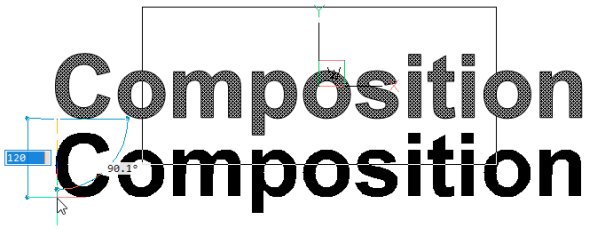
6 Select the **WallType** text and **Tag** value of the properties panel.

7 Replace **WallType** with **Composition** and press ENTER (see the next procedure for the syntax of other property types). Now the attribute text will display **Composition**.





8 Hover over this attribute and select **Copy** from the Quad. Select a base point to paste the copied attribute below the current one.



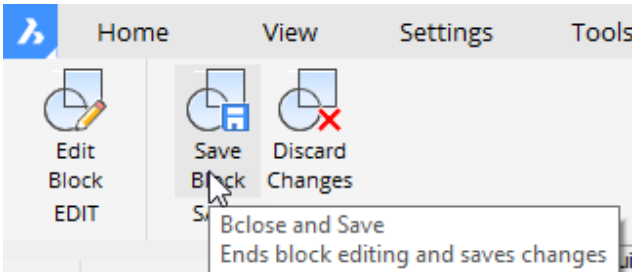
9 Repeat steps 5 - 7 to change the new attribute **Tag** value (**Quantity/Thickness**).

10 Delete the rectangle.

Composition Quantity/Thickness

11 Select the **Save Block** icon from the **Block Editor** tab of the ribbon.

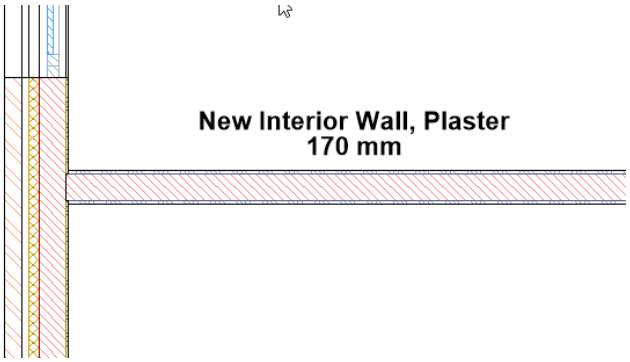
Your section viewport will now display.



12 Highlight the viewport and select **Tag Assign BIM tags automatically** from the Quad.

You are prompted: Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>:

13 Click hit the ENTER key to accept the default option of **Regenerate all**. The wall tags will be regenerated with the new style.



20.7.3.8 Edit the source data to customize BIM tags for all section drawings

1 With any drawing open, type SUPPORTFOLDER in the Command line followed. Hit Enter.

2 Once open navigate to the **BIM** > **Sections** and open the _SectionTag.dwg file.

3 Launch the BEDIT command.

4 Select the tag block you want to edit, then click the **OK** button.

5 Add attributes using the following syntax, in the **Tag** field, in the properties panel:

Property type	Syntax	Examples
BIM	<property name>	name room bounding
Parameters	parameters/<parametername>	parameters/w parameters/framethickness
General	general/<property name>	general/layer general/linetype scale
Mass	mass/<property name> mass/<property name>:<child>	mass/surface area mass/ucs elevation:minimum
Geometry	geometry/<property name> geometry/<property name>:<child>	geometry/position geometry/ucs elevation:maximum
Quantity	quantity/<property name>	quantity/length quantity/net footprint area
IFC	<ifc property set>/<ifc property>	wall common/load bearing door window glazing type/ glass layers
User Defined	<property set label>/<property label>	exterior walls/price
Classification system	<classification system label>/ <classification code label>	omniclass/omniclass classification code

Nota: To know which properties are available for a specific BIM entity type, select such entity in the model, then open the Properties panel.

In the image below the properties of a BIM **Wall** entity are listed.

Click the + button to expand a property type node.

3D Solid

General	
3D Visualization	
Material	ByLayer
Mass	
BIM	
Type	Wall
Name	E1
Description	Ext. Wall type 1
Building	Building
Story	Floor 1
Composition	Cavity Wall, Brick
GUID	1MvYF3FSfA5gFoL$dkVfY5
Wall type	Cavity
Room bounding	On
Centerline	Off
Number	W12
Quantity	
Length	810 cm
Thickness	320 mm
Height	375 cm
Gross volume	9.34 m³
Net volume	9.34 m³
Gross side area	29.18 m²
Net side area	29.18 m²
Gross footprint area	2.49 m²
Net footprint area	2.49 m²
Plies	4
Ply	1
Wall Common	
Reference	
Acoustic rating	45dB
Fire rating	2h
Combustible	Off
Surface spread of flame	
Thermal transmittance	0 W/m²·K
Is external	On
Extend to structure	Off
Load bearing	On
Compartmentation	On
OmniClass	
OmniClass classification code	22-04 27 23 Cavity Wall Unit Masonry

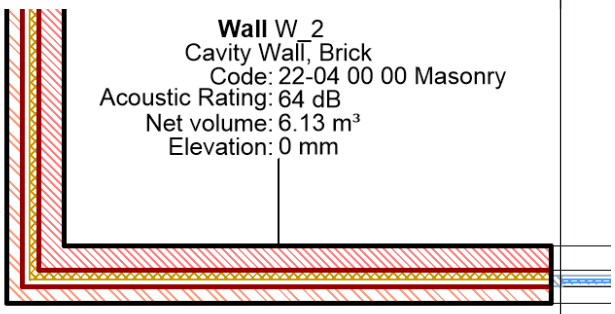
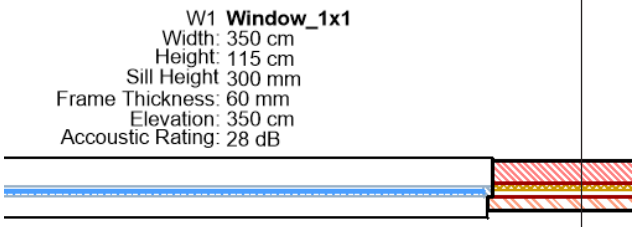
6 For each new attribute do the following:

- Copy an existing attribute or create a new attribute (ATTDEF command).
- Select the new attribute.

  Nota: Do not double click the attribute. The **Edit Attribute Definition** dialog box does not accept spaces in the **Tag** field.
- In the Properties panel, type the property reference in the **Tag** field (see the syntax in the above table).
- **(Optional)** Define the Text properties (**Text Style**, **Justify**, **Height**, …) in the Properties panel.

- **(Optional)** Add linework and fixed text.

7 On the **Block Edit** toolbar: click the **Save** icon to conclude the BEDIT command.

Definitions in the tag block	Resulting tag
type number composition Code: Omniclass/omniclass classification c Acoustic Rating: wall common/acoustic rating Net volume: Quantity/net volume Elevation: mass/ucs elevation:minimum	
_WallTag block	Wall tag in a plan section
number **name** Width: parameters/W Height: parameters/H Sill Height sillheight Frame Thickness: parameters/framethickness Elevation: geometry/ucs elevation:minim Accoustic Rating: window common/acoustic rati	
WindowTag block	Window tag in a plan section

20.7.3.9 Assign composition tags

Composition tags allow you to tag a wall or a slab element in order to specify its composition. These BIM tags are a list of the ply materials which make up the element.

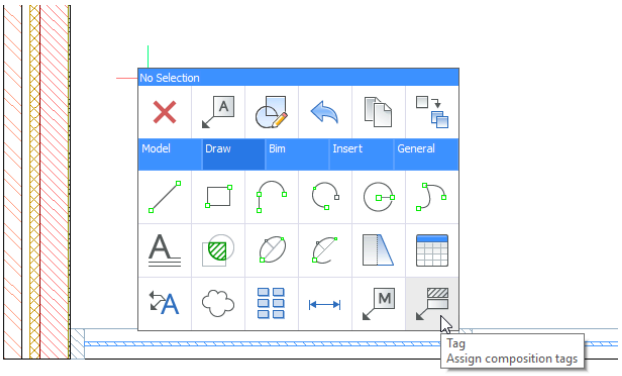
For more information about this command, visit the Command Reference article BIMTAG.



Create composition tags

1 Open the section drawing for which you wish to create composition tags.

2 Select **Tag Assign composition tags** from the Quad.

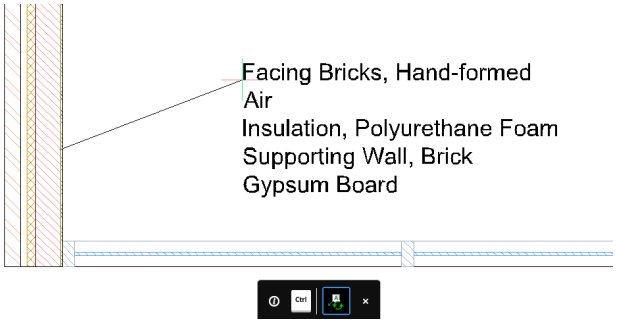


Or type BIMTAG in the Command line, then do one of the following:

- Type M+Enter for the manual option; then T + Enter for **Change Tag** type, followed by O+Enter for the composition option.
- Click the options in the Command line.
- Click the options in the **Prompt** Menu.

3 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

4 Select a point on a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible composition tags.



You are prompted: Place the tag [Next element]:

5 To adjust the position of the tag, move your cursor and left click to place it.

6 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

7 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require any more tags, press the Esc or Enter to complete the command.

20.7.4 Create drawing types with drawing customization

20.7.4.1 About drawing customizing

BricsCAD BIM uses drawing templates to customize your generated drawings. For a general understanding of how these templates are created and work, please visit the Templates for generated drawings article.

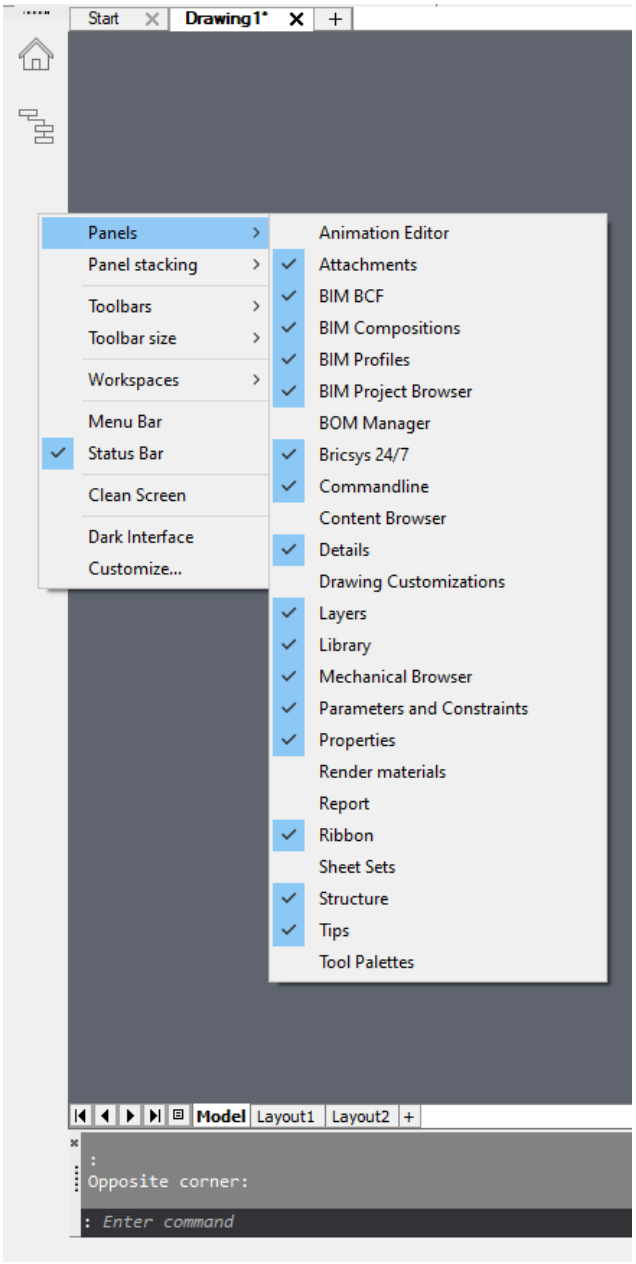
In V21, **Drawing Customization** allows you to create bespoke templates for drawing types based on a set filter parameter rule against an entity type's value or property within the 3D BIM model. This is especially helpful for customizing graphics of model entities independent of their assigned layers or layer states, therefore transferable and usable on other projects with similar entity properties.

The **Drawing Customizations** tool is operated by 3 principles – **Entity Customizations, Filter Rules & Styles** – and is only accessible from the panel.

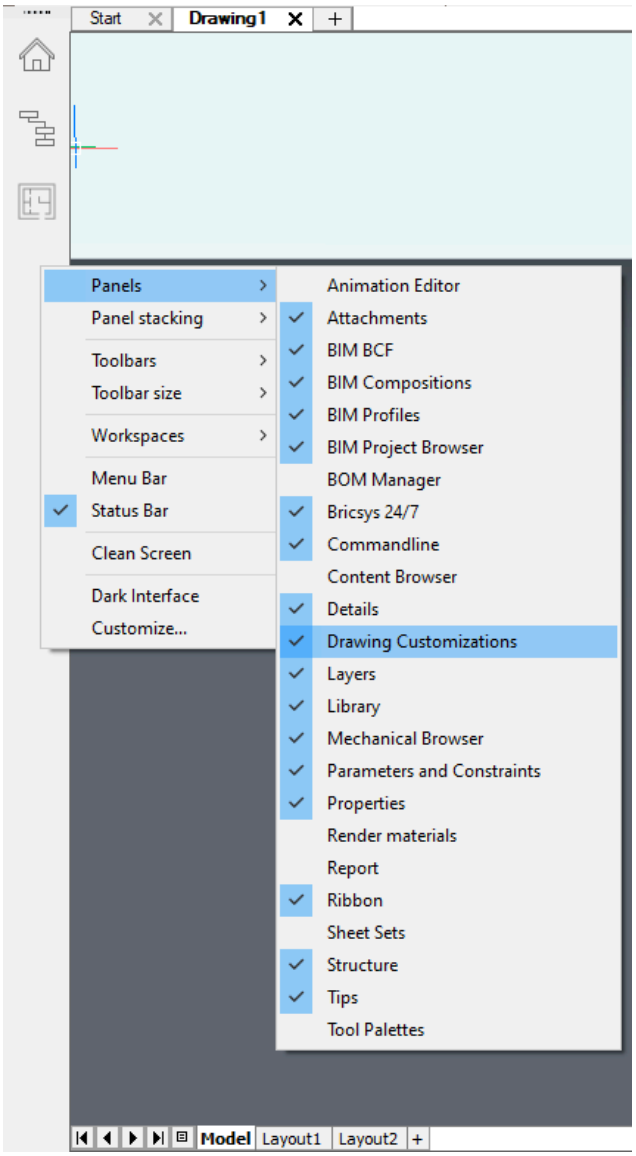
> ***Nota:*** Version Control for **Drawing Customization** will be implemented from V21.1.06. As features and parameters are constantly improved and added in future V21 updates, using older versions of BricsCAD to open **Drawing Customization** templates may affect their behavior and performance. Therefore, a versioning system would help to prevent customizations from being accidentally opened or modified by BricsCAD versions that are too old to handle them properly.

20.7.4.2 Drawing customizations panel

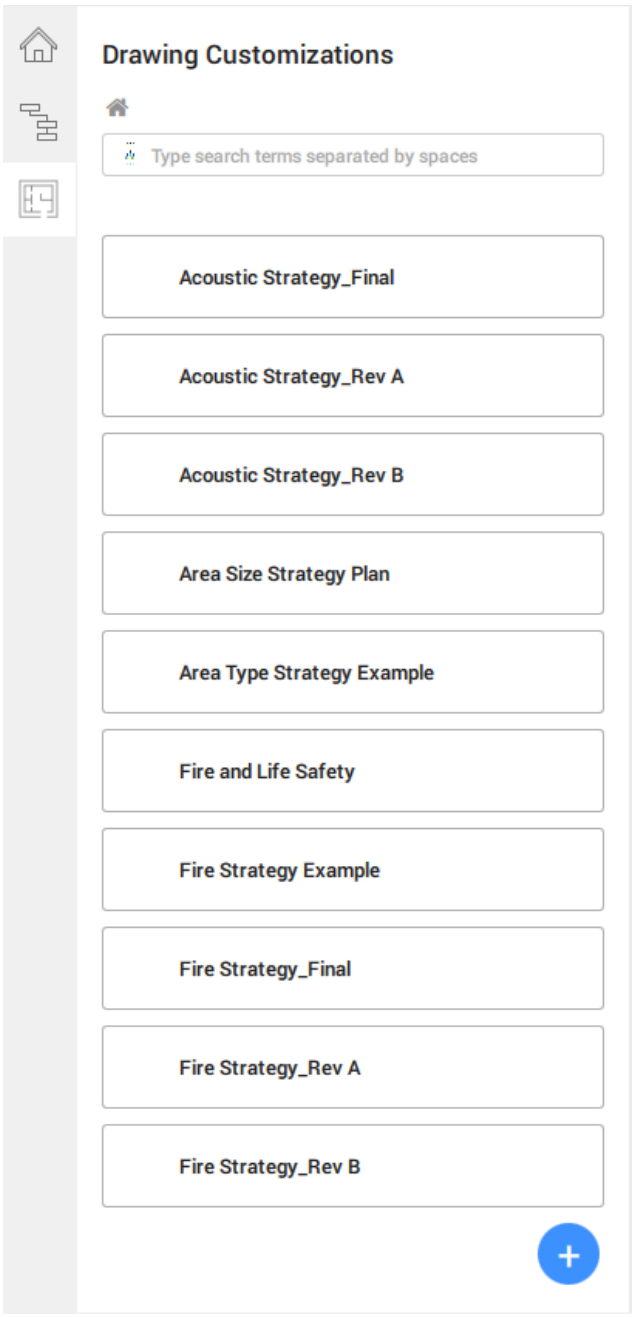
1 Right-click the panel and hover to **Panels**.



2 Select **Drawing Customization** to dock the tool icon in the panel.



3 Open the panel, it allows you to create view templates to control and customize the visibility settings of your drawings on layouts.



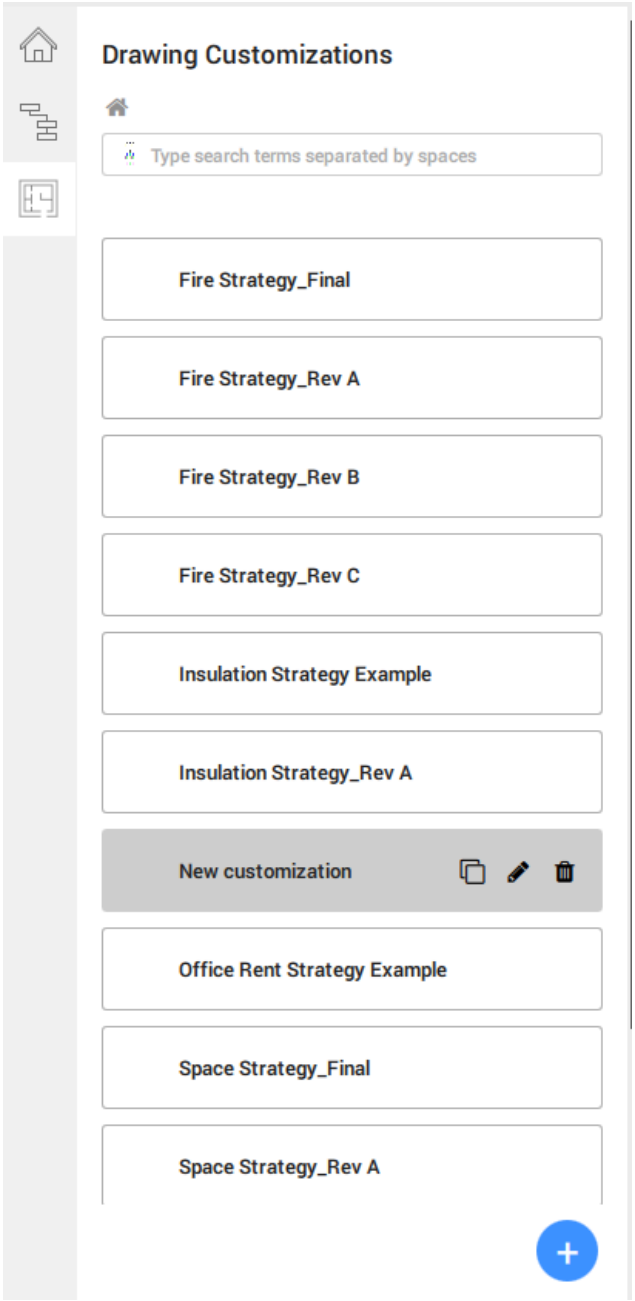
4 Click to create a new template. By default, the template is named **New Customization** with the text highlighted.



Nota: All the controls of **Drawing Customizations** are represented as graphic icons within the **User Interface** panel, such as to create a new control item. They are repeated throughout the tool in

different settings, for example **Entity Customizations**, **Ply Customizations**..., but their functions and purposes remain consistent.

5 Rename the template whilst the text is being highlighted. Once done, press Enter to save and apply the name.

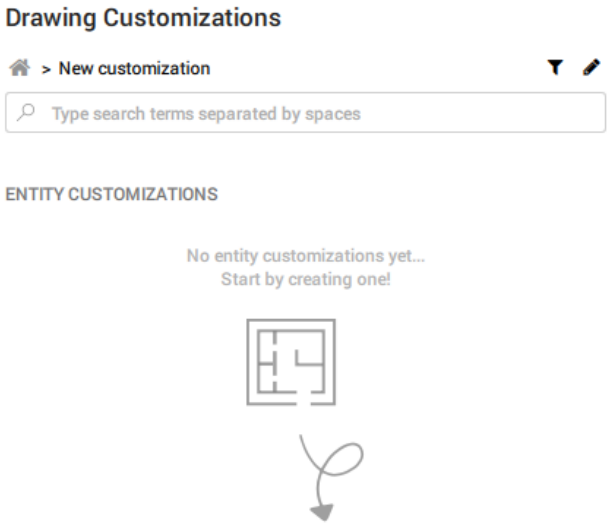


Nota: You can duplicate, rename or delete your customization templates.

6 Click the newly created **Drawing Customization template** tab to begin defining its customization settings.

The first step into the template brings you to **Entity Customizations**, where you will create the

necessary entity customizations template and will be first described later in this article.

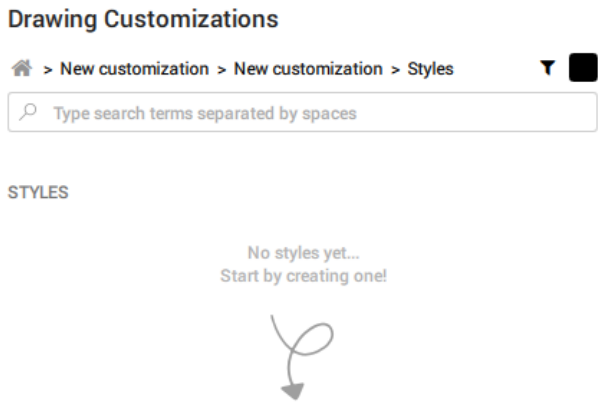


You will also notice a funnel icon and a pencil icon on the top right part of the panel. They are the **Filter Rules** and **Styles** tabs respectively.
The 3 principles – **Entity Customizations, Filter Rules & Styles** – which make up the **Drawing Customization** tool as described earlier in the article, are now accessible on this main page.

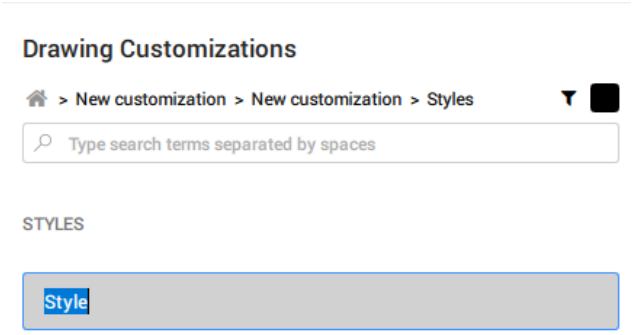
20.7.4.3 Drawing customizations styles

1 Click the pencil icon to access the **Styles** tab.

This is where your desired style options, such as line weight, line colors..., are stored and will appear as drop-down options in the **Entity Customizations** and **Center Customizations** settings as shown in the previous section above.



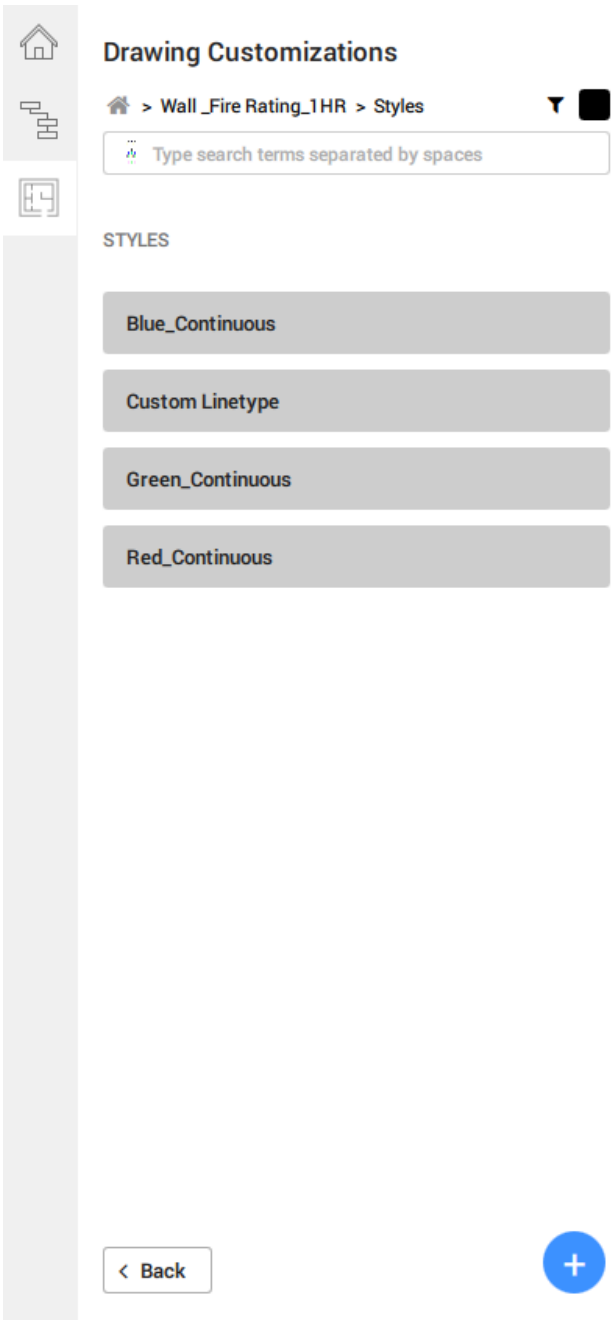
2 To create a new style, click + at the bottom right of the panel and a new **Style** template tab will appear.



3 By default, the tab is named **Style**. Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.

4 Create a list of desired styles that you wish to use in this **Drawing Customization** template. Each tab contains its individual styles settings, such as the style's color, line type, line weight and line type scale.

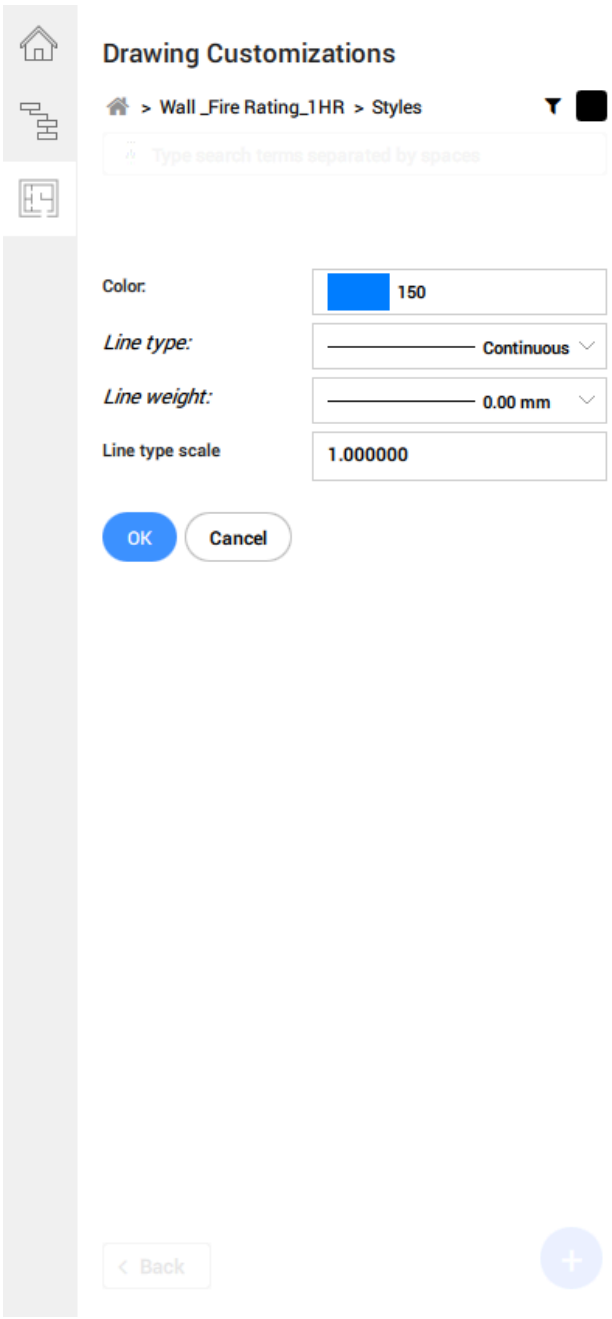
An example of the style pages a list of templates:



5 To edit any of the styles template available, click on its relevant tab to access its settings.

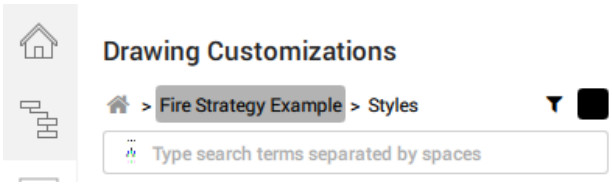
Nota: Similar to the **Drawing Customization template** tabs, hover over the styles template tabs to reveal the function icons to duplicate, rename or delete.

Style settings include the options to define **Color**, **Line Type**, **Line Weight** and **Line type scale** in their respective drop-down dialogs. Once done, click **OK** to save and apply the changes.



6 To leave the **Styles** page, click **Back** at the bottom left side of the panel to return to your previous page. From there, you can navigate to modify the **Entity Customizations** or **Rules** templates.

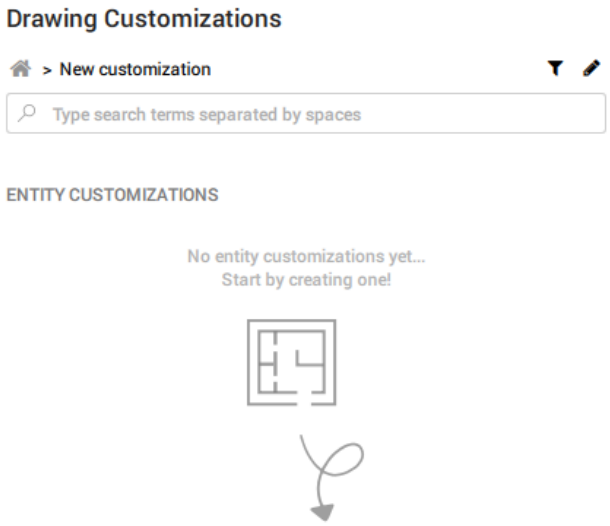
Or click on your desired branch in the tree structure map above the search bar to navigate to your desired page.



Nota: The tree structure map is helpful to keep yourself orientated within the **Drawing Customization** tool. Similarly, the search bar allows you to narrow to a specific template if you have a long list of values to navigate through.

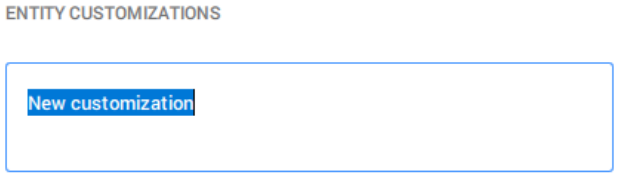
20.7.4.4 Entity customizations

Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

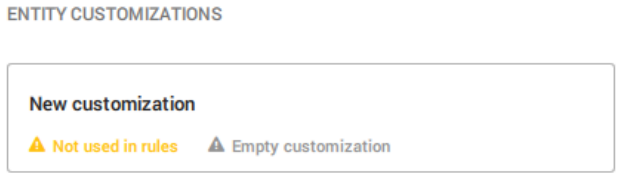


1 Click ⊕.

2 To create a new **Entity customization** template. By default, the template is named **New Customization** with the text highlighted.



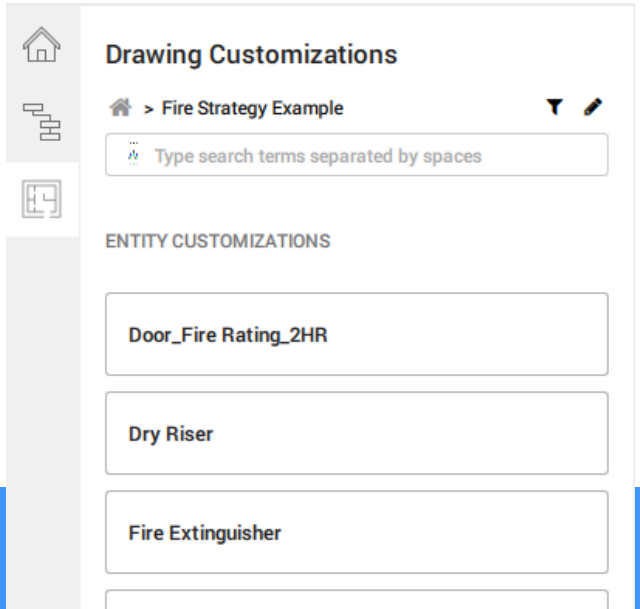
3 Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.



As the **Entity customization** was newly created and not fully set up, you will find the relevant notices at the bottom of the **Entity customization** tab. This is a helpful mechanism to remind you if any customization settings have not been defined yet, saving you the need to check the settings manually.

4 Create a list of templates for the entities you wish to customize.

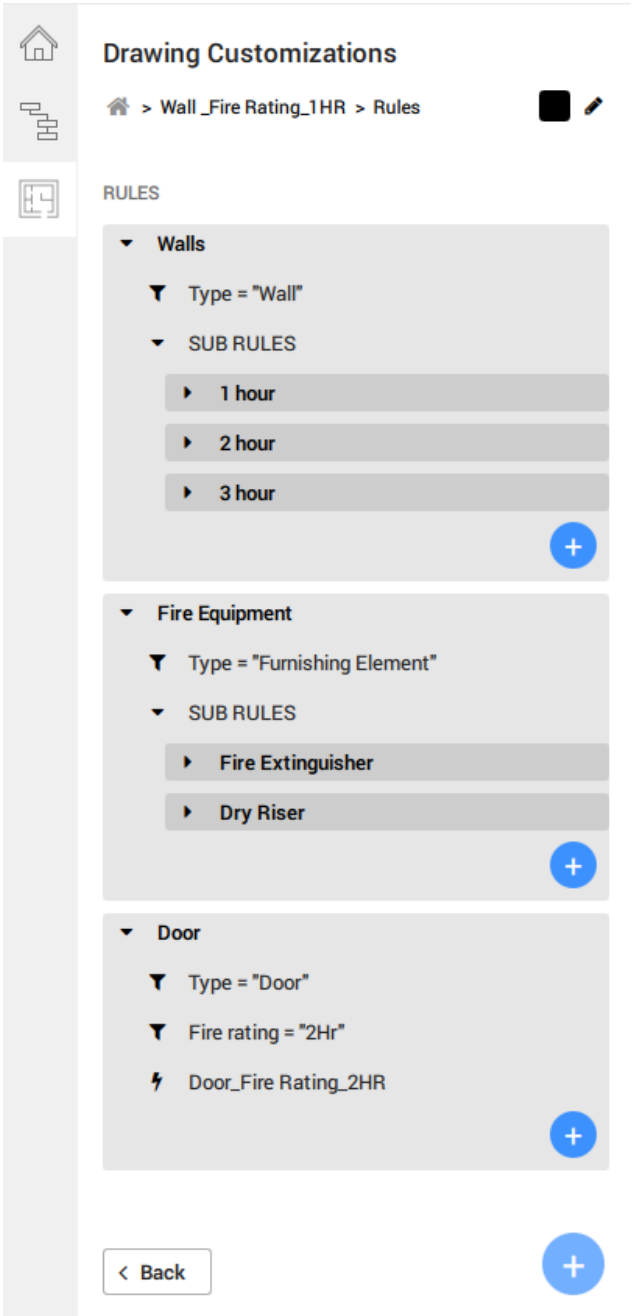
5 A list of **Entity Customizations** templates could look like this:



20.7.4.5 Filter rules

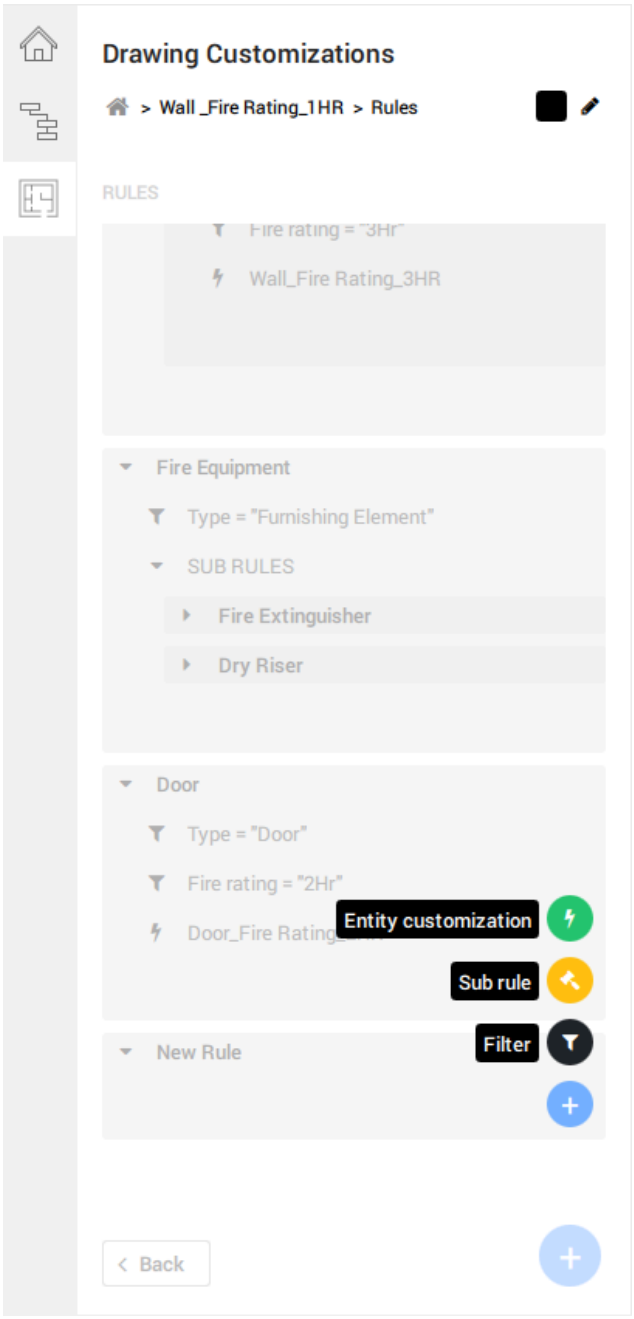
To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the **Filter Rules** tab located as a funnel icon on the top right-hand part of the panel next to the **Styles** icon.

Nota: The **Nested Property Search** function has been introduced from V21.1.06 onwards, allowing you to specify how nested objects within an assembly are filtered by and displayed.

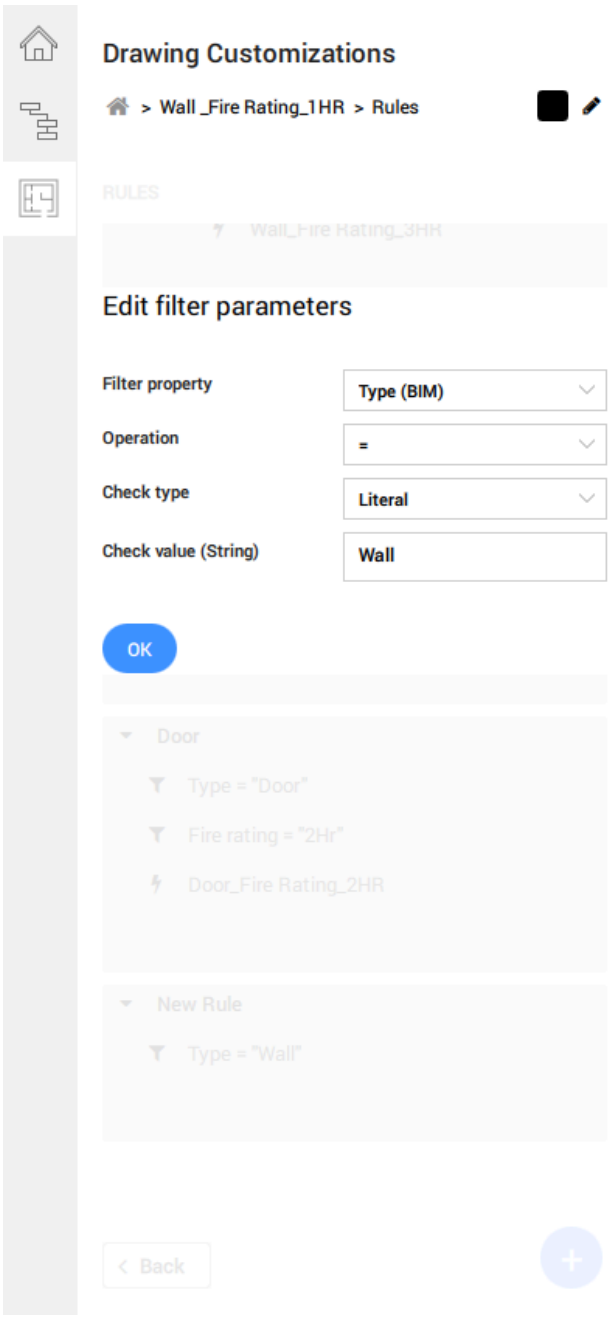


1 **Filter rules** can be further classed into **sub rules**, which falls as a sub-set of the parent rule, e.g. **Fire Equipment** as a **Filter Rule**, whereas **Fire Extinguisher** is classed as a sub rule.

- also appears respectively and accordingly at each rule level. At the end of each rule, regardless of it being a subset or otherwise, it has to end with an **Entity Customization**.

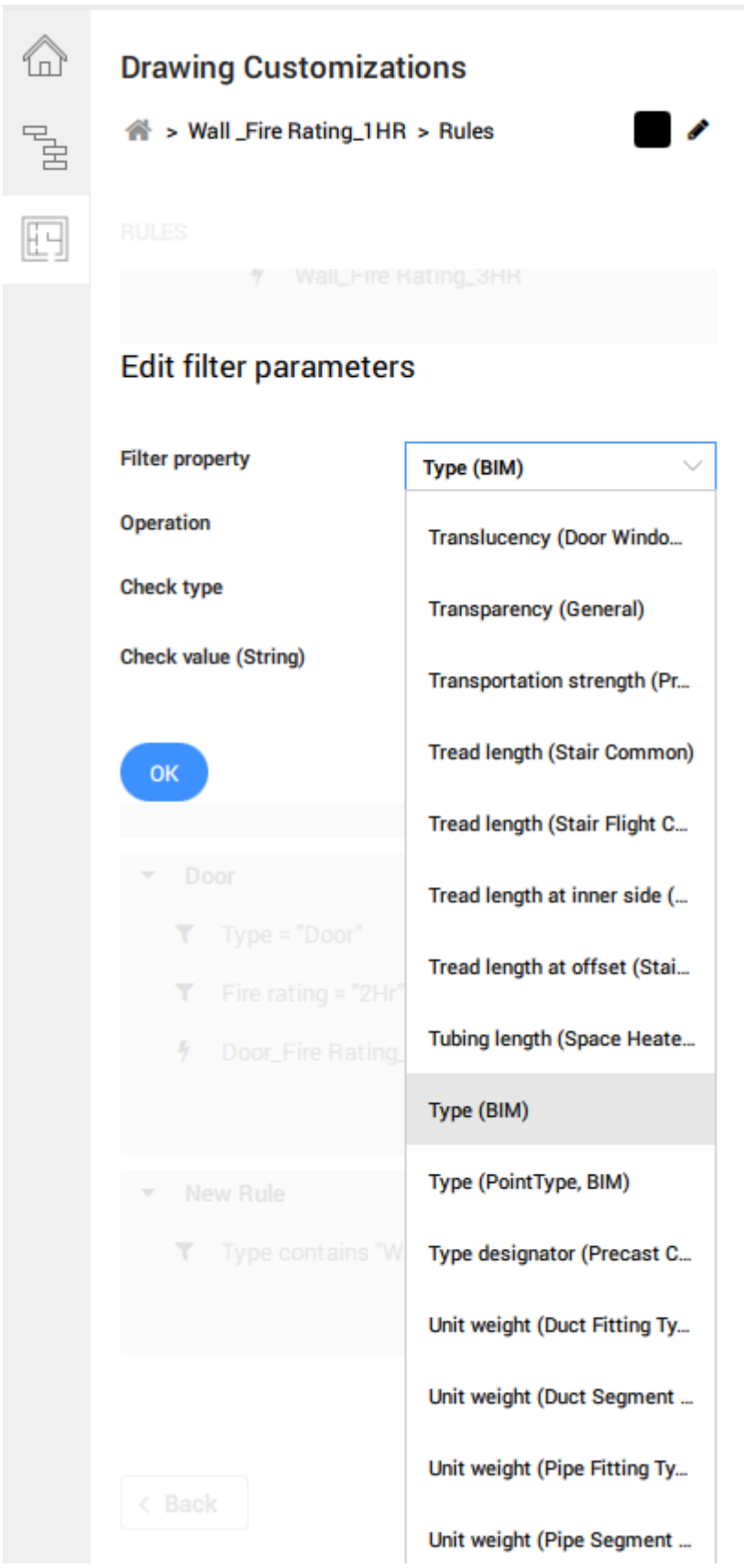


2 Adding and editing a filter parameter will lead you to a prompt, where the options of filter property are determined by the BIMPROPERTIES imported in the **namespace** settings.



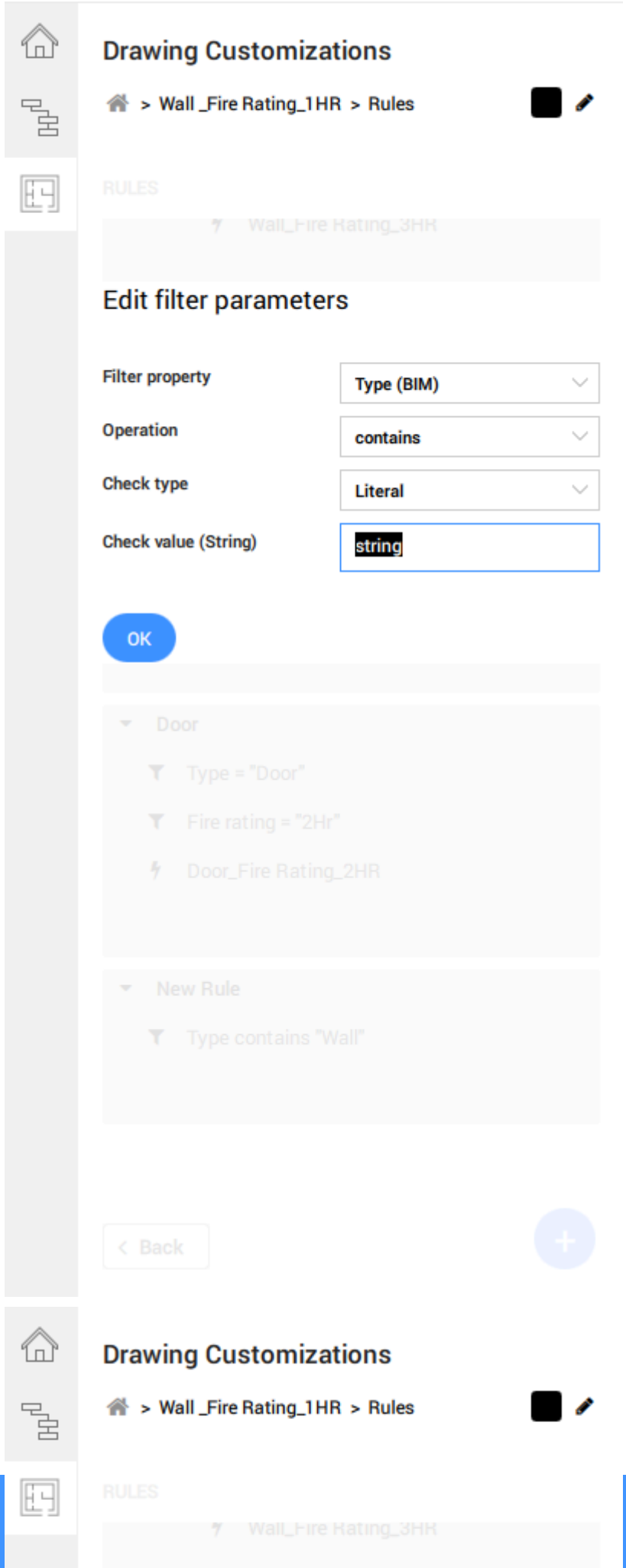
3 The filter **Operation** values are also available as a drop-down list.

Nota: You can also start typing a certain keyword to search a specific property that you want to filter on.

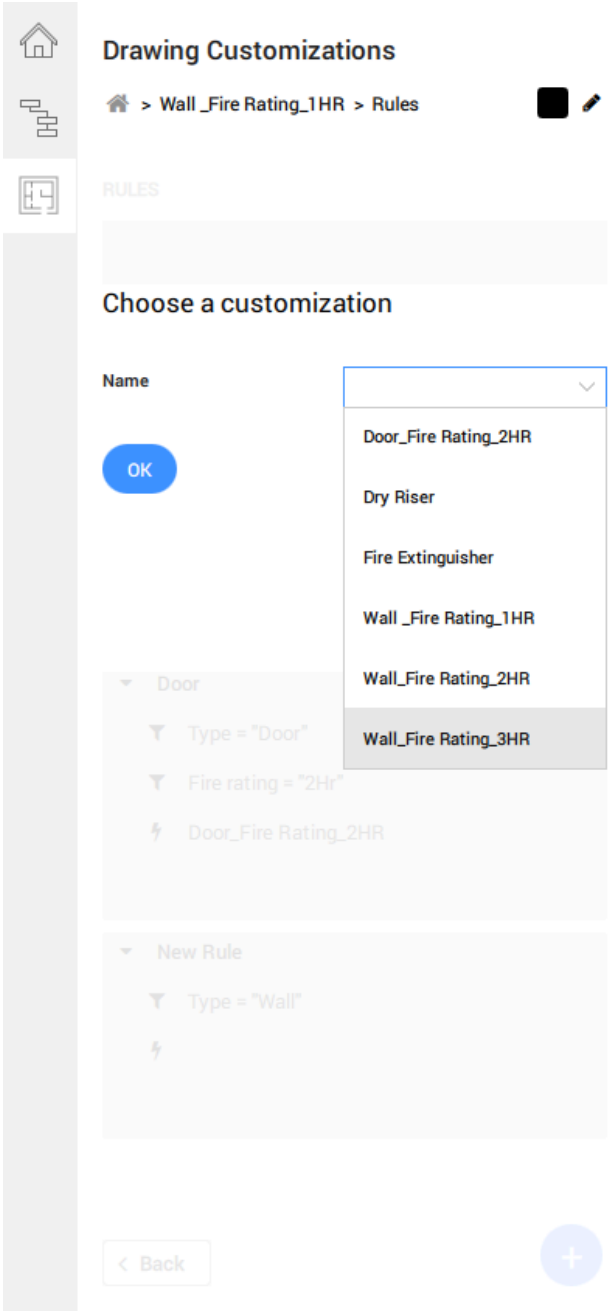


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES

4 You can specify a string manually in your check value if your **Check** Type is a **Literal**. If you have selected a **Property** value for **Check Type**, your **Check Property** value will refer to the same values found in the **Filter Property** drop-down.


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR
Edit filter parameters
Filter property
Type (BIM)
Operation
contains
Check type
Literal
Check value (String)
string
OK
Door
Type = "Door"
Fire rating = "2Hr"
Door_Fire Rating_2HR
New Rule
Type contains "Wall"
Back
Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR

5 To end and legitimize a **filter rule**, the **Entity Customization** value must be added and filled in. The options available in the drop-down selection reflects the existing customizations tab available in the **Entity Customization**page.

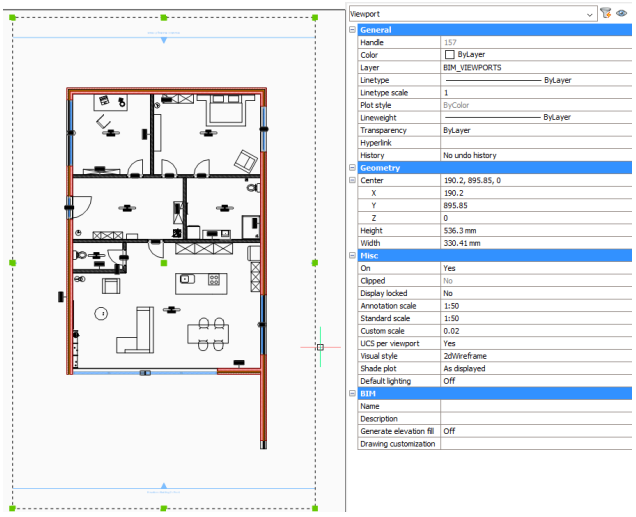


20.7.4.6 Applying templates to viewports on sheets

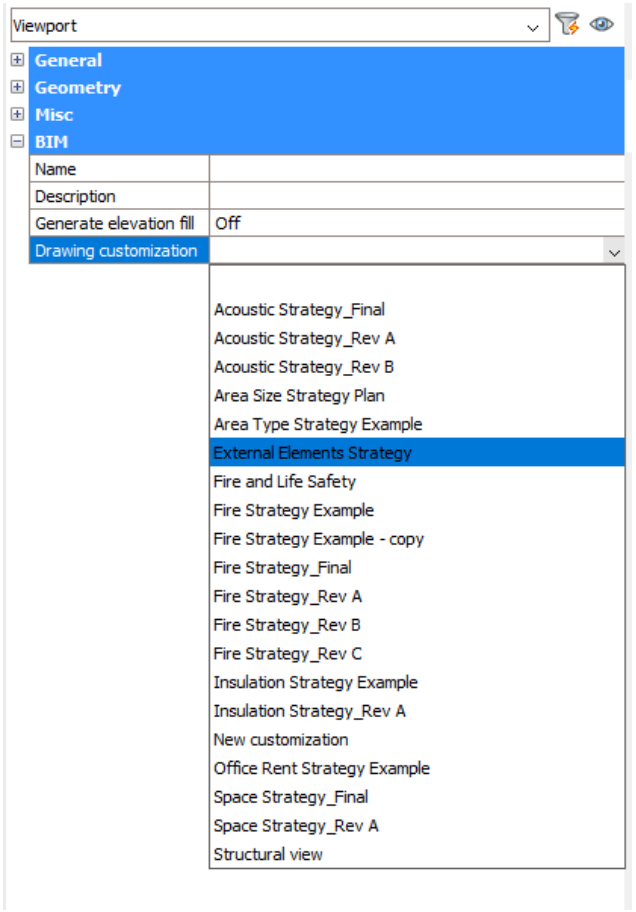
With your **Entity Customizations**, **Filter Rules** and **Styles** all set, your **Drawing Customization** template is now ready.

1 To apply the template, navigate to your sheet drawing with your desired viewport.

2 Open the Properties panel and select the viewport to view its properties.

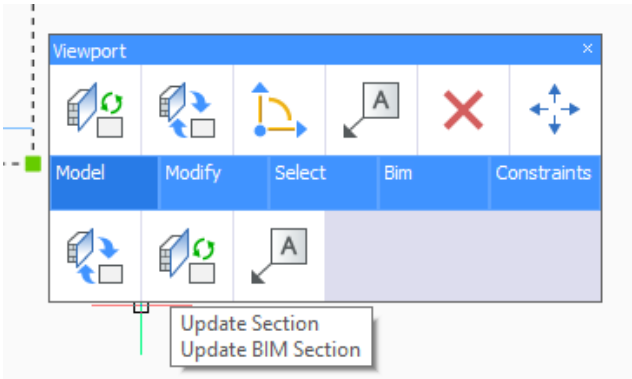


3 Navigate to the **Drawing Customization** property and click on the drop-down selection to the right.



4 Select your desired **Drawing Customization** template.

5 Once done, hover over your selected viewport to access the Quad. Click **Update Section** in the Quad under **Model** tab.



6 You should see a refreshed **Viewport** with the graphical changes as per your **Drawing Customization** settings.

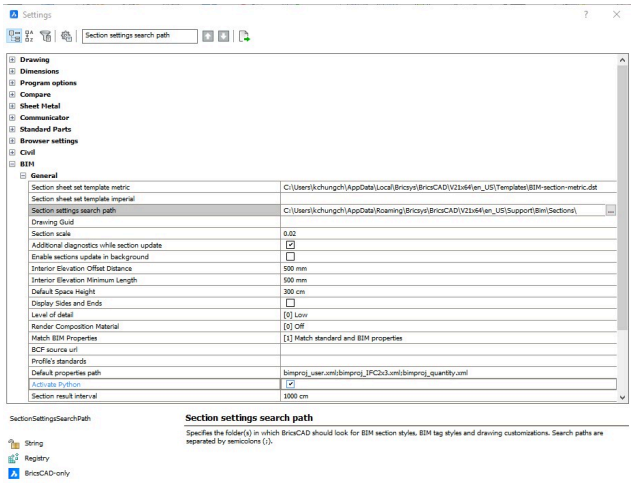
20.7.4.7 Accessing customization templates

By default settings, these templates are stored in the **Customizations** folder within your **Support** folder in the following path, under the **Bim** folder and then **Sections** subfolder:

C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support\Bim\Sections\Customizations

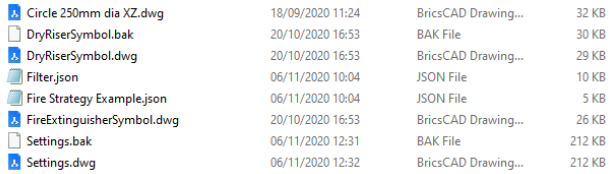
The **AppData** folder is hidden by default, so you may have to unhide it first. This path can also be accessed by entering SUPPORTFOLDER in the Command line.

You can specify a different path if your Customization files are stored elsewhere in an accessible folder. Simply redefine the path in the **Section settings search path** in the **Settings** dialog box or typing SRCHPATH in the command-line:



New customizations created with the panel will be stored in a **Customizations** subfolder in the specified path. Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the **Create New** buttons.

Settings.dwg contains the values created in the **Style** tab in the form of layers and if you have created any custom properties that you wish to use as a **Filter** parameter, you have to ensure they are also reflected in the Settings.dwg.



Nota: As these **Customization** folders are stored locally in the machine you have created it with, you need to ensure the folders and their contents within are copied and pasted into the same location on other machines, should you wish to share it with other users on other machines. This is the current, temporary procedure to do so, as a dedicated function to streamline the sharing process within the **Drawing Customization** tool will be released shortly in an update.

20.7.5 Define detailed section

20.7.5.1 Command

BIMSECTION

20.7.5.2 Define a detailed section from scratch

1 Select **Detail Section** from the ribbon or use the Command line by entering BIMSECTION followed by **D** to choose the detail option.

You are prompted: Select the first corner of the section box or [Based on an existing section/Scale]:

2 Select the first corner of the area you wish to be contained within the detailed section.

3 **You are prompted**: Select the opposite corner of section box:

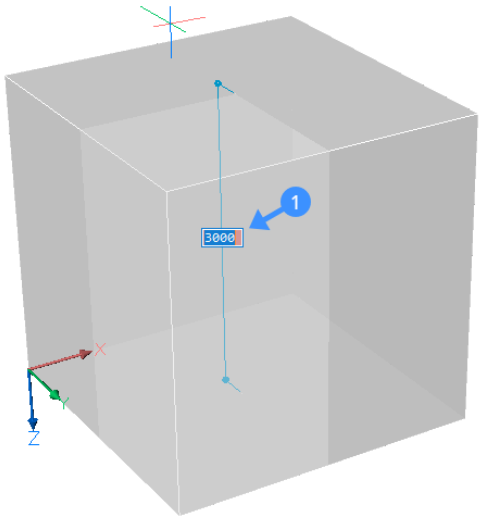
4 Select the opposite corner snap to a point or enter the distances. Use the TAB key to switch between dynamic dimension fields.

5 **You are prompted**: Select the height of volume box <4000>:

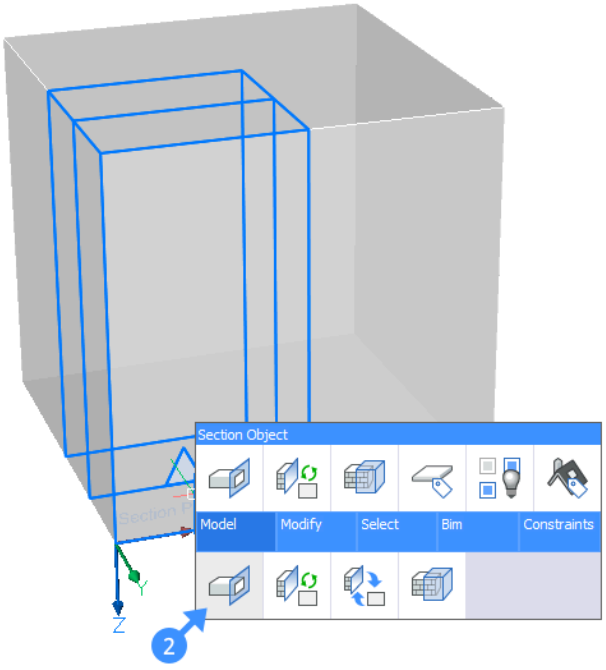
In this case, the default height is 4000mm.

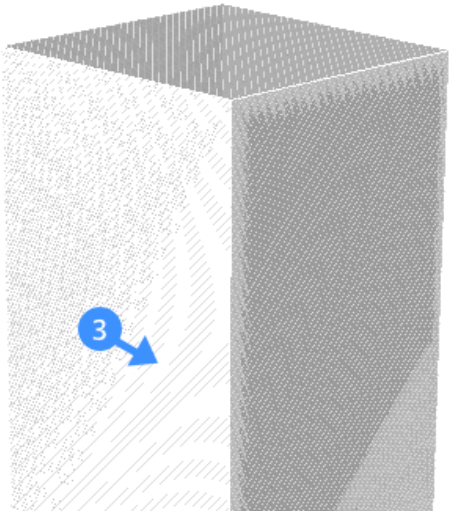
6 To complete the volume, do one of the following:

- Press Enter to accept the default height.
- Specify a point.
- Type a value in the dynamic dimension field (1).



To view the detailed section which has just been defined, hover over the section and select **Clip display** (2) from the Quad **or** double-click the selected section. Now only the detailed section will be visible (3).

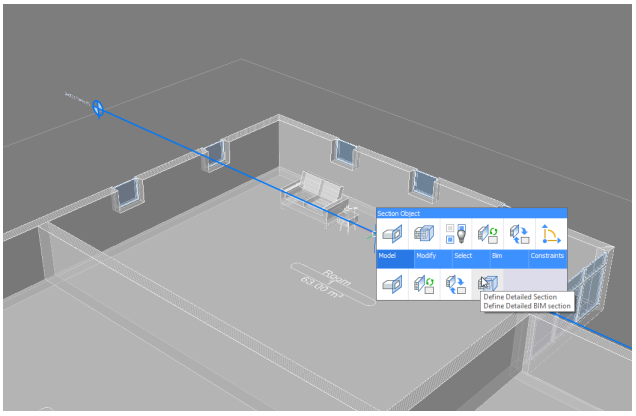




20.7.5.3 Define a detailed section based on an existing section

Here you will be shown how to position the base of your detailed section on an already existing section.

1 Highlight the existing section that you want the detailed section base to be positioned on and select **Detail Section** from the Quad.

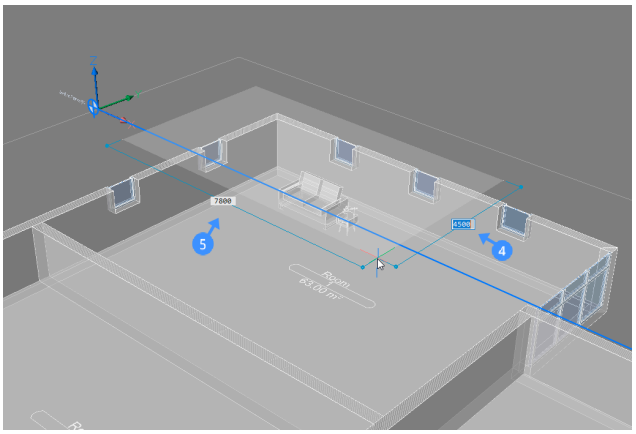


You are prompted: Select first corner of volume box base rectangle:

2 Draw a rectangle around the desired area by typing the values in the dynamic dimension fields.

3 Hit the TAB key to go back and forth between the dynamic dimension fields (4, 5). Your selection will be aligned with the plane of the existing section.

4 Press Enter to confirm the values. The **volume section** box displays dynamically.



You are prompted: Select the height of the volume section box.

5 Press Enter to accept the default height.

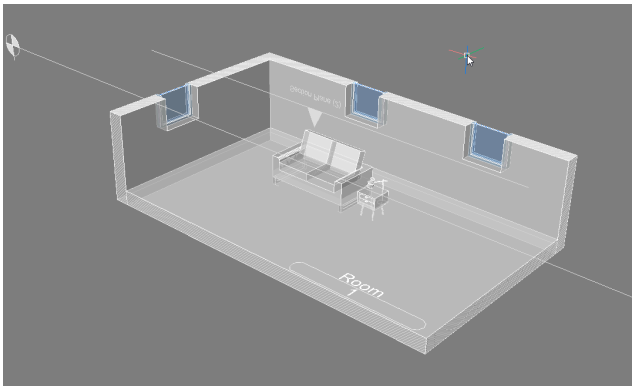
6 Specify a point.

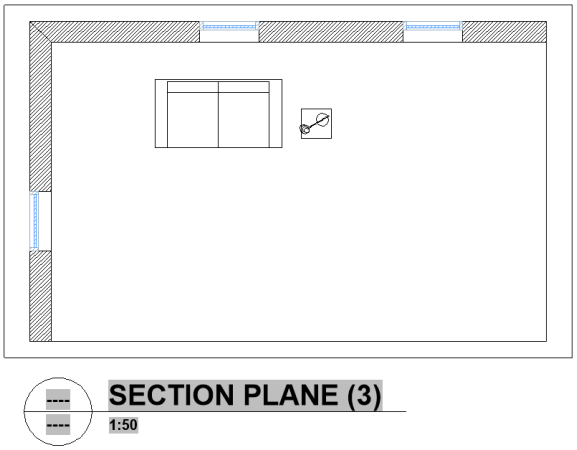
7 Type a value in the dynamic dimension field.

8 The detailed section has now been defined and can be found in the structure browser. Hover over the section and select **Clip display** from the Quad **or** double-click the selected section to view it.

9 To generate the detailed section on a sheet, select the detailed section and select **Generate Section** from the Quad.

10 You can open the section results by clicking **Open Model** when the section plane(s) are selected. The sheet is saved in the same folder containing the model drawing.





If you need more help with how to generate drawings, visit the **Generating Drawings** page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

20.7.6 Define reflected ceiling plan

20.7.6.1 Command

BIMSECTION

20.7.6.2 About

Reflected ceiling plans show a mirror image of the ceiling. These are useful to show the layout of lights and other installations in the ceiling. To produce the reflected ceiling plan a section is created. This section displays the part of the building above the section boundary.



For more information about this command, visit the Command Reference article BIMSECTION.

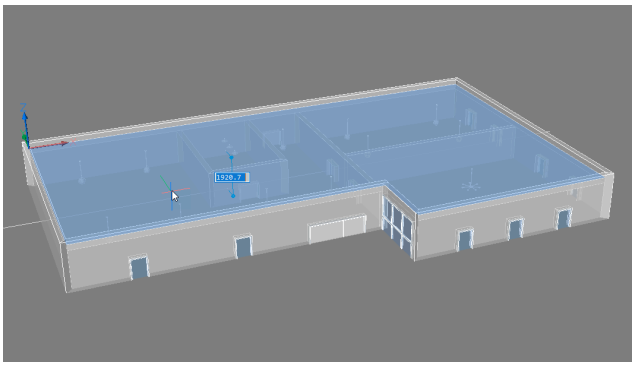
20.7.6.3 Create a reflected ceiling plan

1 Select **Reflected Ceiling Plan** from the ribbon **or** enter BIMSECTION into the Command line followed by

R to choose the **Reflected ceiling** option.

You are prompted: Select a point to place section:

2 Select the ceiling or a face with a plane parallel to that of the ceiling.

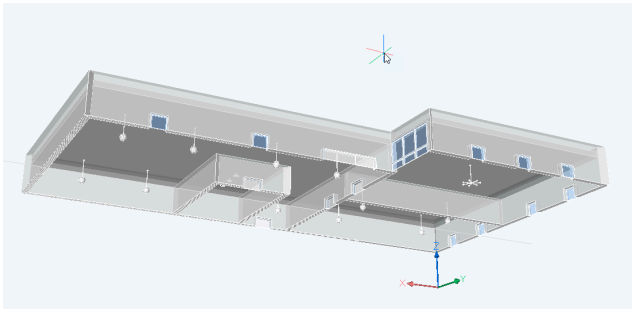


You are prompted: Specify distance:

3 Do one of the following:

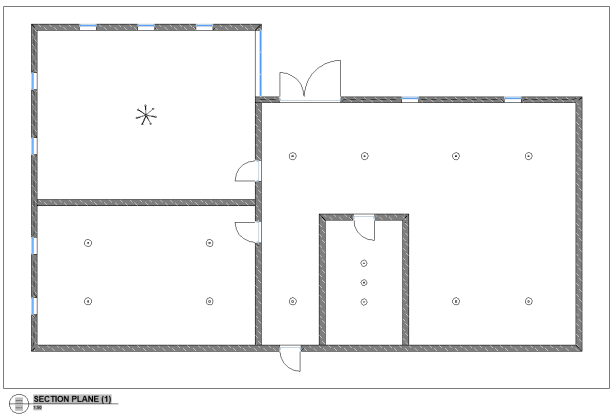
- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want.

4 The reflected ceiling section has now been defined. To view hover over the section and select **Clip display** from the Quad.



5 To generate the reflected ceiling plan on a sheet, hover over the section in the model and select **Generate Section** from the Quad.

6 To open the section result drawing click **Open Model**. The sheet is saved to the same folder that contains the model drawing.



If you need more help with how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

20.7.7 Detailing with BimPatch

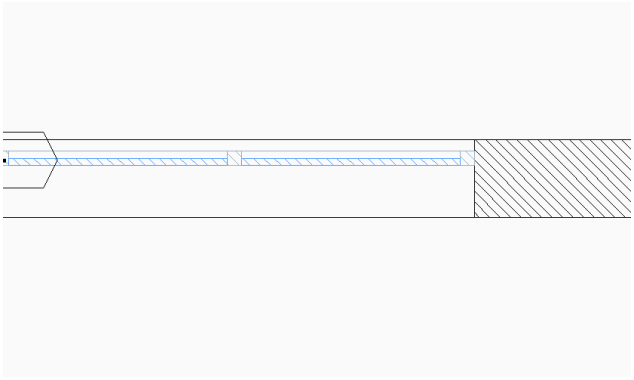
Nota: BIMPATCH cuts out part of geometry and replaces it by a block reference. In this cutting process, entity data linked to the 3D model are lost. Therefore, starting from V21.2, this command is no longer available.

20.7.7.1 Command

BIMPATCH

20.7.7.2 About

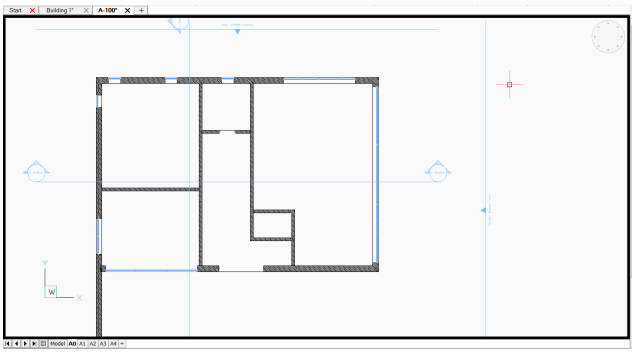
In this procedure, we will add fixtures and a vapor barrier to our window-wall connection. We will detail this window-wall connection.



20.7.7.3 Procedure: detailing a window connection

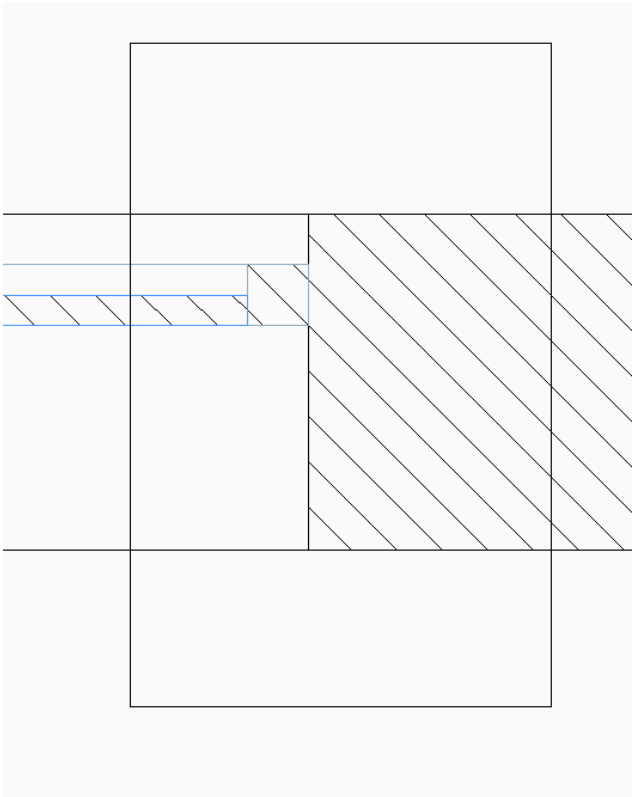
1 Open the section.

2 Go to the model space by double-clicking in the viewport **or**toggle the viewport mode in the **Status** Bar.

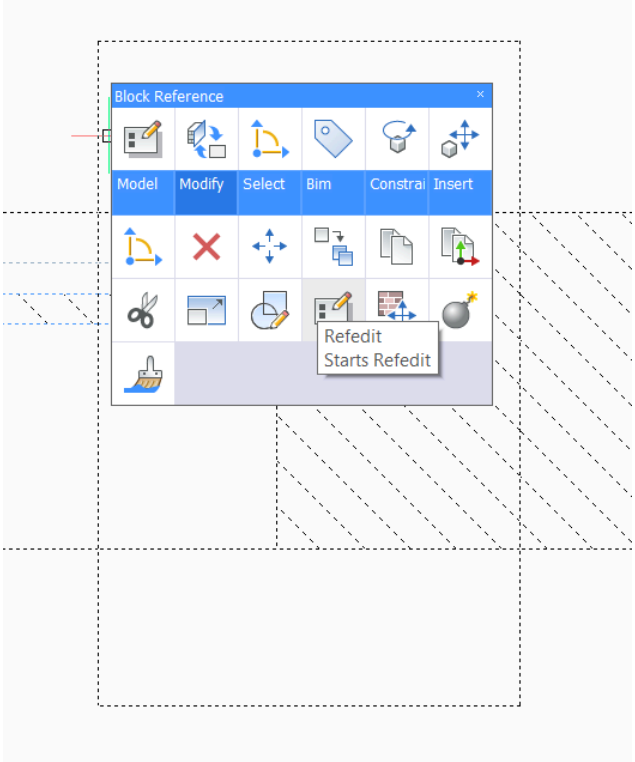


3 Launch the BIMPATCH command in the model space.

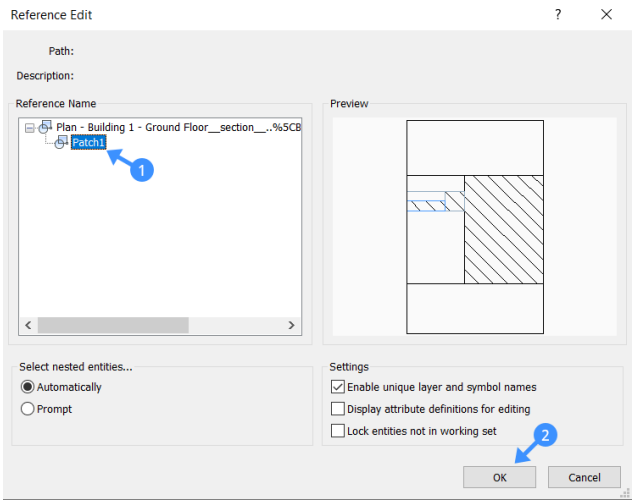
4 Draw the rectangle over the detail and hit enter.



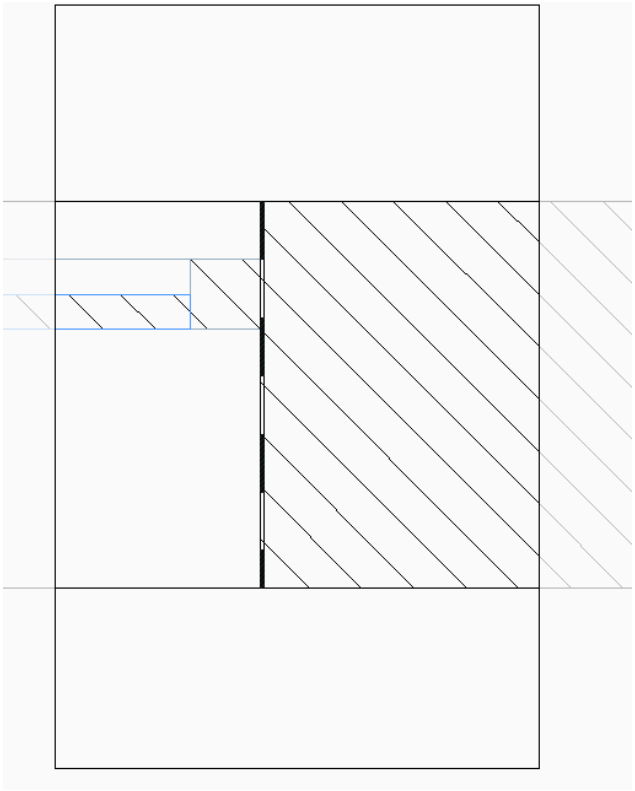
5 Hover over the section object and launch **Refedit** from the quad.



6 A menu appears, select the patch (1) and click **OK** (2).

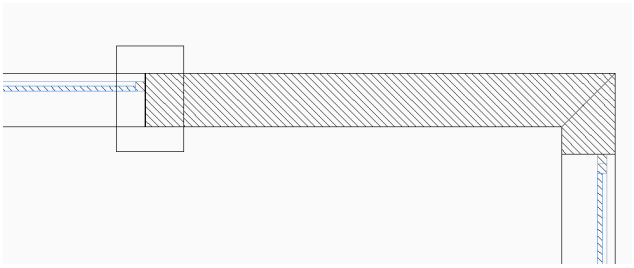


7 Draw the required detail for the window-wall joint. We will add a vapor barrier.

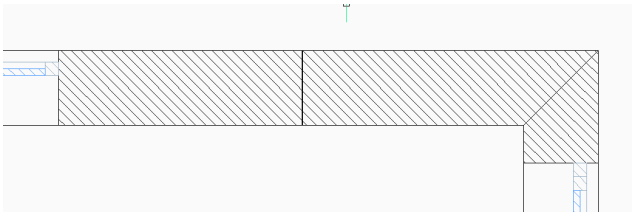


8 Save the changes made in the **RefEdit** session with the **Save** option of the REFCLOSE command.

9 Go back to the paper space. The detailing of the window connection is done. If we now move the window to the left in the 3D model, you will see that the detail you made a patch of moves along.



Nota: If you would have added the vapor barrier in the model space of the section without making a patch of it, the vapor barrier wouldn't be seen as a part of the detail. If you would move your window, the vapor barrier would stay in the same position as you drew it before.



20.7.8 Editing data extraction definition (.dxd) files

DXD (Data Extraction Definition) files are plain text files. You can edit DXD files created by the DATAEXTRACTION command using a text editor such as MS Notepad and use them in the Creating Schedules procedure in the **Project Browser**.

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**

20.7.8.1 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 blocks into 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.

20.7.8.2 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 **MInsert**(s).
- **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.

20.7.8.3 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: **Entity Property**, **Attribute**, **Drawing**, **Formula** or **Block View**.

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 to **EntityPropertyName**.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

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.

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.

Formula | **EntityPropertyName** | **DisplayEntityPropertyName=FormulaExpression** | **ValueOutputFormat**

- **Formula** is a predefined ECI word.
- **EntityPropertyName** is a 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 contain constants such as PI or functions such as SIN (see Using Expressions). 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.

BlockView | **BlockViewPropertyName** | **DisplayBlockViewPropertyName** | **BlockViewOptions**

- **BlockView** is a predefined ECI word
- **BlockViewPropertyName** is a unique name for this block view item
- **DisplayEntityPropertyName** is the output name of the property; cannot be omitted
- **BlockViewOptions** is a string in the following format:

  ViewType;DwgPath;SectionName

  - **ViewType** is one of the following: **Front**, **Top**, **Left** or **Right**; cannot be ommited.
  - **DwgPath** is path to dwg file, may be relative with respect to the dxd file; can be ommited.
  - **SectionName** is the name of the section to extract the section settings from for the block view construction,;can be ommited.

If **DwgPath** or **SectionName** are ommited or not found, default settings will be applied.

An example:

BlockView|BlockViewProperty1|Elevation symbol|Front;../Sections/_SectionSettings.dwg;_BlockViewSettingsSection

*

If present all entity properties are taken into account; all other lines in this section, except Drawing and Formula lines, are ignored.

20.7.8.4 The [Filter] section

Syntax description:

Relational operators:

Operator	Meaning
==	Equal to
!=	Not equal to
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to

- Relational expressions are enclosed in '()' brackets.
- Logical operators AND and OR are case-insensitive and enclosed 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.
- Use the **EntityPropertyName** to refer to properties (not the **DisplayEntityPropertyName**).
- Lines can be commented with ';'.
- String values are enclosed in double quotation marks "" .

<OR (Type == Window)(Type == Door)>

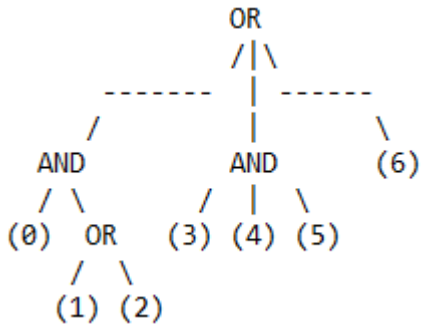
The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BIMCLASSIFY command to learn more about BIM entity types.

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).



20.7.9 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 **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.

20.7.9.1 Inserting the title block

1 Open the **Drawing Explorer - Blocks** dialog.

2 Double click the **Title** block.

The **Drawing Explorer**dialog closes and the block is attached to the cursor.

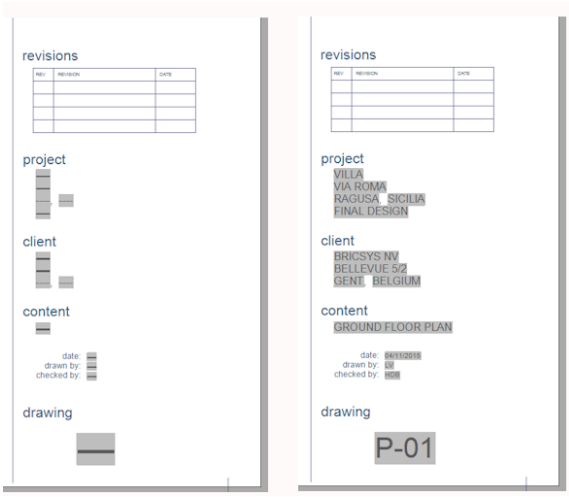
In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm)

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 Open the **Sheet Sets** panel (SHEETSET command) and define the **Project** and **Custom Properties** available in the context menu.

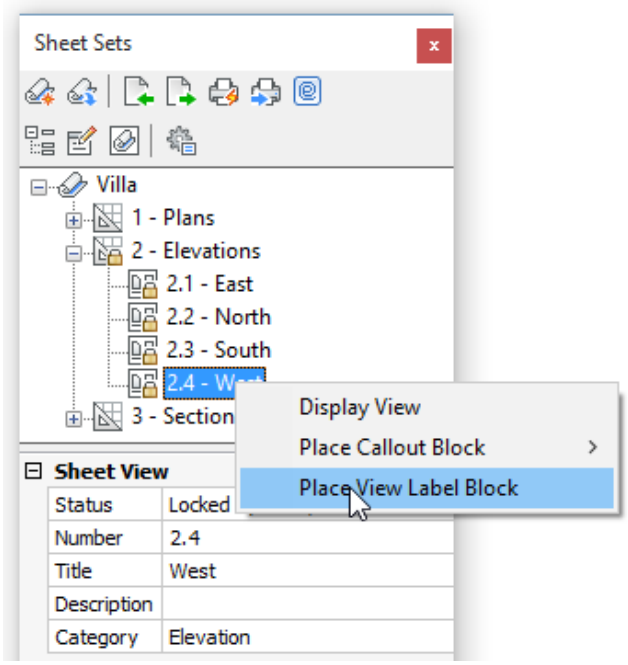
5 Select the **Title** block, then click the **Update Fields...** tool button ( ) in the **Redraw/Regen** toolbar.



20.7.9.2 Inserting a view label for a sheet view

1 Select the sheet view in the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the view label block.



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.

5 **You are prompted**: Scale factor for block <1.0>:

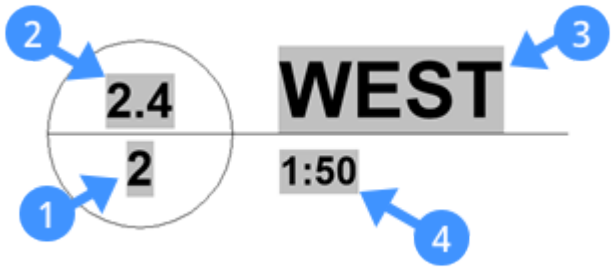
6 Specify the scale factor or right-click to accept the current scale.

7 **You are prompted**: Rotation angle for block <0>:

8 Specify the rotation angle or right-click to accept the current angle.

9 The View Label block is inserted.

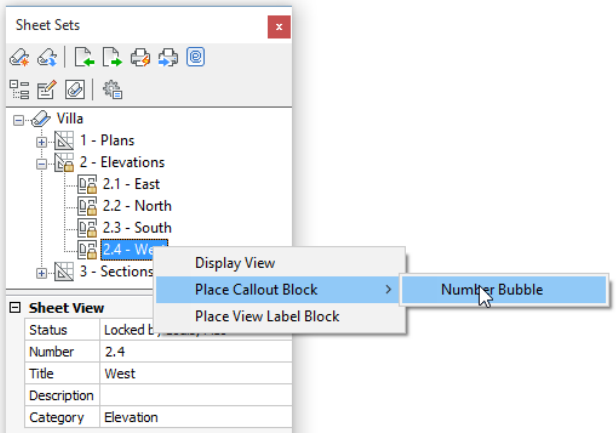
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.



20.7.9.3 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 for which you want to insert the **Number Bubble** block.



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.

7 **You are prompted**: Scale factor for block <1.0>:

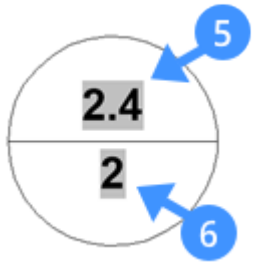
8 Specify the scale factor or right-click to accept the current scale.

9 **You are prompted**: Rotation angle for block <0>:

10 Specify the rotation angle or right-click to accept the current angle.

11 The **Number Bubble** block is inserted.

The Sheet Number (5) and Sheet View Number (6) of the selected sheet view are filled in automatically.



20.7.9.4 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.

20.7.10 Elevation hatch

20.7.10.1 About elevation hatch

Generate elevation fill is a property of a section that can display the texture or solid hatch of a composition when a 2D elevation is generated. Hatches can also be generated for other types of sections when activated in the properties panel.

The hatch is generated for every visible boundary on the 2D elevation including boundaries that are not perpendicular to the section. Hatches are generated without perspective corrections. The union of elevation lines can be changed independently from the 'union section' settings in the **Physical Materials** dialog.

20.7.10.2 Make elevations

Nota: **Elevation** here refers to a *Section Plane* in a 3D model that has its **Type** property set to **Elevation**. This can then be used to generate 2D drawings that represent the elevations of your model.

1 Create BIM elevations by using **Bimify**.

2 Select the BIM elevation and choose **Generate BIM Section** in the **Model** command tab in the Quad.

3 Select the BIM elevation and choose **Open BIM Section Model** in the **Model** command tab in the Quad.

The section result displays in the layout viewport of the destination drawing.

4 Select the paper space viewport.

5 The viewport properties display in the Properties panel.

6 Under BIM, set the **Generate Elevation Fill** property to **On**.

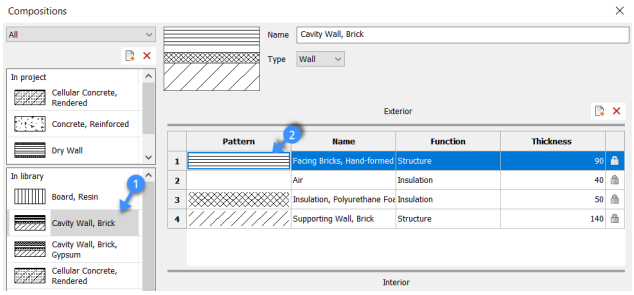
7 Select the viewport and choose **Update BIM Section** in the **Model** command tab in the Quad.

8 **(Optional)** Copy the viewport, then set the **Generate Elevation Fill** property to **Off** to create two versions of the section result, one with and one without elevation hatch.

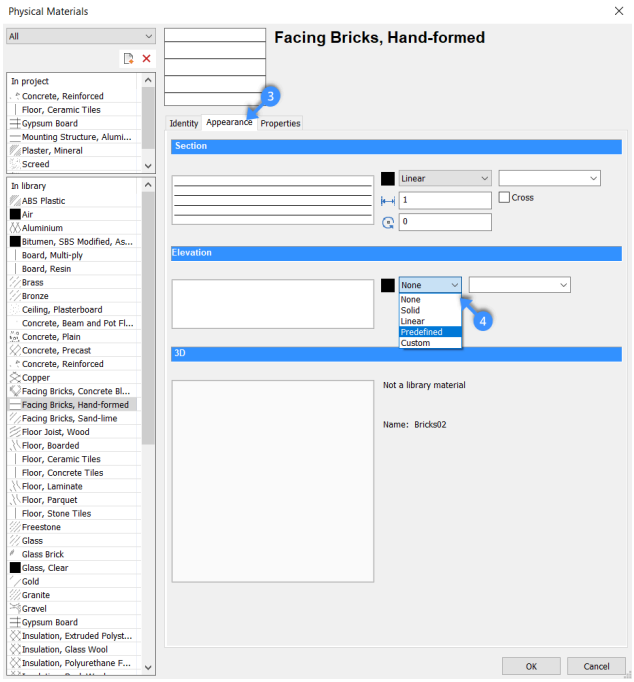
20.7.10.3 Configure the hatch

1 Go to the **Physical Materials** dialog by typing BLMATERIALS or BLCOMPOSITIONS (1) into the Command line.

2 Click on the composition and double click on the pattern of the composition to open the **Physical Materials** dialog (2).

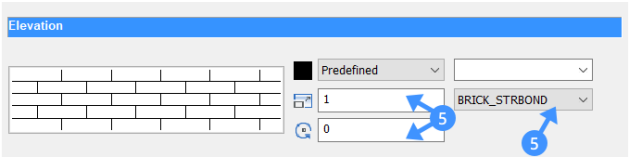


3 To configure the texture or solid fill, go to the **Appearance** tab page (3). The elevation hatch is set by default to **None**. Click the elevation hatch type button and select the type of hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom** (4).



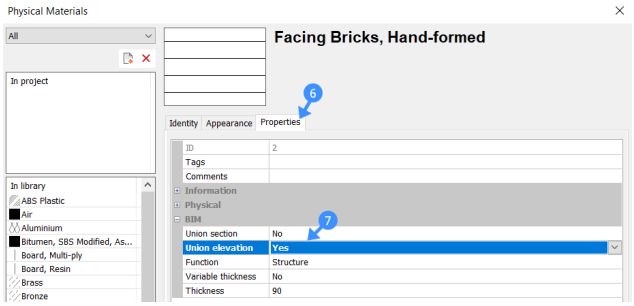
4 Configure the hatch. Define the scale, angle, color and/or layer of the pattern. (5)

Nota: Type a name in the layer field or click the arrow button and select a layer in the list.

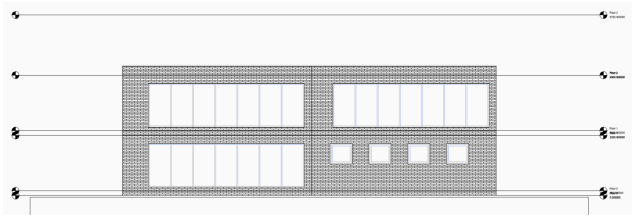


5 To remove connection lines in the elevation between different solids of the material go to the **Properties** tab (6) and set **Union elevation** to **Yes** (7).

Nota: **Union elevation** works independently from the **Union section** setting.



6 After regeneration, the hatch will appear on the elevation.



20.7.11 Enscape integration

20.7.11.1 Commands

STARTENSCAPE, CLOSENESCAPE, ENSCAPEEXPORTSETTINGS, ENSCAPEIMPORTSETTINGS, ENSCAPEBATCHSCREENSHOT, ENSCAPELOADVIDEOPATH, ENSCAPEPANORAMA, ENSCAPESAVEVIDEOPATH, ENSCAPESCREENSHOT, ENSCAPESETTINGS, ENSCAPESTARTVIDEO, ENSCAPESTOPVIDEO, ENSCAPETOGGLELIVEUPDATE, ENSCAPECREATESTANDALONE

20.7.11.2 About

Enscape is a real-time rendering plugin for BricsCAD. With a single click, you can start Enscape in seconds and walk through your fully rendered 3D model. There is no need to export to other programs. All changes in BricsCAD are immediately available for display and evaluation in Enscape.

You can quickly explore different materials and design options to present to clients. With the option to create output renders, videos, and panoramas of your project, you can send the output files to your clients or colleagues, which allows for quick demonstrations. You can edit the changes in BricsCAD, and Enscape will immediately show the changes you have made to the project, even in VR.



20.7.11.3 Enscape system requirements

The minimum requirements to run Enscape:

- Windows 7 64 Bit or higher.
- Intel i5 CPU or higher.
- NVIDIA or AMD GPU with 2GB VRAM.
- Supports OpenGL 4.3.
- NVIDIA GeForce GTX 660/Quadro K2000 and newer.
- AMD Radeon R9 260/FirePro W5100 and newer.
- 4 GB RAM (8 GB recommended).

- 2 GB Video-RAM (4 GB recommended).

For more information about the Enscape system requirements, visit https://enscape3d.com/knowledgebase/system-requirements/.

For more information about specific Enscape features, you can visit their knowledge-base on their website.

20.7.11.4 How to install/uninstall Enscape

1 Download the Enscape-BricsCAD Connection from the Application Store.

2 The installer copies the necessary files to the 'Program Files\Bricsys\Enscape'-folder.

3 Restart BricsCAD.

4 You now have a new tab in the ribbon. Use this to launch Enscape.

Nota: The first time you will start the Enscape-plugin, you will be asked to provide a license key. If you don't have a license key yet, you can start by choosing the trial option.

How to uninstall Enscape

Uninstall the connector from **Add or Remove Programs** in Windows, by searching Enscape and clicking **Uninstall**.

20.7.11.5 Navigate your model in Enscap

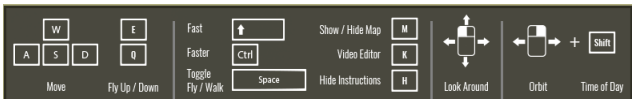
1 Click on **Start Enscape** in the ribbon **or** type in STARTENSCAPE in the Command line.

Changes made in BricsCAD (e.g. deleted furniture, new material, a new section, etc.) can be seen instantly in Enscape.

2 Follow the instructions to navigate through your model.

3 You can switch between **Walk** mode and **Fly** mode by pressing the Space bar.

Type H to hide the instructions.



20.7.11.6 Settings in Enscape

Open the **Enscape Settings** dialog by clicking **Settings** in the ribbon or type in ENSCAPESETTINGS in the Command line. When you change the settings, they will be automatically saved for the user, not in the drawing. There is also an option to create and import presets for the Enscape settings.

You can find more information about saving and importing presets in the following link: https://enscape3d.com/community/blog/knowledgebase/settings-presets/.

The **Enscape Settings** dialog consists of 7 tabs:

- **General**: general **Rendering Style** options, **Depth of Field** and **Exposure** settings.
- **Image**: Contrast and Color settings.
- **Atmosphere**: Horizon/background options, **Fog**, **Clouds** and **Sky Orbs**.
- **Input**: Mouse navigation settings.
- **Advanced**: Spectator options.

- **Capture**: **Image**, **Video** and **Panorama** options.
- **Customization**: Interface options.

20.7.11.7 Render materials in BricsCAD

In BricsCAD BIM, there are two methods to add render materials to your model.

- Directly apply to render materials (without compositions) to the solids/surfaces in your model, using the **Render Materials** panel.
- Apply render materials via compositions, using the **Compositions** panel.

Nota: When render materials are attached to an entity, both applied directly with the **Render Materials** panel and compositions attached, the composition materials will be shown (when the LEVELOFDETAIL system variable = 2).

Thus, if an entity already has a composition and you want to change the render material, don't try to over-rule it with the Render Materials panel, but change the composition plies instead.

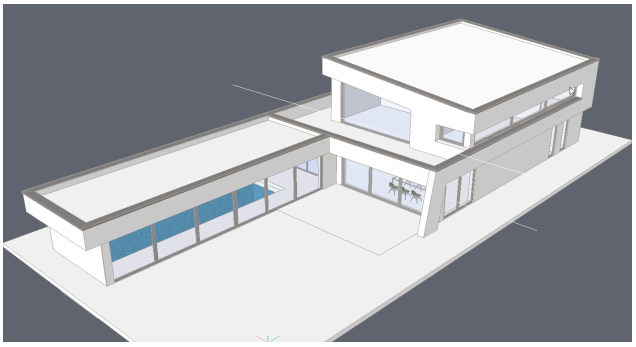
Render materials with the Render Materials panel

- To assign a material to a single entity
  a Select the material in the **Render Materials** panel.
  b Click on the entity or press and hold the left mouse button, then drag the material onto the entity.
- To assign a material to multiple entities
  a Select the entities.
  b Search the material in the **Render Materials** panel.
  c Click on the material.

  For more information about render materials, see the article Rendering materials.

Render materials with Compositions

1 Set the LEVELOFDETAIL system variable to 2. The materials of the composition plies will display.



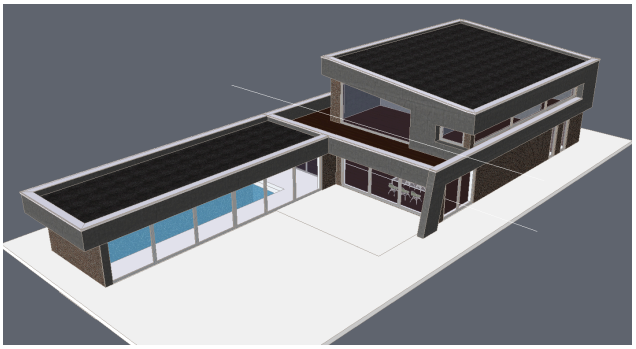


Figure 1: Level of detail = 0,Figure 2: Level of detail = 2

2 Open the **BIM Compositions** panel.

For more information about compositions, see the article BIM compositions.

3 Double-click on the composition **Cavity Wall, Brick**. The **Compositions** dialog box will display.

4 Double-click the material **Facing Bricks, Hand-formed** to edit the ply material. The **Physical Materials** window displays.

5 Click on the material image under **Appearance** > **3D** to change the render material for the ply.

Editing render materials in BricsCAD

1 To edit all the render materials used in your drawing, type in MATERIALS in the Command line.

20.7.11.8 Keywords in Enscape

Grass

Any time the word ‘Grass’ is used in a material name, grass will be rendered in Enscape. You can use diffuse colors to choose options of green, but also texture maps can be used as well.

1 Go to the **Materials** dialog box.

2 Click on **Add New Material**.

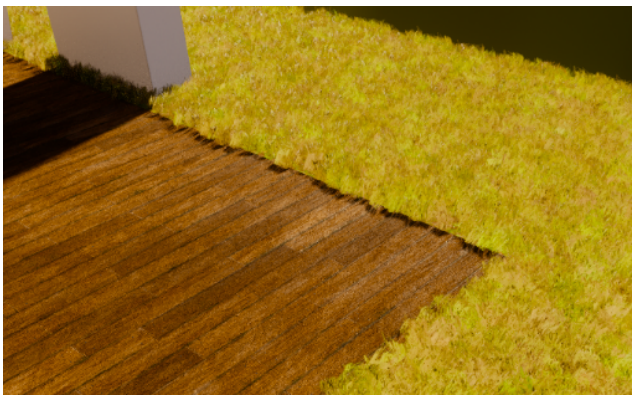
3 Give the new material a name that includes the name 'Grass', for example, ‘Grass_ground’.

4 Close the **Materials** dialog.

5 Select the solid that you want to render in grass.

6 Open the **Properties Panel**.

7 Change the property **3D Visualization** > **Material** from **ByLayer** to **Grass_ground**.



Water

Any time the word 'Water' is used in a material name, water will be rendered in Enscape. You can also create your own water material by including the word 'Water' in the material name and give it a diffuse color in a shade of blue.

1 Select the solid that you want to render in water.

2 Open the **Render Materials** panel and choose the folder named **Water**.

3 Drag and drop material **Water Blue Two** into the drawing.



20.7.11.9 Saved views in Enscape

1 Navigate in the BricsCAD drawing until you get a view that you want to save.

Nota: Make sure your viewing style is in **Perspective** to have a more accurate saved view in Enscape. You can check this in **Properties Panel** > **View** > **Perspective** > **ON**. And close the **Properties Panel** afterward.

2 Click on **Create View** in the Enscape ribbon or type -VIEW in the Command line.

You are prompted: View [? to list/Delete/Orthographic/Restore/Save/Window]:

3 Type **S** and press Enter to save the view.

You are prompted: Save view as:

4 Name the view and press Enter to confirm.

5 You can delete or adjust existing views by clicking **Views** in the Enscape ribbon **or** type in VIEW in the Command line.

6 In the Enscape window: on the right-hand side of the window, you can find all the saved views in the drawing.

7 Click on the view you want to see to switch between views.



20.7.11.1 Render images with Enscape 0

1 Navigate through the Enscape window.

2 **Or** you can click on a saved view on the right side of the Enscape window. The saved view will display.

3 Go back to BricsCAD and select **Render Image** in the Enscape ribbon to render the current view or type in ENSCAPESCREENSHOT in the Command line.

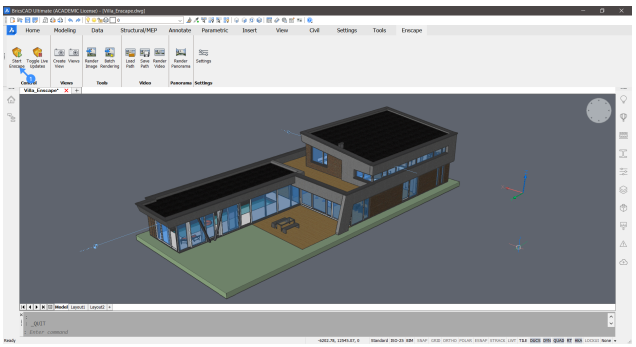
4 Select **Batch Rendering** in the ribbon, or type in ENSCAPEBATCHSCREENSHOT in the Command line, to render all the saved views to a folder.

5 Specify a destination file to save the rendered image and click **Save**.

The rendered image is now created.

20.7.11.1 Render a video with Enscape 1

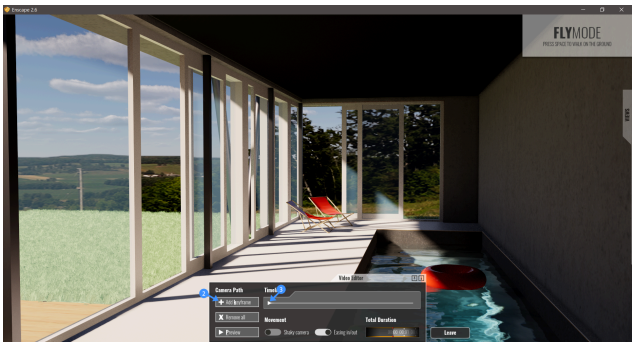
1 Click on **Start Enscape** in the ribbon or type in STARTENSCAPE in the Command line.



2 In the Enscape window: type K to open the **Video Editor**.

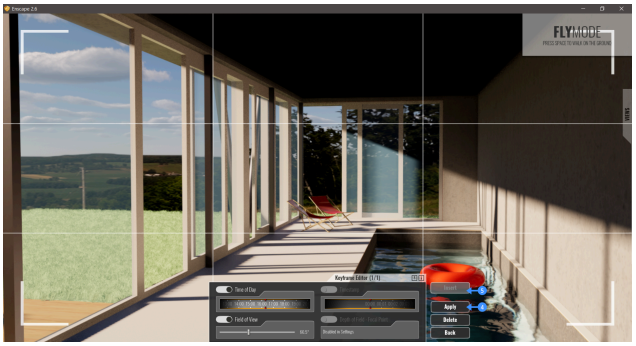
3 Click **Add Keyframe** (2) to add the first keyframe.

4 Click on the triangle to open the keyframe settings (3).



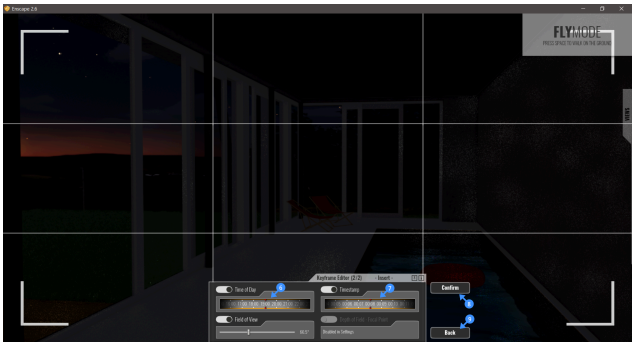
5 Change the parameters as required.

6 Click **Apply** (4).



7 Click **Insert** (5) to add a second keyframe.

8 Change the **Time of Day** to 7 pm (6) and the **Timestamp** to 8 seconds (7).



9 Click **Confirm** (8).

10 Click **Back** (9).

Nota: Before you click **Leave** to exit the **Video Editor**, your video path will not be automatically saved. So, before you exit your **Video Editor**, you should save your video path using the command ENSCAPESAVEVIDEOPATH.

11 Go back to BricsCAD and click on **Render Video** in the Enscape ribbon. Specify a destination file and

click **Save**. The video is now rendered and saved.



20.7.11.1 Export a standalone .exe
2

The Executable export function provides the same experience and quality that Enscape does, but does not require Enscape and BricsCAD to run.

1 Click **Export Standalone** in the Enscape ribbon or type in ENSCAPECREATESTANDALONE in the Command line.

2 A new dialog will pop up asking you where to save your standalone file. Choose your destination file, name your standalone and click Save.

20.7.12 Generating Drawings

20.7.12.1 Commands

BIMSECTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

20.7.12.2 About BIMSECTIONUPDATE

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 in a generated drawing. The result is placed in a new drawing, saved to the same folder as the model drawing unless a different folder is specified in the **Destination File** property of the BIM Section entity.



For more information about this command, visit the Command Reference article BIMSECTIONUPDATE.

20.7.12.3 Procedure: generating BIM sections

BIM Sections can be generated in 2 ways:

- In the model space.
- Using the BIM **Project Browser**.

Model space

1 Select one or more BIM section entities.

2 **(Optional)** Edit the section properties (e.g. **Scale**, **Name**, **Destination File**).

3 Choose **Generate Section** in the **Model** command tab of the Quad.

BricsCAD reports the number of generated sections and their name(s) in the Command line 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

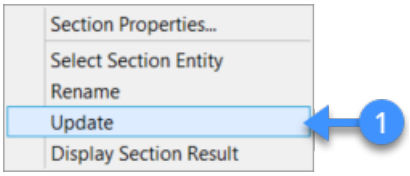
In the example above 2 plans, 4 elevations, and 2 cross-sections were generated and exported to 3 drawings.

Project browser

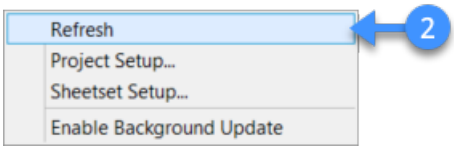
1 In the **Sections** segment of the **Project Browser**, right-click the section.

2 **(Optional)** Choose **Select Section Entity** to change its properties in the properties panel.

3 Choose **Update** (1).



4 Choose **Refresh** in the project browser menu (2).



A new section is generated in the **Sheets** segment of the **Project Browser**.

Notes	• BIM_SECTIONS_PREVIOUS layer: the result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. This is frozen by default. • BIM_VIEWPORTS layer: generated viewports are placed on the dedicated BIM_VIEWPORTS layer if desired. • The **SheetSet manager** automatically and silently adjusts the file paths after the project folder has been relocated if the same folder is used for all files.

20.7.12.4 Open BIM sections

BIMSECTIONOPEN navigates between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directionally.



For more information about this command, visit the Command Reference article BIMSECTIONOPEN.

BIM Sections can be opened in different ways:

- From the 3D Model.

- Using the BIM **Project Browser**.

From the 3D model

To open a 2D document from the 3D model:

1 Select one or more BIM sections.

2 Choose **Open Model** in the **Model** command group of the Quad.

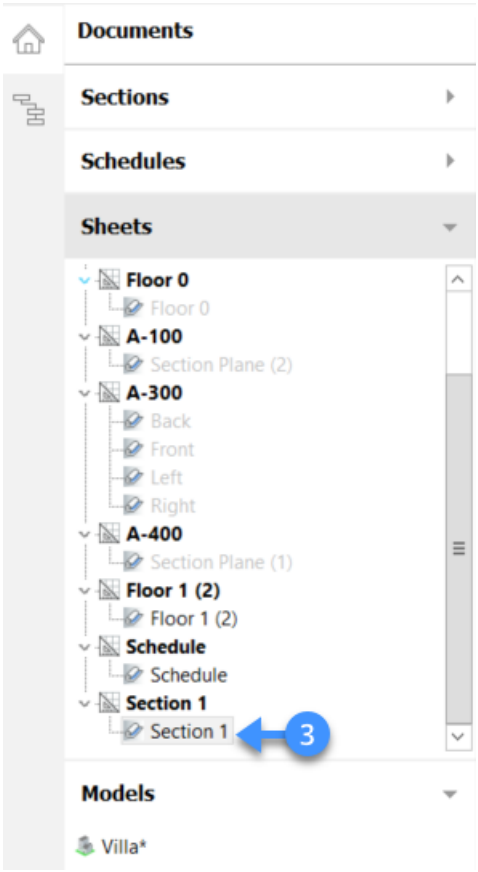
3 The 2D section drawing(s) open(s).

Project browser

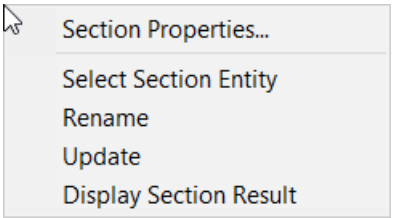
To open the 2D section drawings from the **Project Browser**, do one of the following:

- Double click the sheet view in the **Sheets** segment of the **Project Browser** (3).

  The 2D section drawing opens with the selected sheet view.



- In the **Sections** segment of the **Project Browser**, right-click the section and choose **Display Section Result** in the context menu.

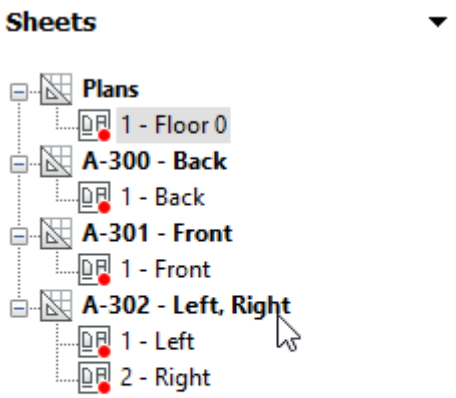


20.7.12.5 Update BIM sections

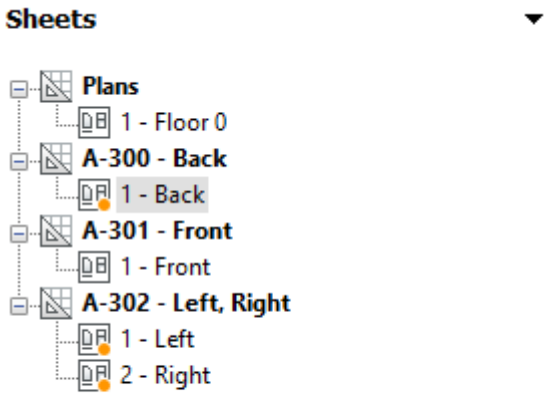
BIM Sections can be updated in different ways:

- In the 3D model.
- In the generated 2D documentation file.
- In the BIM **Project Browser**.

When the 2D documentation is no longer in sync with the 3D model, the corresponding sheet views display a red mark in the **Sheets** segment of the **Project Browser**.



Type BIMIFY command, then the corresponding sheet views display an orange mark in the **Sheets** segment of the **Project Browser**.



20.7.12.6 Update BIM sections in your 3D model

To update BIM Sections via your 3D model using the same process as used to generate views.

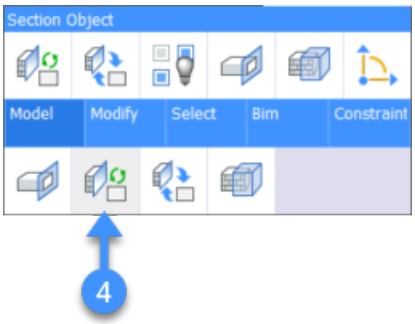
Update BIM sections in the 2D documentation file

A red frame around the section result indicates that the BIM Section generated block is no longer in sync with the 3D model and needs to be updated. This can be done in **Model** space or in the **Layout**.

Update a BIM section in model space

1 Place the cursor over the section block to update.

2 Choose **Update Section** in the **Model** command group in the Quad (4).

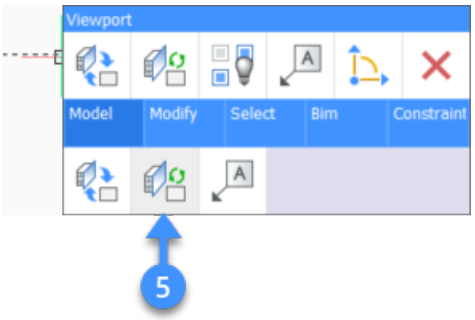


Update a BIM section in a layout

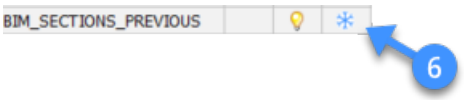
1 Click the **Layout** tab.

2 Place the cursor over the viewport of the BIM Section to update.

3 Choose **Update Section** in the **Model** command group of the Quad (5).



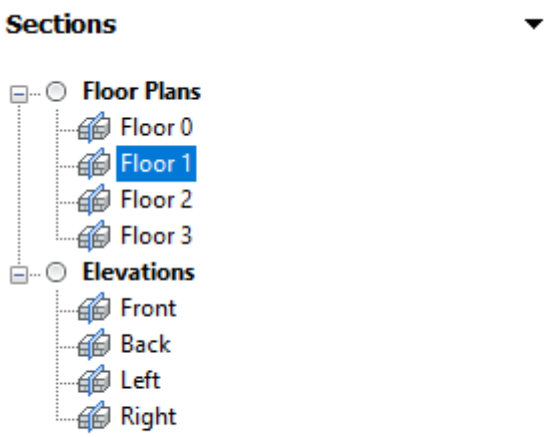
The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer (6) to check the modifications.



The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta.

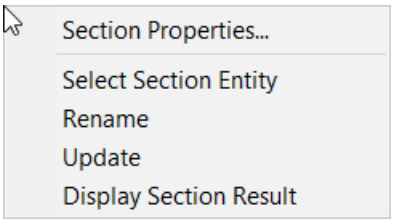


Update BIM sections in the project browser



1 Right-click on a section in the **Project Browser**.

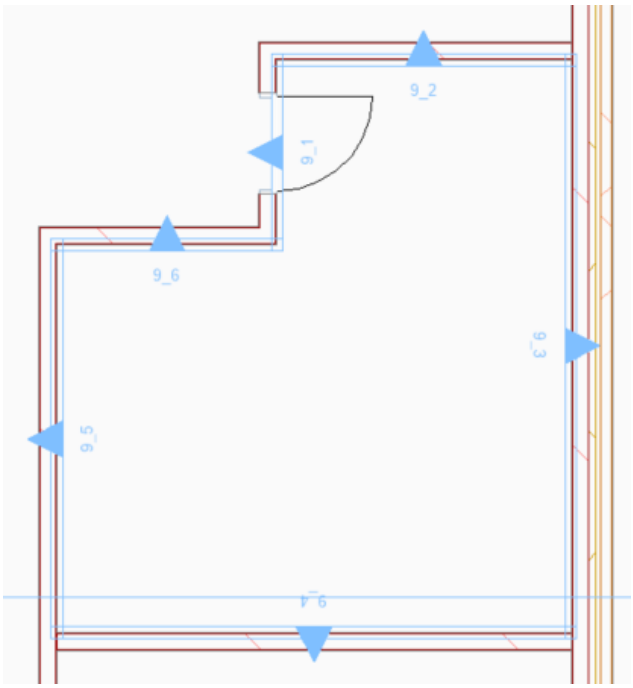
2 Select **Update** in the context menu.



20.7.12.7 Section indicators

A generated plan and reflected ceiling plan provide section indicators for interior elevations and detail sections if they intersect with the section plane.

The section indicators are stored on the dedicated BIM_SECTIONS layer. To change the visibility of indicators, turn the layer ON or OFF.



20.7.13 Working with Point Clouds in BIM

20.7.13.1 Commands

POINTCLOUDDETECTFLOORS, POINTCLOUDFITPLANAR, POINTCLOUDPROJECTSECTION

20.7.13.2 About POINTCLOUDDETECTFLOORS

The new POINTCLOUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

Section planes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and section planes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

If the point cloud building has multiple floors, there will be multiple section volumes.



For more information about this command, visit the Command Reference article POINTCLOUDDETECTFLOORS.

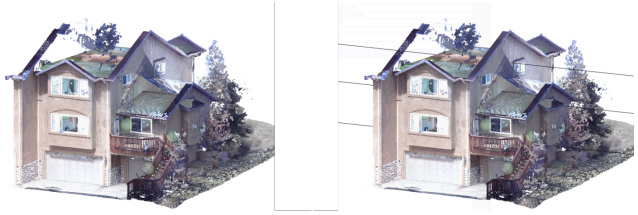
Detecting floors from Pointclouds

1 Type POINTCLOUDDETECTFLOORS in the Command line.

2 **You are prompted**: Operate on full point cloud or a selection? [Full/Selection]

- **Full**: The command will search the entire point cloud for floors and generates a section for each of them.
- **Selection**: The command only runs on the selected area.

3 Press Enter.



Nota: This volume selection is a detail section of the selected building. You can manipulate the section box if you want to include more or fewer points from the point cloud using the green grips.

20.7.13.3 About POINTCLOUDFITPLANAR

POINTCLOUDFITPLANAR is used to semi-automatically make 3D geometry based on a point cloud scan. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

Select planes in the point cloud to generate 3D solids based on the given geometries. These 3D solids can be further used in the process of creating a full BIM model.



For more information about this command, visit the Command Reference article POINTCLOUDFITPLANAR.

Creating 3D solids from the bubble view

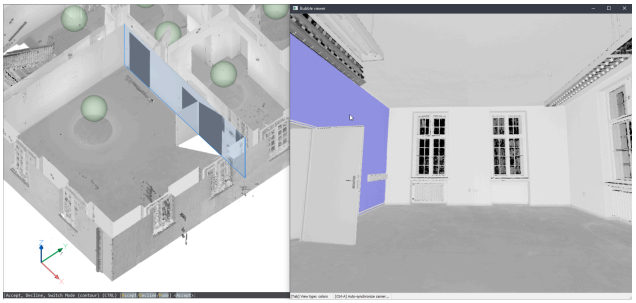
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.

1 Double-click a green point cloud bubble to open the **Bubble viewer**.

2 Type POINTCLOUDFITPLANAR in the Command line.

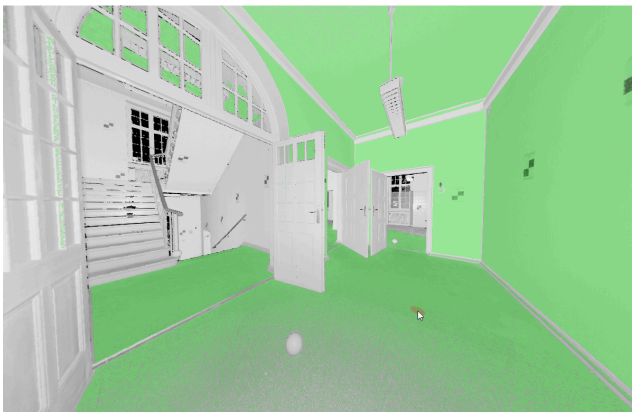
3 Select the desired wall.

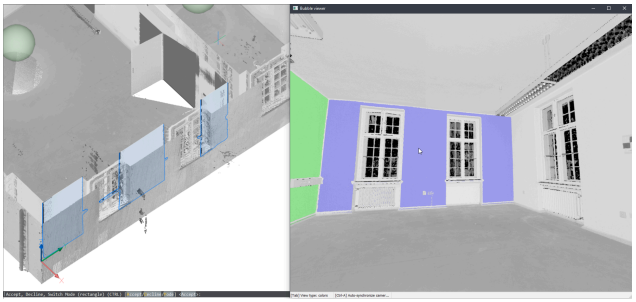




4 Press A to accept the selection.

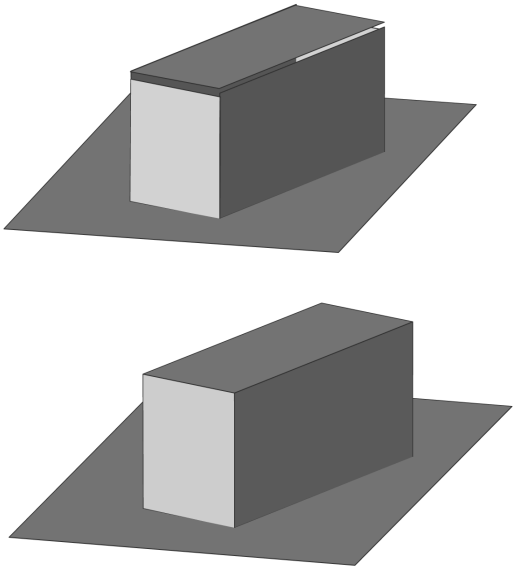
5 **(Optional)** Select more walls.





6 Close the **Bubble viewer**.

The surfaces are created and you can edit them to closed spaces with LCONNECT or other connecting tools.



Create 3D entities from the model space

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time but it has 2 advantages by searching multiple scan positions:

- It can create larger surfaces where only parts are visible in each scan position.
- It can detect wall and slab thickness since it can take the opposite surface into account.

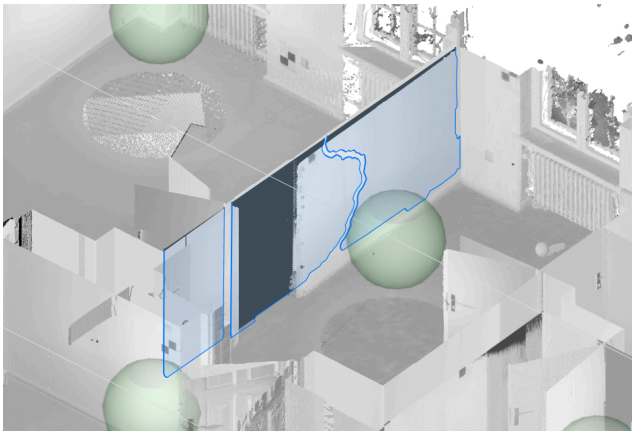
1 Type POINTCLOUDFITPLANAR in the Command line.

2 Select a wall where you want to create a plane or solid.

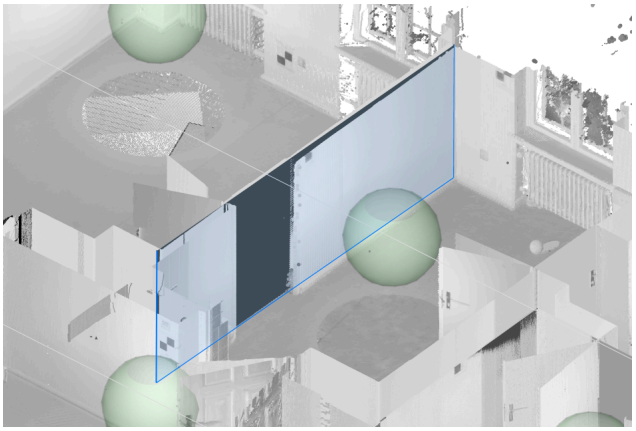
3 You are prompted: Accept, Decline, Switch Mode (contour) (CTRL) [Accept/Decline/Mode]:

- Press A to create the wall.
- Press D to decline.
- Press M or CTRL repeatedly to switch between different modes.

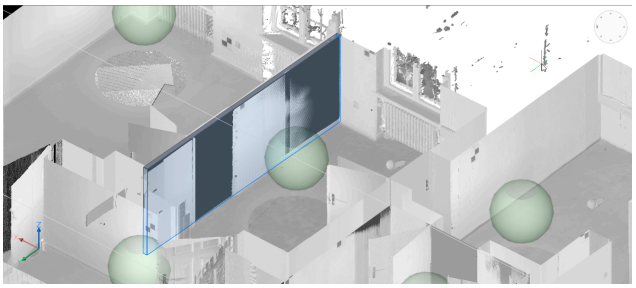
Contour mode will create the best fitting plane:



Rectangular mode will create a rectangular plane around region of planar points:



Solid mode will create a solid from two parallel point cloud planes:



Nota: If the point cloud exists out of multiple rooms, you can use the solid mode to automatically create solid walls.

4 Press A to create the wall.

These solids can be the base for a more complex BIM Model. Use modeling commands to complete the building and add BIM data with **Bimify**, **Automatch** and **Propagate** to quickly transform your point cloud into a complete BIM Building.

It may be helpful to turn **on** the **Point Cloud Closest Point Esnap** setting. This snap setting significantly improves your ability to select relevant point cloud points.

20.7.13.4 About POINTCLOUDPROJECTSECTION

POINTCLOUDPROJECTSECTION automatically generates a raster image with optional contour lines out of a defined section box. These lines can be easily manipulated into a detailed 2D drawing.

The command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building with POINTCLOUDDETECTFLOORS. You can use these sections to generate 2D lines to create a 2D floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



For more information about this command, visit the Command Reference article POINTCLOUDPROJECTSECTION.

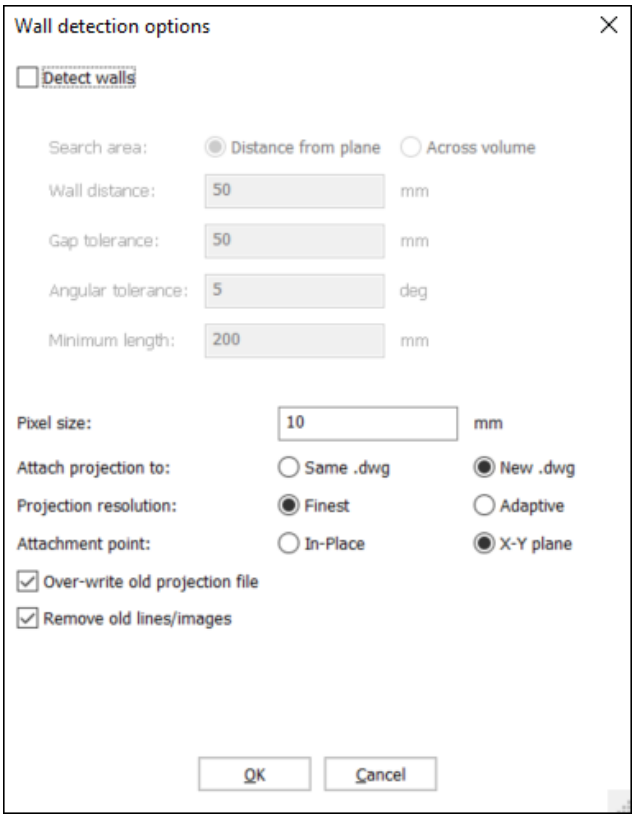
20.7.13.5 Procedure: creating a raster image from a point cloud

1 Type POINTCLOUDPROJECTSECTION in the Command line.

2 Select a **Section** volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section. Turn **Detect Walls** off.



4 Click **OK** to generate the raster image.



Point Cloud scan by Realistic Visual styles.

20.7.13.6 Procedure: detecting walls from a point cloud and generate in 2D

Making horizontal sections from point clouds

1 Type POINTCLOUDPROJECTSECTION in the Command line.

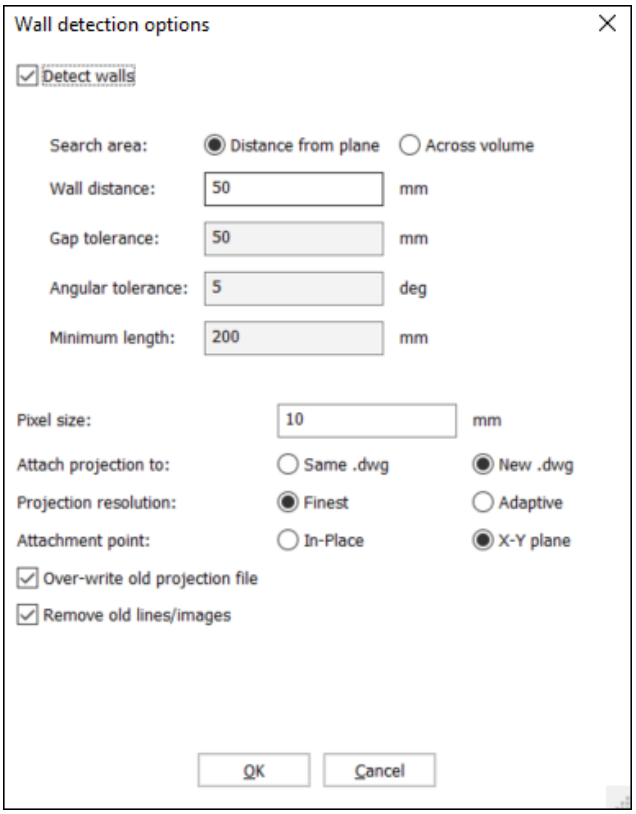
2 Select a section volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section.

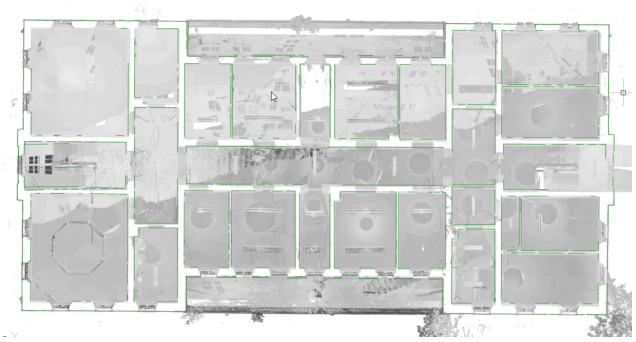
Nota: See the command reference page for more information about the customization options;

Nota: Turn on **Detect Walls** to generate the green lines as seen in the image below.



4 Press Enter to create the projection.

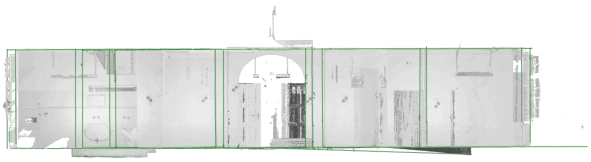
Nota: If there are some missing lines, you can draw them yourself by tracing the background image.



Making vertical sections from point clouds

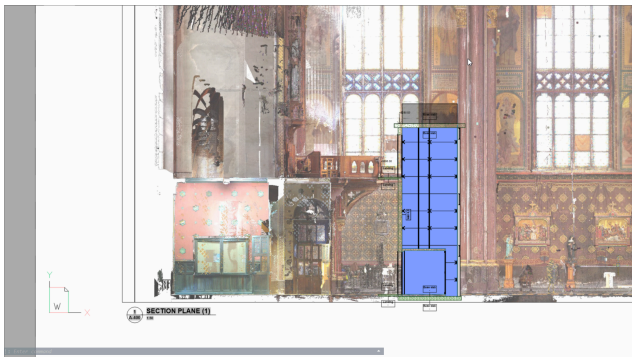
Make a vertical section box on the place you want the vertical projection.

Make sure that the blue round arrow aims to the side you want your projection.



20.7.13.7 Generating BIM sections with point clouds

In some cases it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2D geometry, in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern interventions are made in historical buildings.



Point Cloud scan by Realistic Visual styles.

20.7.14 Interior elevations

20.7.14.1 Command

BIMSECTION

20.7.14.2 About

Interior elevations show wall elevations and floor plans for individual Spaces. They provide the details and dimensions of building elements. This can be particularly useful for interior designers or to make a layout of power sockets and light switches. In BricsCAD they can be automatically defined by a BIM Space.

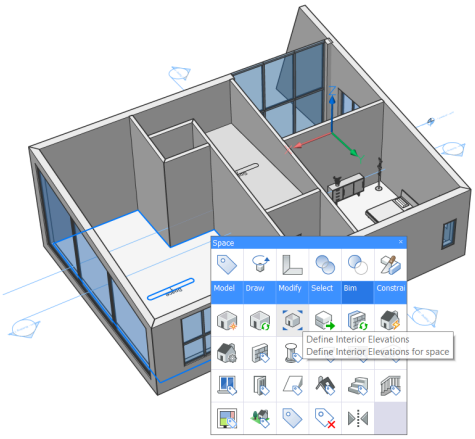


For more information about this command, please read BIMSECTION.

20.7.14.3 Create interior elevations from a BIM space

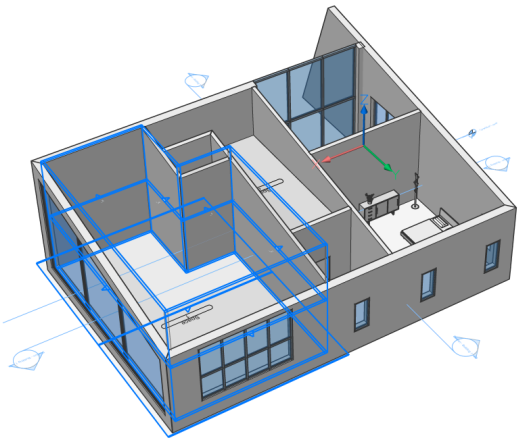
1 Hover the cursor over a Space Tag and select **Define Interior Elevations** from the **BIM** Tab in the Quad
Or type BIMSECTION in the Command line, followed by i and Enter to choose the interior option.

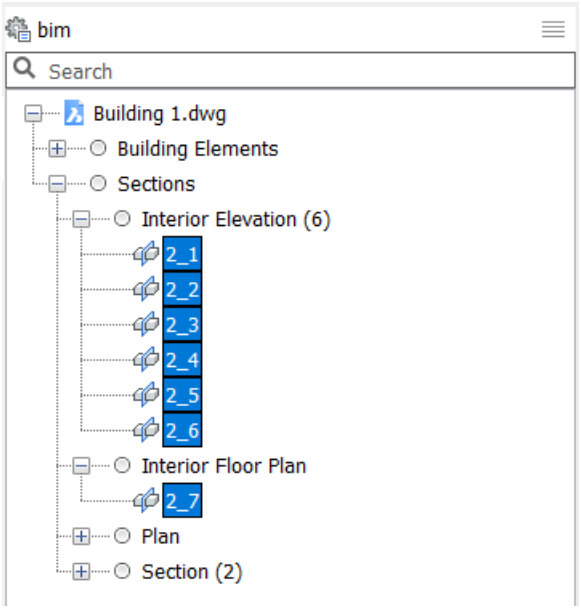
You are prompted: Select BIM Spaces to generate Interior Elevations or [Attach section]:



2 (Skip if you used the Quad) Select the BIM Space for which you want to create the interior elevations and hit Enter.

3 The interior elevations for each wall and the interior floor plan are now defined. They can be found in the **Structure Browser**.

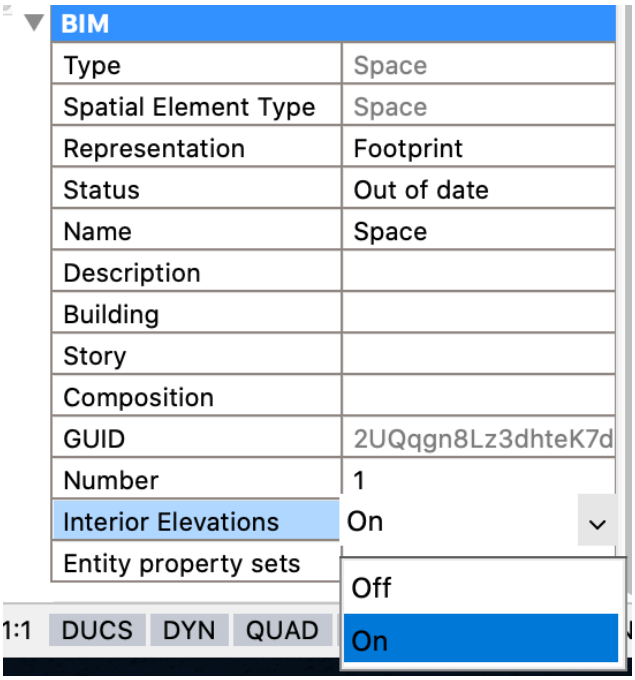




Six interior elevations and one interior floor plan have been created and will appear in the structure browser.

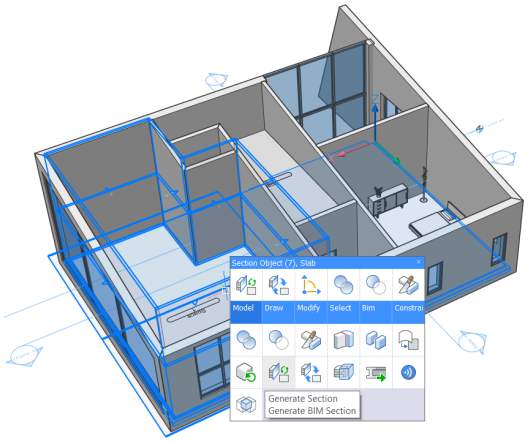
Nota: If there already are interior sections for a Space, running BIMSECTION again on that Space will have no effect. To regenerate the interior elevations, you must delete the existing interior elevations in that Space and run BIMSECTION again.

4 To show or hide the interior elevations you can use the HIDE/ISOLATE/UNISOLATE objects commands **or** you can go to the properties of the Space and switch the interior elevations on and off in the **BIM** tab.



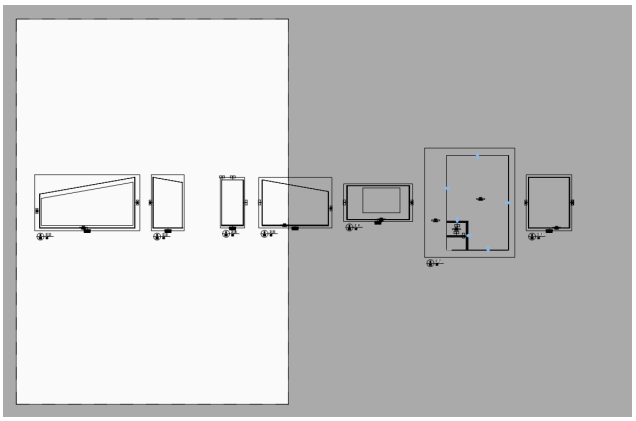
5 Select all elevations, as illustrated above and select **Generate Section** from the Quad to generate the

interior elevations on a sheet.



6 To open the section results click **Open Model**whilst the section planes are selected.

The sheet is automatically saved in the same folder as the model drawing. The following elevations have been produced and can be edited for the selected Space in the model above:



Nota: The section drawings are stored in the file **InteriorElevations** by default. If you generate sections of a second Space, the sections will be added to the same file as the previous ones.

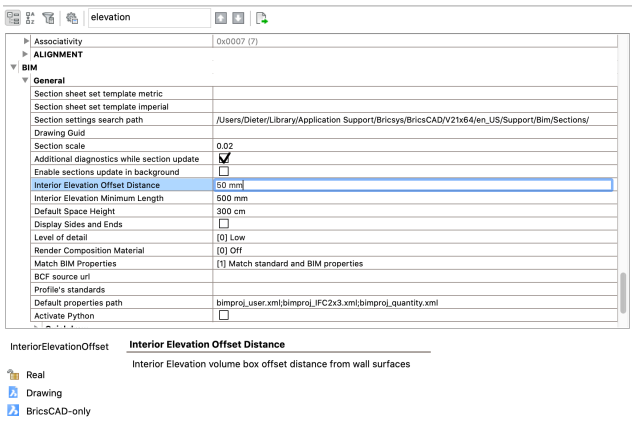
If you need more information on how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see **Annotate** > **Creating linear dimensions**.

20.7.14.4 Edit the size of the interior elevations

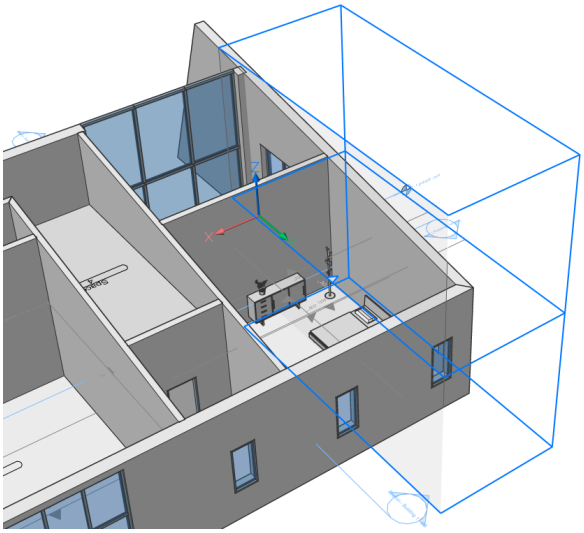
It's possible to edit the size of these sections to include furniture for example. If you create interior elevations for a BIM Space, the offset value is 50mm by default. To include furniture in a section, you have to change this value.

1 To change this setting type INTERIORELEVATIONOFFSET in the Command line and hit Enter. Type a new value and press Enter **or** click on two points in your drawing and the distance between them will be selected.

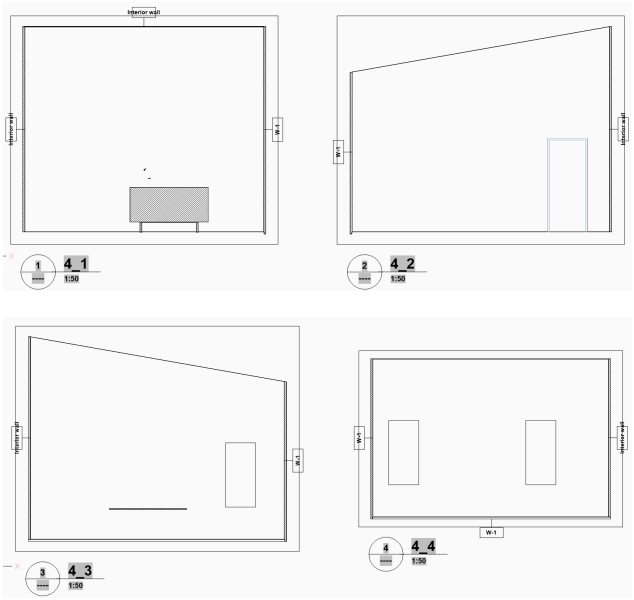
You can also change the offset by typing Settings in the Command line, search for elevation and adapt the **Interior Elevation Offset Distance** value.



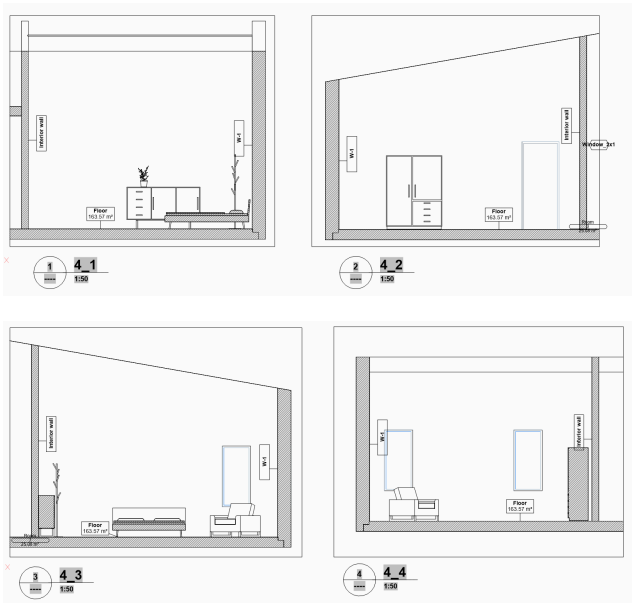
2 If you make new Interior Elevations, the offset distance will be the chosen value. This is also visible when you select the interior elevations in your drawing:



3 This results in more detailed section sheets with the furniture included.



Without specified offset distance (50 mm)



With specified offset distance (2100 mm)

20.7.15 Schedules

20.7.15.1 Open the BIM project browser panel

1 Move the cursor over a toolbar, ribbon item or a dockable panel.

2 Right-click.

3 Choose **Panels** > **BIM Project Browser** in the context menu.

See the article Project browser, for more information about the BIM **Project Browser**.

20.7.15.2 How to create schedules

Schedules are exports, entity properties, block attributes and drawing information to tables on sheets and tables.

Step 1: create a project

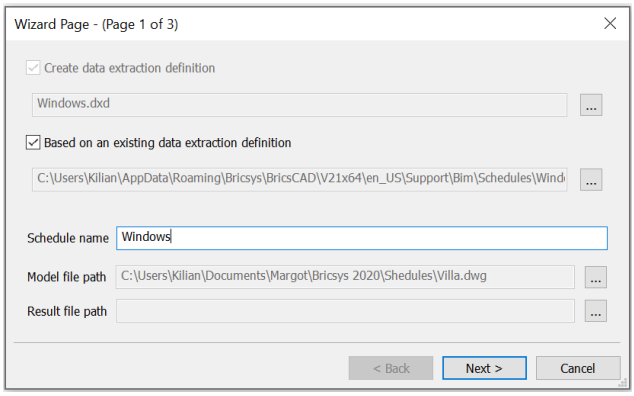
For more information about creating a project, see the article Project browser.

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (Wizard)...** to open the wizard.

Step 2: the schedule wizard

A wizard of 3 pages displays, this will help you to create a schedule.

- Page 1



(1) Create data extraction definition creates a new .dxd (data extraction definition) file:

a Turn the option on.
b Click the **Browse** button (...).
c In the file dialog box, select a folder.
d Enter a name in the **File** name field.
e Click the **Select Folder** button.

Nota: By default, the .dxd file will be placed in the project folder. To change this path click the **Browse** button.

(2) Based on an existing data extraction definition uses an existing .dxd file as the template for the new one:

a Turn the option on.
b Click the **Browse** button (...).

Nota: The .dxd files are stored in ..\bim\schedules serve as templates.

c In the file dialog box, select a .dxd file.
d Click the **Open** button.

Nota: By default .dxd files are saved in the ...\Bim\Schedules subfolder of the **Support** folder. You can use newly created custom .dxd files too. Also, you can create a .dxd files by yourself

using a text editor. Please scroll down for the procedure. "How to create and edit your own Data Extraction file" in this article.

You can either choose a new name for your schedule or keep the default name.

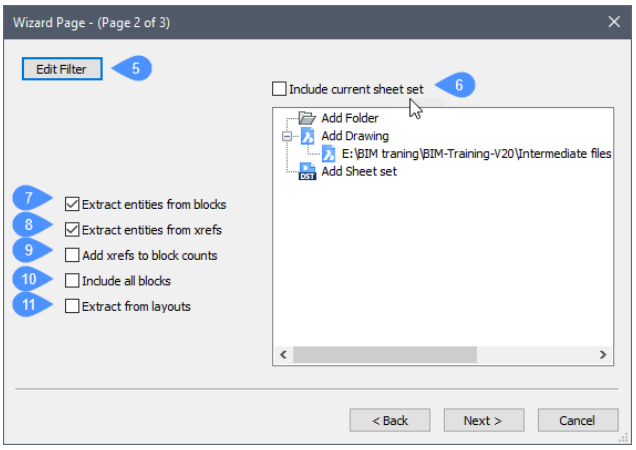
(3) Model file path: The model path links to the model. This is filled in automatically if the wizard is started in the model file. If the wizard is started in another file from the project, link the model file:

a Click the **Browse** button (...).
b In the file dialog box, select the model file.
c Click the **Open** button.

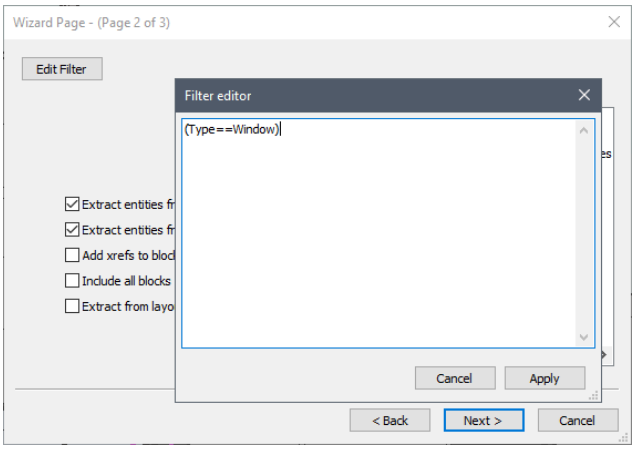
(4) Result file path: The **Result file path** determines where the schedule is created. When left blank, the Schedule is created as a new sheet:

a In the file dialog box, select the sheet file.
b Click the **Open** button.
Click **Next** for the next page.

- Page 2



(5) Edit Filter: The filter selects which BIM entity types are included in the Schedule. Use an existing data extraction definition to automatically define the filter. See the Editing Data Extraction Definition (.dxd) files to learn more about the filter syntax.



(6) Include current sheet set: Adds the current sheet set for data extraction.

(7) Extract entities from blocks: Adds entities in blocks which will also be processed in the schedule.

(8) Extract entities from xrefs: Adds all entities in xrefs which will also be processed in the schedule.

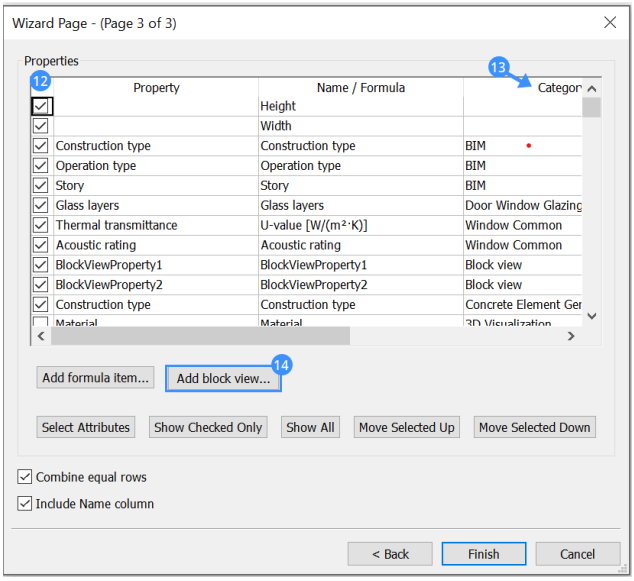
(9) Add xrefs to block counts: Counts selected xrefs as blocks.

(10) Include all blocks: Includes all blocks into the schedule.

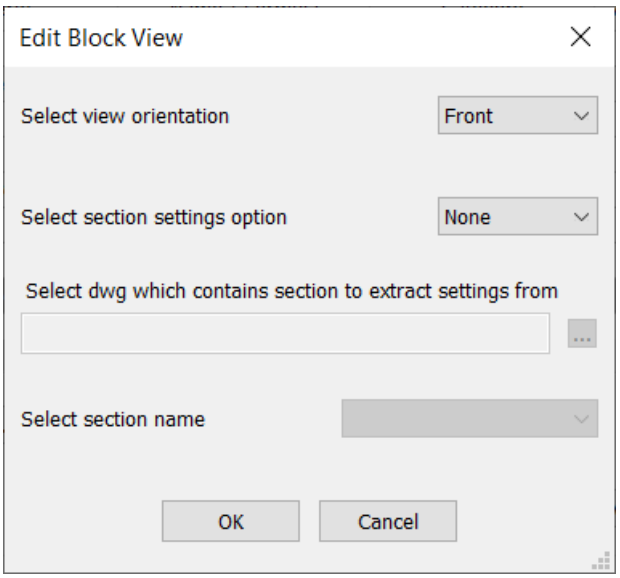
(11) Extract from layouts: When toggled on, all entities layouts will also be processed in the schedule.

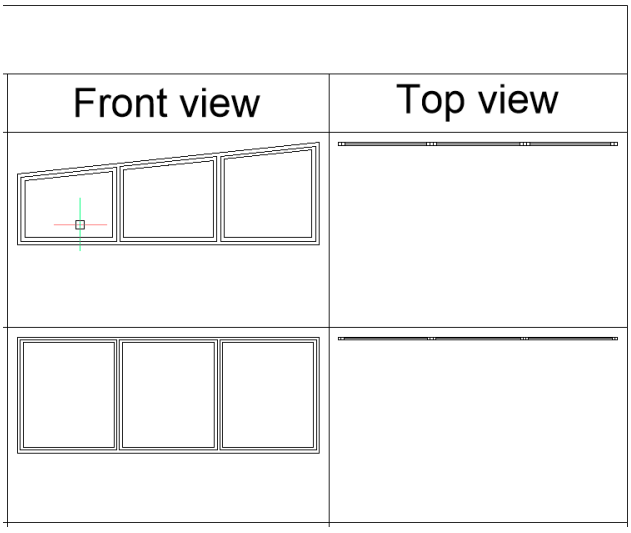
Nota: The Wizard automatically selects all entities in the drawing.

- Page 3



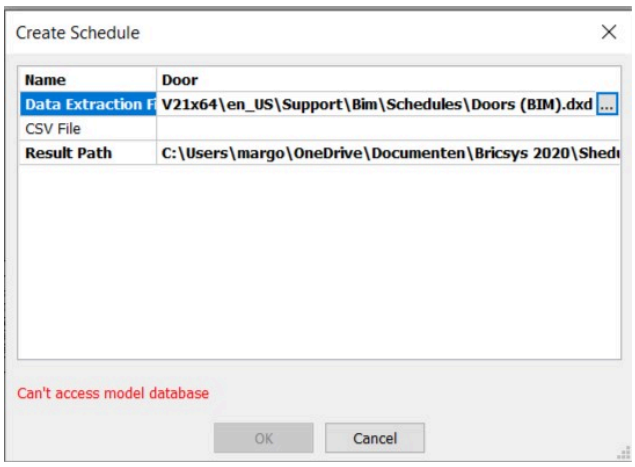
(12) Select the properties to be included in the schedule: click the checkbox to check/uncheck a property: selected (☑) or not selected (☐).
(**13**) Click on the **Category** column to change how the property is shown in the schedule.
(**14**) To include plan and elevation symbols, click on the **Add block view...** button.





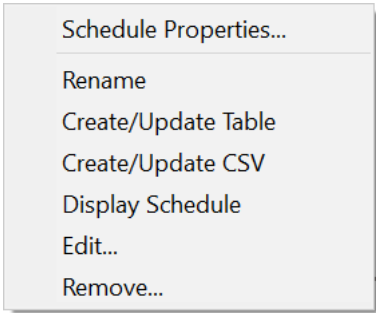
To sort the list, click the title in the column heading. Click again to sort in reverse order.
Click **Finish** to close the wizard (or click **Back** to switch back to page 2).

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (From dxd)...** to open the wizard.



20.7.15.3 Update and display schedules

1 Right-click on the new schedule (created in the BIM **Project Browser**). A context menu appears:



Schedule Properties...	**Name**: Renames the schedule. **Model**: Displays the link to the model file. **DWG Handle**: Displays the DWG Handle. **Data Extraction File**: Displays the link to the Data Extraction Definition. **CSV File**: Displays the link to the .csv file. **Result Path**: Displays the resulting path.
Rename	Renames the schedule.
Create/Update Table	Creates a sheet with the schedule or updates the sheet.
Create/update CSV	Creates a CSV file with the schedule information and saves it in the folder where de model file is located or updates the CSV file if it already exists.
Display Schedule	Displays the schedule in the resulting drawing. (If not already open, the drawing is opened.

Edit...	Opens the **Wizard Page** dialog box where you can edit your schedule. More information about this setup can be found in the procedure in this article.
Remove...	Removes the schedule.

2 Click on **Update**.

3 Once the schedule is updated, click on **Display Schedule**. The schedule is shown in the desired sheet. **Or** double-click on the schedule's name to display the schedule.

20.7.15.4 Procedure: create a window schedule

In this example, you will learn how to make a window schedule from scratch. You can create a door schedule in the same way.

Step 1: open schedule wizard

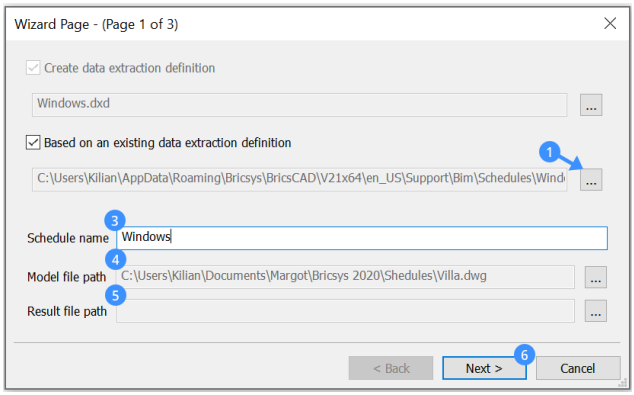
1 Open the BIM **Project Browser**.

2 Right-click on **Schedules**. A context menu appears.

3 Click on **Add Schedule (wizard)...** to open the wizard.

Step 2: the schedule wizard

- Page 1



For this example, use an existing data extraction definition.

a Click on the **Browse** (1) button. A folder pops up. This contains the predetermined data extraction definitions.

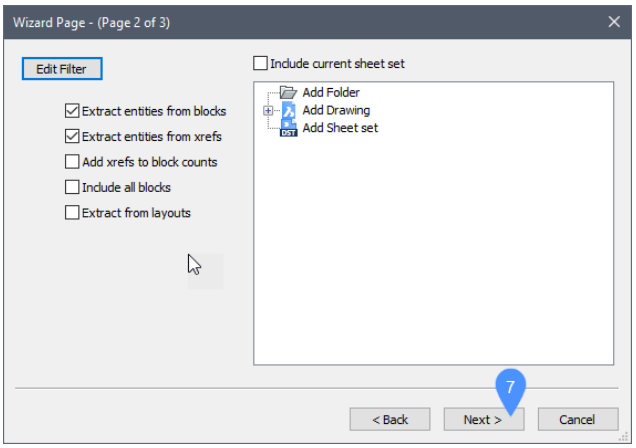
b In the folder, select *'Windows.dxd'*.

c Change the **Schedule name** (3) to, in this case, **Windows**.

d The **Model file path** (4) is correct because it was taken from the model file. To create the schedule on a separate sheet keep the **Result file path** (5) empty.

e Click **Next** (6).

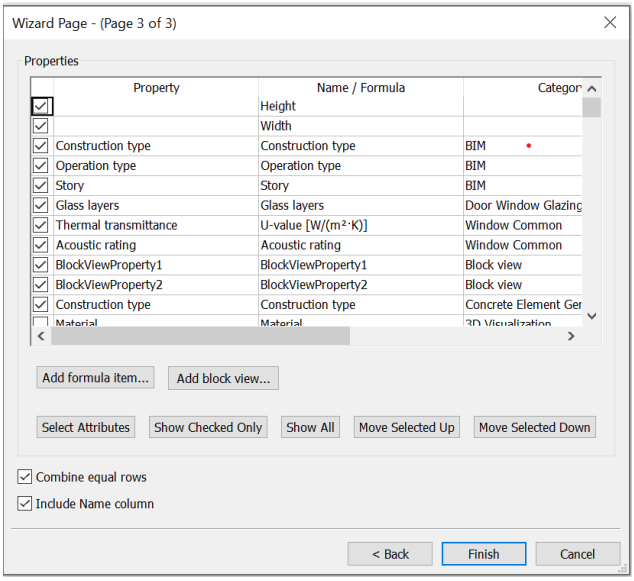
- Page 2



a The wizard automatically selects all the entities in the model.

b Click **Next** (7).

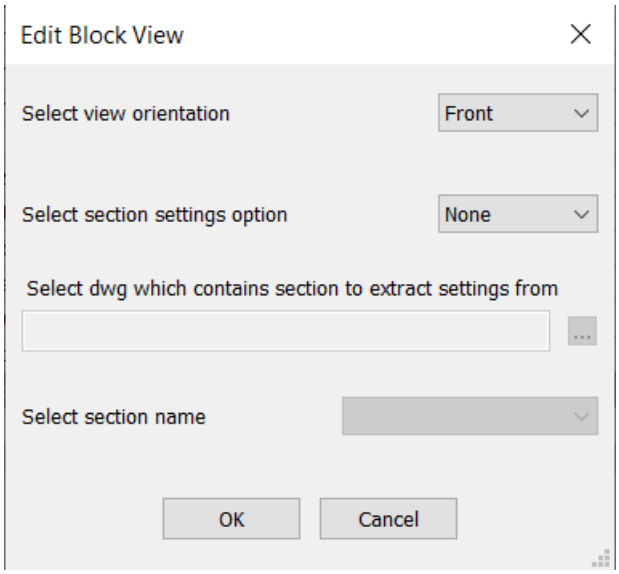
- Page 3



a Because you use an existing .dxd, the properties are already pre-selected. Views such as **Front** and **Top** are also pre-set.

b Click on the **Block view** category, the **Edit Block View** window appears.

c Choose the view.



d Click **Finish**.

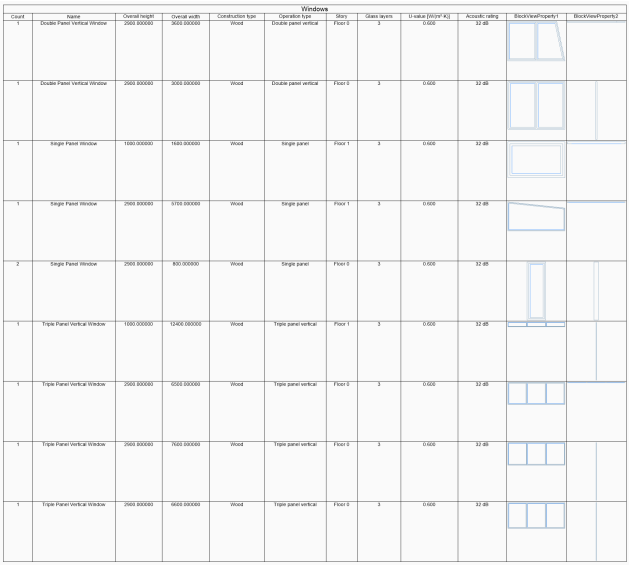
Step 3: open the schedule

1 Right-click on the new schedule.

2 In the context menu select **Create/Update Table**.

- Double click on the schedule.
- Right-click on the schedule and select **Display Schedule**.
- Go to **Sheets** and double-click on the schedule sheet. Open the schedule.

3 The result is as follows:



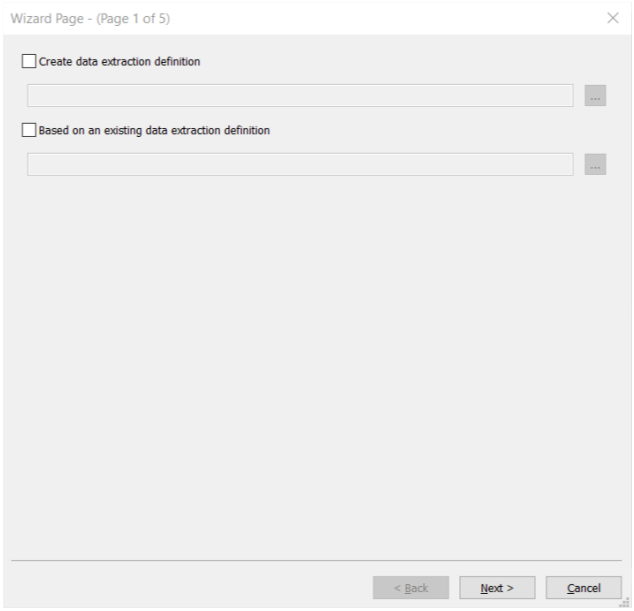
20.7.15.5 Procedure: how to create and edit your own data extraction file: furnishing elements

You can also create a Data Extraction File by yourself and use it to make new schedules. The DATAEXTRACTION command will, just like the Schedules wizard, make a new .dxd file with your settings, but it will also make a .csv file with your data in it.

1 Open your model file.

2 In the **Structure Browser**, select all Furnishing Elements.

3 Type in DATAEXTRACTION in the Command line. A wizard of 5 pages appears:

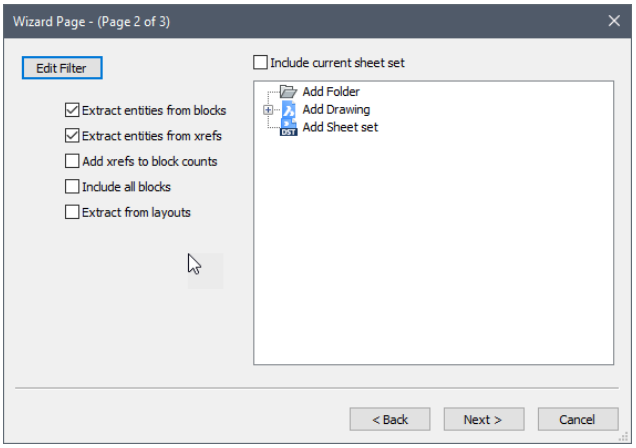


4 Check the **Create data extraction definition** box and choose a file. Name it “FurnishingElements.dxd” and save it in your project folder.

Nota: You can base you dxd file on an existing Data Extraction Definition file by checking **Based on an existing data extraction definition**. And choose the file you want to base on.

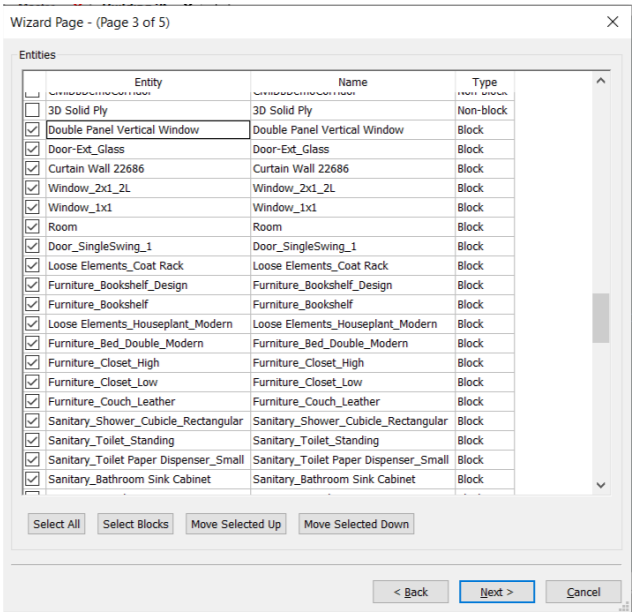
5 Click **Next**.

6 Select all entities you want to include in your schedule.



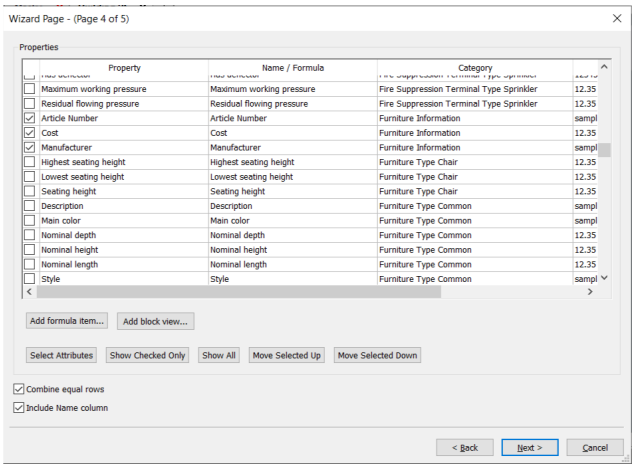
7 Click **Next**.

8 Select all entities you want to have a data extraction of.

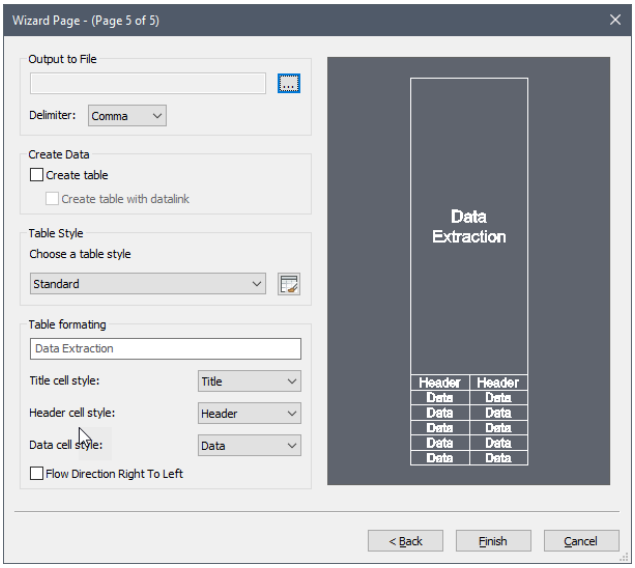


9 Click **Next**.

10 Check the all properties you want in the **Furniture Information**.



11 Click **Next**. The next page appears:



12 Click on the three dots next to the **Output to File**. Give in a new name for your .csv file and click Save. You can choose between different options for the delimiter, but it's best to use a semicolon.

13 You can create a Data table as well. When checking this, you will have the option to place the table any where inside your drawing either model space or paper space.

14 Click **Finish**.

15 This should have created 2 files: a .csv file containing the results of the data extraction (basically the same as a schedule, but in an external file) and a .dxd file. This .dxd file can now be re-used for creating a schedule. (See procedures above).

16 Open the .dxd file in a text editor. We can see 5 sections: **Settings**, **Entity Types**, **Properties**, **TableFormatOptions** and **SELECTION SET**.

17 Leave the **Settings** as they are.

18 The **Entity Types** are chosen because of the selection set you had active when using data extraction.

However, instead of explicitly choosing which types (and handles, see later) you want to extract data from, you can replace the explicit entity types by an asterisk (*), to define that we want to extract information from all entity types.

19 Leave the **Properties** as they are.

20 Leave the **TableFormatOptions** as they are.

21 You can delete the entire **SELECTION SET** section, if you don't want to explicitly define which entities you'll be extracting data from.

22 You can add a filter by adding a new section: The section title is called [Filter] The section content is (Type == Furnishing Element) (in the .dxd files that are given as example in the support folder, a Filter on Door and Window type is used).

23 Thus, the contents of the .dxd file should be as follows:

24 *[Settings]*

ExtractFromBlocks=0

ExtractFromXrefs=1

CountXrefs=0

CombineEqualRows=1

IncludeNameColumn=1

FooterStyle=0

FlowDirectionRtoL=0

CVSdelimiter=1

IncludeLayouts=0

[Entity Types]

*

[Properties]

Furniture Information:Article Number|Article Number|

Furniture Information:Cost|Cost|%1u2%pr2

Furniture Information:Manufacturer|Manufacturer

[TableFormatOptions]

TableStyle=Standard

TitleCellStyle-TITLE

HeaderCellStyle-HEADER

DataCellStyle-DATA

[Filter]

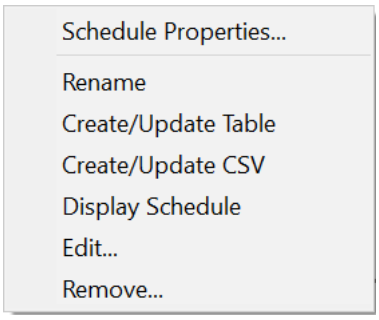
(Type==Furnishing Element)

25 Save this .dxd file in your project folder.

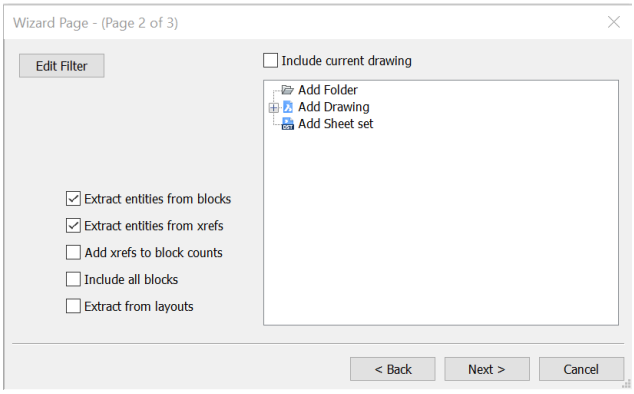
20.7.15.6 Procedure: how to edit a schedule

1 Go to the **Schedules** tab in the **Project Browser**.

2 Right-click the selected schedule. A context menu appears:



3 Click **Edit...** A Wizard of 2 pages appears:



4 Check or uncheck the options you want.

5 Click **Next**.

6 Check all properties you want to be in the schedule or uncheck all properties you want to leave out.

7 Click **Finish**.

8 Right-click on the schedule again and choose **Create/Update Table**.

9 Repeat this action but choose **Create/Update CSV**.

10 The table and CSV file are now updated. You can update and open your sheet with the schedule again and see the changes.

20.7.16 Spot elevation level

20.7.16.1 Command

BIMTAG

20.7.16.2 About

The Spot Elevation tag displays the absolute elevation of a selected point of a BIM object displayed in a section view, based on the **World Coordinate System (WCS)**.

As the Spot Elevation tag is an associative tag type, it can only be used in the **paper space** of a drawing sheet.

For more information on how to set up Sheets and create Projects, please visit the **Project Browser** article.

20.7.16.3 Procedure: how to place a spot elevation tag on a section sheet

1 Open a section sheet of the model. Right-click on the section in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line **or** click on **tag** in the **Draw** menu of the Quad.

3 **You are prompted**: [Manual/selection options (?)]

Type **M**t o choose **Manual** and press Enter.

4 **You are prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

5 **You are prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

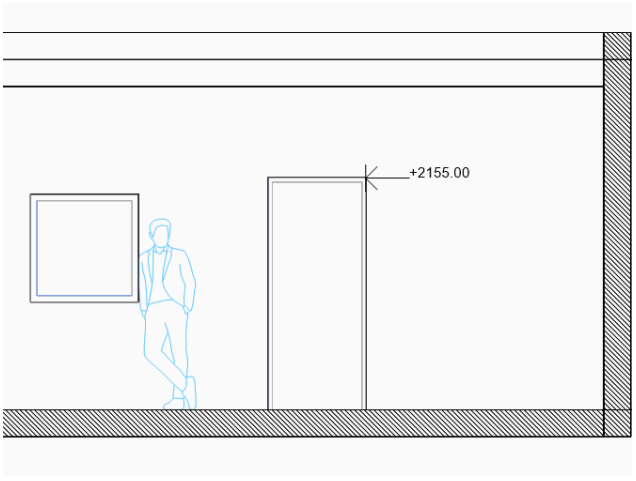
Type **S**to choose **Spot-elevation** and press Enter.

6 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the elevation level measured relative to level 0 of the model.

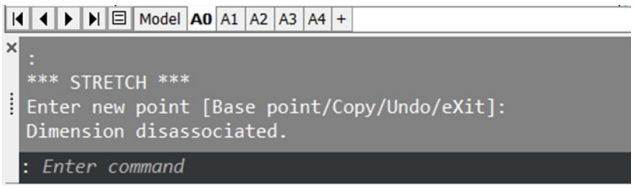
Nota: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

7 When there are several tags possible on the location where you place the tag, hit CTRL to scroll through the different options while the Hotkey Assistant (HKA) appears at the bottom of the screen.

8 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

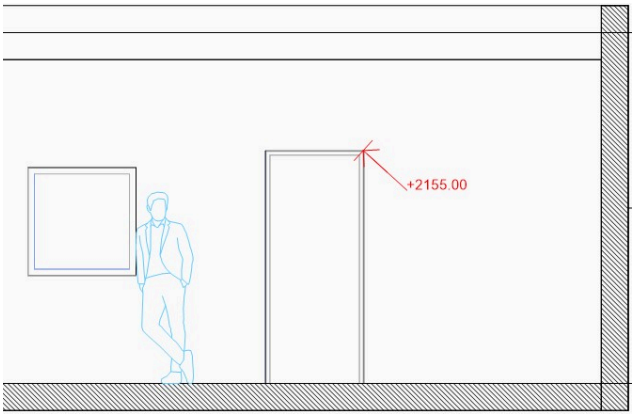


Nota: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value becomes dissociated in the Command line.

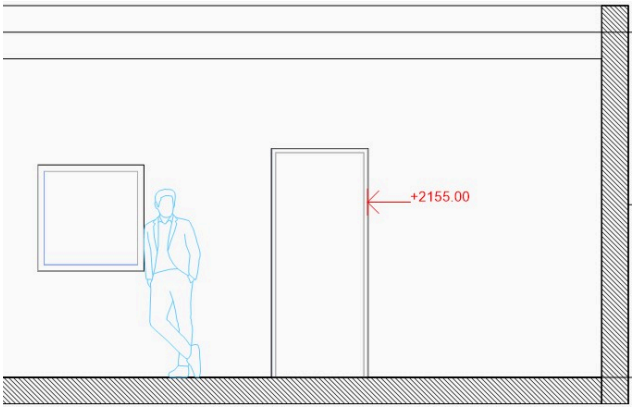


9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

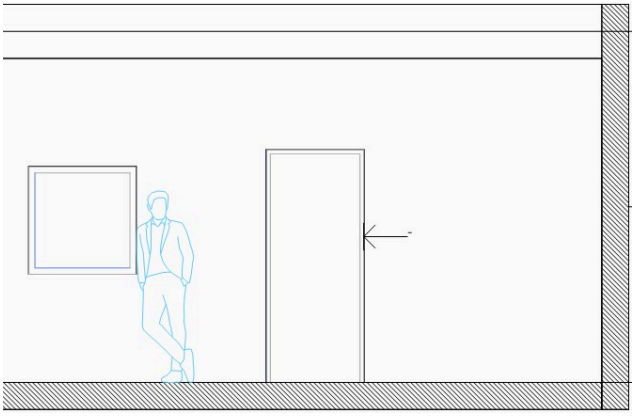
Nota:If the tag has been displaced, it may behave irrationally or haphazardly by either turning red or omitting its value, as illustrated below:



Tag turned red after displacing the arrowhead of the leader and updating the section.



Tag turned red after displacing the entire tag and updating the section.



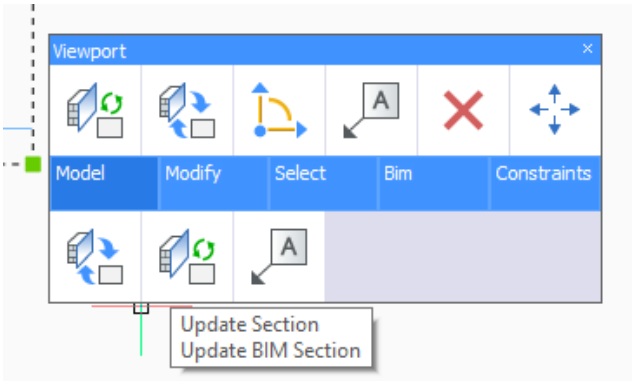
Tag value disappeared after displacing the tag and updating the section.

20.7.16.4 Procedure: how to place a spot elevation tag on an elevation sheet

1 Open an elevation sheet of the model. Right-click on the elevation sheet in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line,

3 **or** click on **Tag** in the **Model** menu of the Quad.



4 **You will be prompted**: [Manual/selection options (?)]

Type **M** to choose **Manual** and press Enter.

5 **Later, you will be prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

6 **Finally, you will be prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

Type **S** to choose **Spot-elevation** and press Enter.

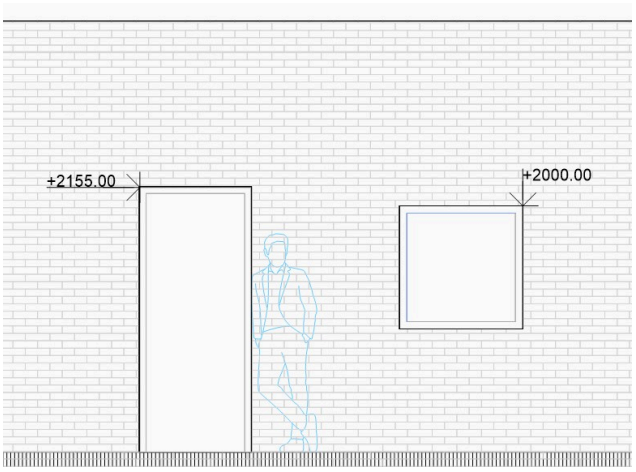
7 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the absolute elevation level measured relative to Floor 0 of the model.

Nota: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

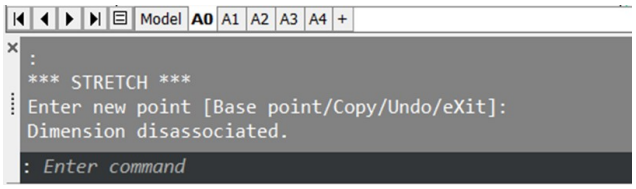
8 When there are several tags possible on the location where you place the tag, hit CTRL to scroll

through the different options while the HotKey Assistant (HKA) appears at the bottom of the screen.

9 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

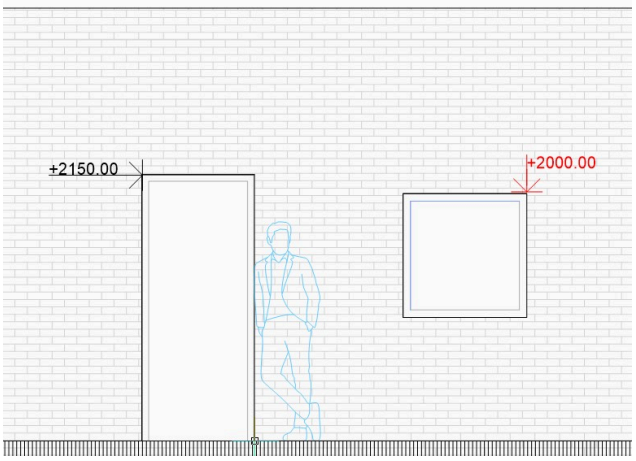


Nota: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value is dissociated in the Command line.

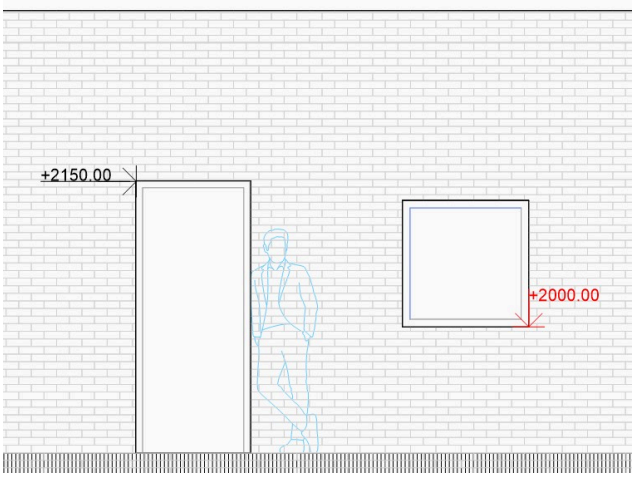


10 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

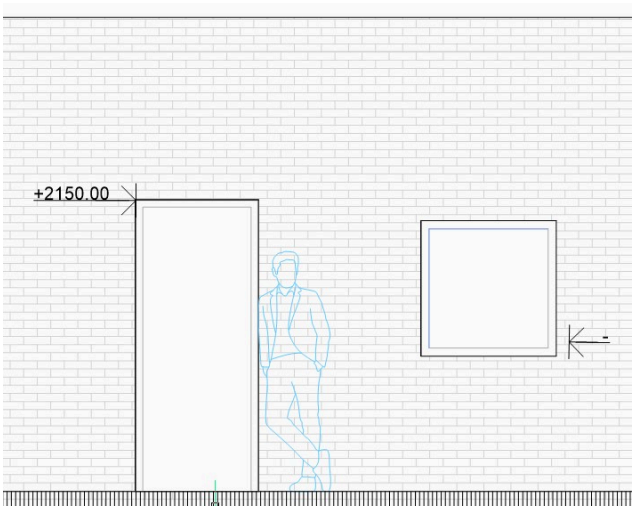
The tag should not behave irrationally or haphazardly by either turning red or omitting its value when it's displaced.



Tag turned red after displacing the arrowhead of the leader and updating the section.



Tag turned red after displacing the entire tag and updating the section.



Tag value disappeared after displacing the tag and updating the section.

20.7.16.5 Procedure: placing spot elevation level tags on floor plans or reflected ceiling plans

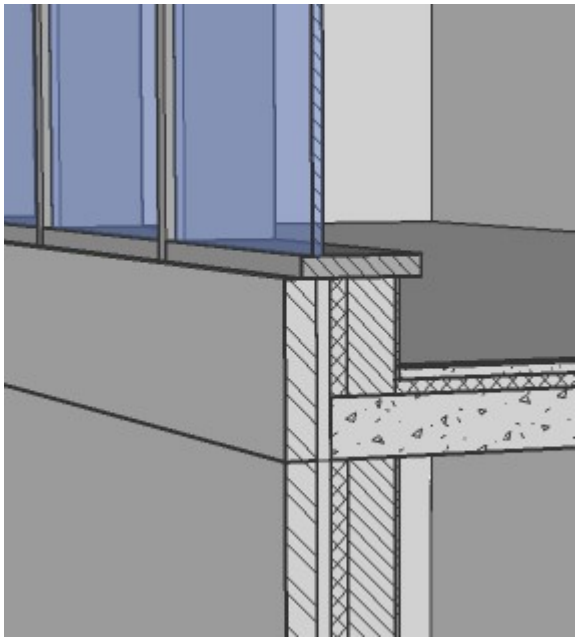
1 Open a floor plan or a reflected ceiling plan from the **Project Browser**.

2 Go through step 2-7 of 'How to place spot elevation level tag in a section sheet'.

3 In a floor or ceiling plan, there are 2 different tags possible.

- **With MLeader**

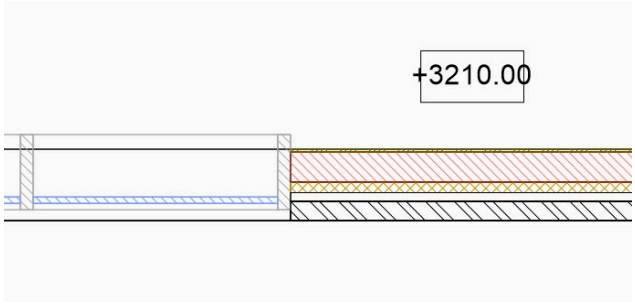
  This annotation tag type, which illustrates with a leader line, is also normally used in the section or elevation sheets. You can use this tag on entities that do not reach or aligned to the floor/ceiling level, such as a window sill in the picture below.

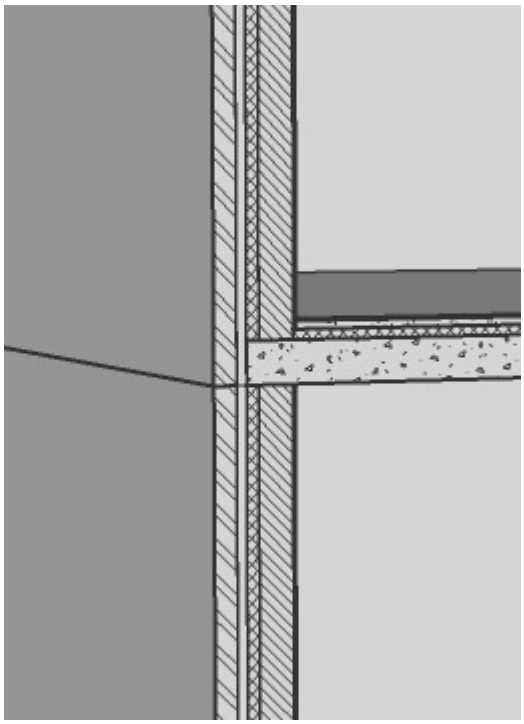


Spot elevation level tag on window sill level which is not on floor level.

- **Without MLeader**

  This tag has a different layout to the tags found in the elevation or section sheets. This tag appears when you place a spot elevation tag on entities that reaches the floor/ceiling plane, thereby displaying the elevation level of the floor plane.



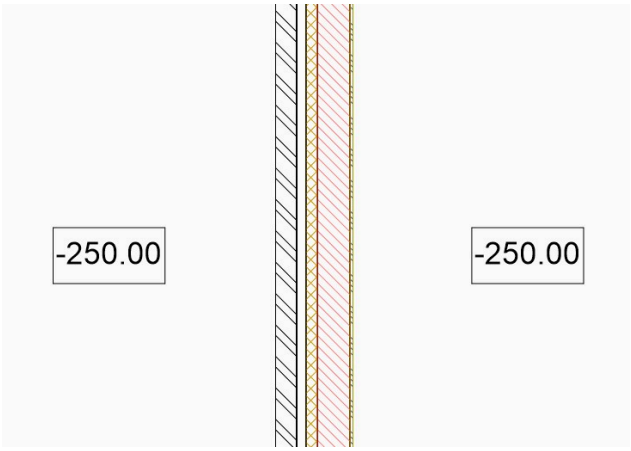


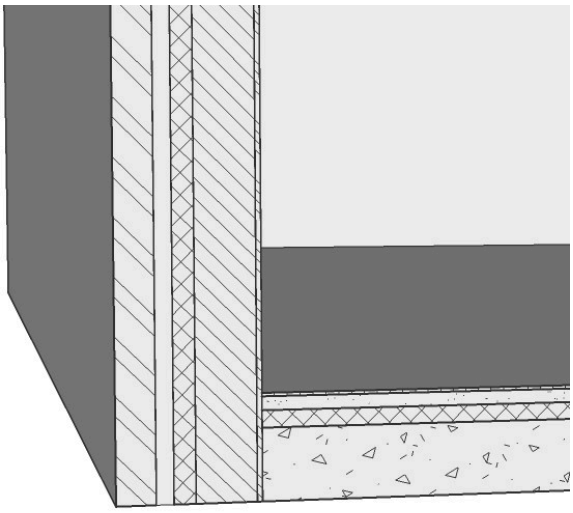
Spot elevation level tag on wall that reaches till the floor slab.

4 When placing a tag on a wall in plan view, you may encounter different results for different plies of the wall. This is due to how elevation levels are measured In BricsCAD. When placing a spot elevation tag on an entity in plan view, it refers to the elevation value of the lower level of the entity's ply, if applicable. Similarly, when you place a tag on a ceiling plan, the upper level of the ply is measured. Since plies of a wall may differ in their lengths based on how they are designed at detail junctions, the spot elevation tag value may differ according to the ply you have selected.

- **Construction detail with unedited plies**

  The interior and exterior plies have the same length. All the spot elevation tags you place on this wall in plan view will therefore display the same value.

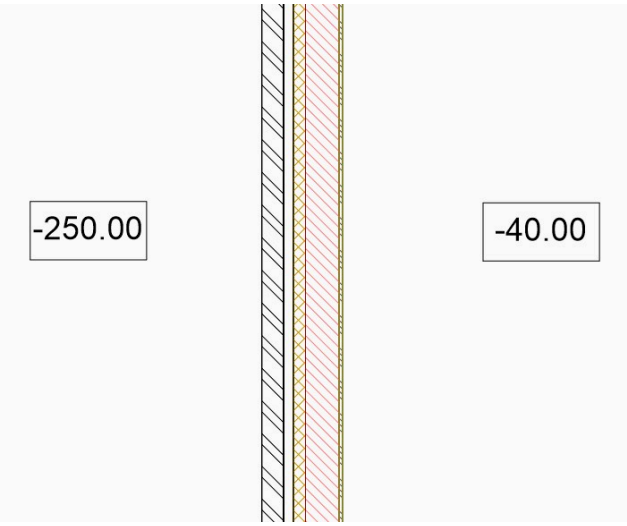


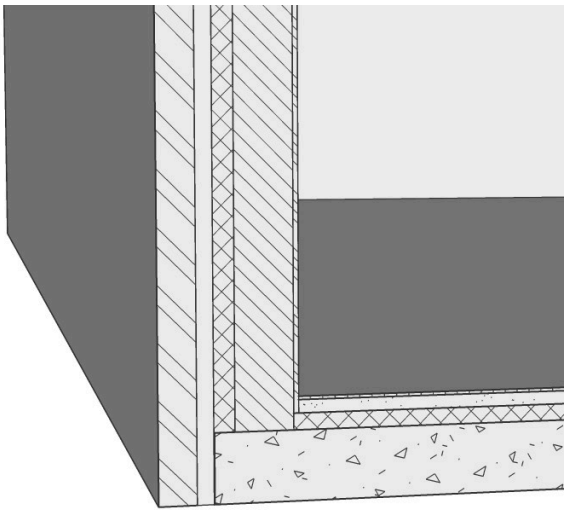


Nota: The value is -250 instead of 0 because Floor 0 is defined as the level of the upper side of the ground floor in this model.

- **Edited construction detail**

  If you have adjusted how the wall would connect to a floor slab based how it would be constructed, the plies of the wall will therefore have different lengths. Depending on which ply you select to place the spot elevation level tag, a different value will be displayed.

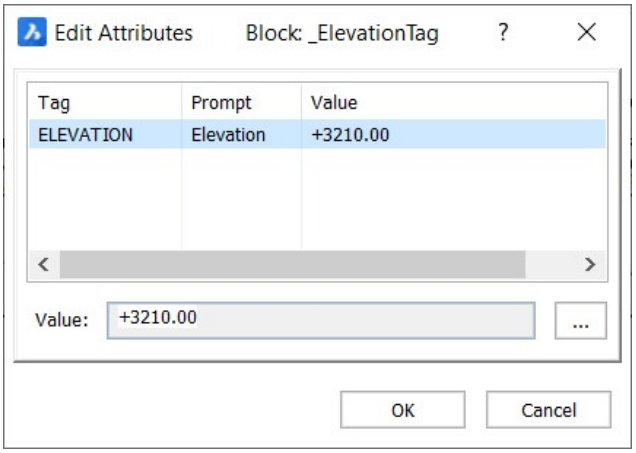




Elevation level of interior ply is higher because ply does not reach till the base level of the floor.

20.7.16.6 Procedure: adjusting the layout of the tag

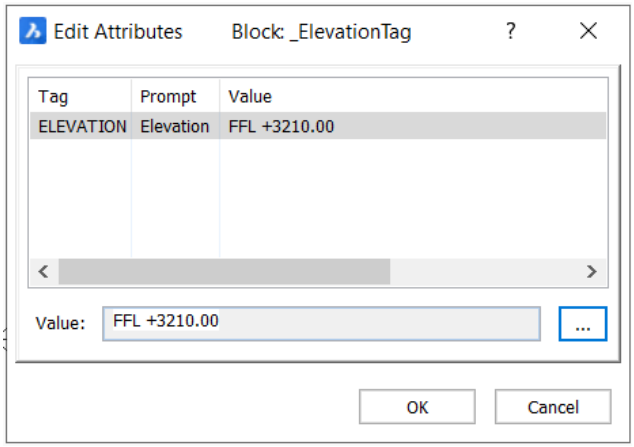
1 Double-click on the value of the spot elevation tag to adjust the layout. A dialog box will pop-up where you can edit the attributes.

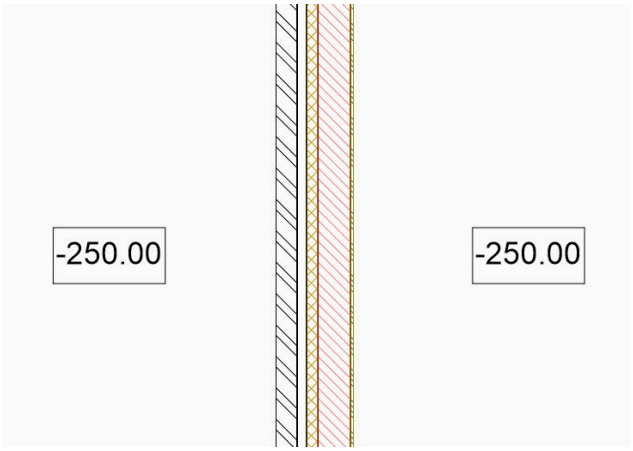


2 Click on the three dots to open the **Text Formatting** dialog box.



3 Here you can edit the attributes of the displayed value of the spot elevation level tag. You can put prefixes and/or suffixes to the values, e.g. FFL, SSL, to better illustrate the levels you are referring to.





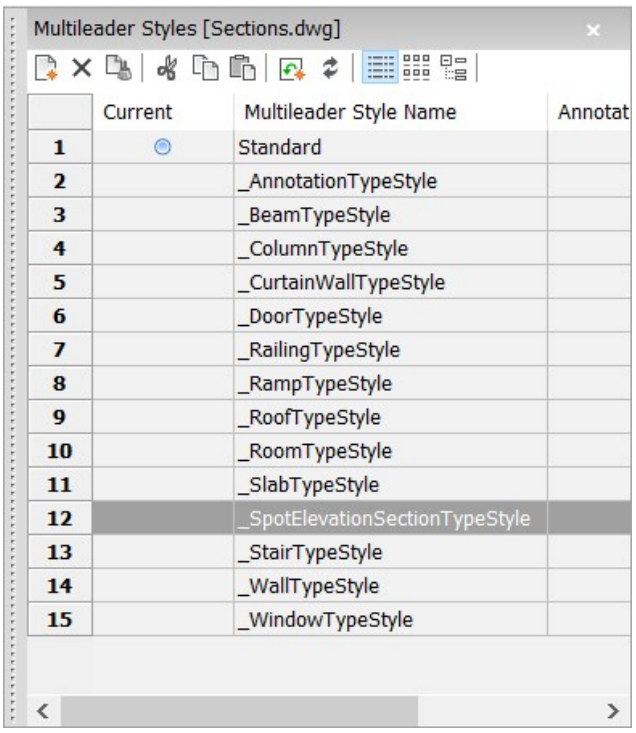
4 If the prefixes and/or suffixes are commonly used, you can create or edit the tag blocks so you do not have to repeat Step 3. For more info, visit the command article BEDIT.

5 For a detailed explanation about attributes, visit the command article MText.

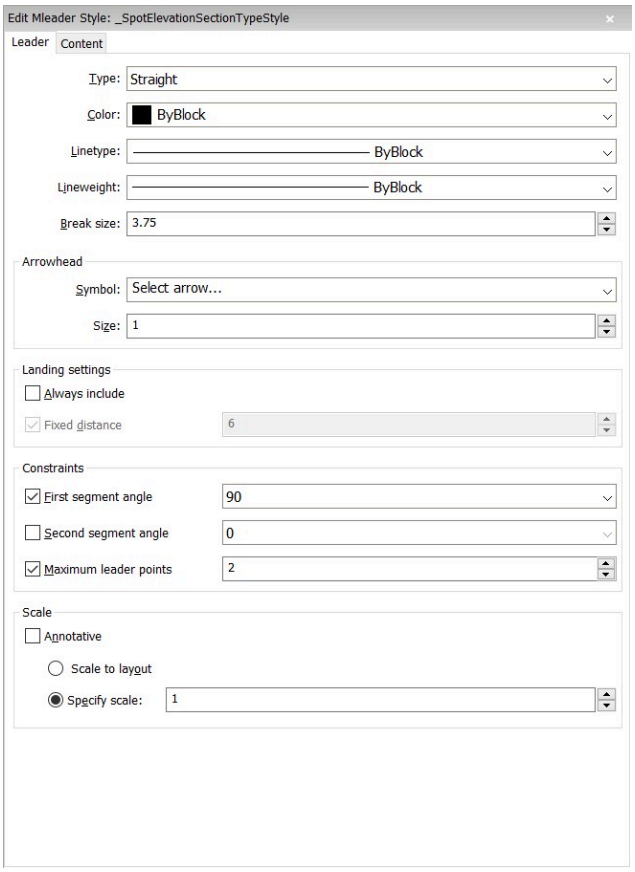
20.7.16.7 Procedure: adjusting the layout of the leader

1 Type MLEADERSTYLE in the Command line and press Enter. The **Drawing Explorer** dialog box will pop-up where you can adjust the layout of the leader.

2 Click on **SpotElevationSectionTypeStyle** in the **Multileader Styles** dialog box.

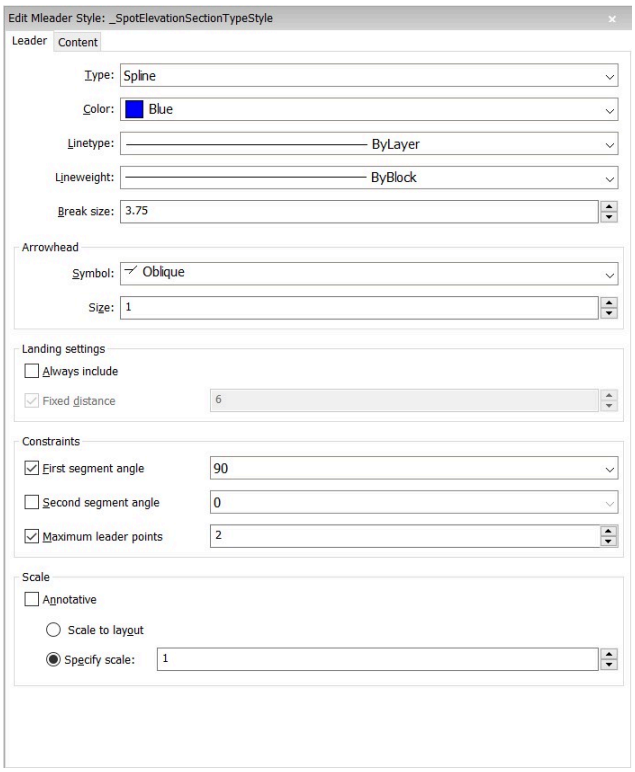


3 You can edit the settings of the leader in the **Edit MLeader Style** dialog box. For more info about the MLeader style settings, visit the article MLeaderStyle.

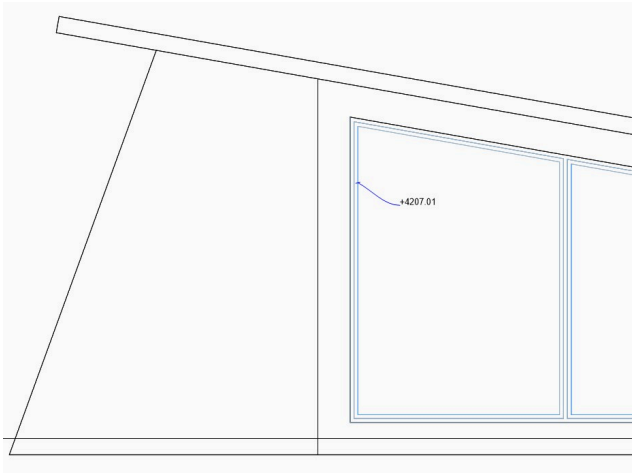


4 After adjusting the settings, click on regen, close the **Drawing Explorer** and update the section to load the new settings.

Nota: If the new settings don't load, type BIMTAG in the Command line, select the viewport, click **R** to select **Regenerate All** and press Enter to load the new settings.



New Settings



Resulting BimTag

20.7.17 Templates for generated drawings

20.7.17.1 About

BricsCAD BIM has a series of templates to customize your generated drawings. These templates are stored in your **Support** folder in the following path:

C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\Bim\Sections.

The **Appdata** folder is a hidden folder by default, you may have to unhide folders first. This path can also be accessed by the SUPPORTFOLDER command.

To understand how it works, we must know how BricsCAD creates layers, hatches, tags, and call-out blocks. Then, for each template, we will discuss how to customize them.

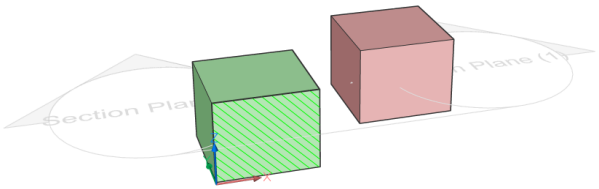
20.7.17.2 Procedure: understanding the created layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

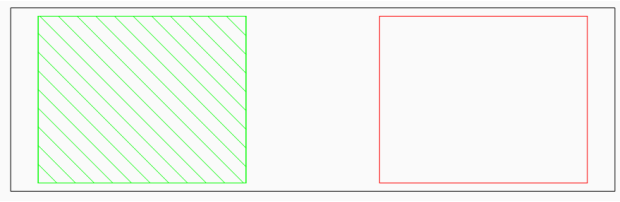
2 Draw two boxes of 500x500x400, one being a bit behind the other.

3 Put the first box on the layer **Layer1** (with layer color green) and the second box on the layer **Layer2** (with layer color red).

4 Create a BIM section, aligned with the front face of the first, green box and cutting through it in the middle, but not cutting through the second, red box.



5 Generate and open the section drawing.



6 Take a look at the **Layers** panel: three groups of layers have been created: **Background**, **Boundary**, and **Fill**.

	Current	Name	D..	On/Off	Freeze	Locked	Color
1		0					White
2		Background_Layer1					Green
3		Background_Layer2					Red
4		BIM_SECTIONS_PREVIOUS					Magenta
5		BIM_VIEWPORTS					White
6		Boundary_Layer1					Green
7		Defpoints					White
8		Fill_Layer1					Green
9		Title Block Frame					175
10		Title Block Labels					157
11		Title Block Text					251

7 The generated layers for the green solid are:

- **Boundary_Layer1**: the boundary of the section through the box.
- **Fill_Layer1**: the hatching of the section.
- **Background_Layer1**: shows subentities (e.g., vertices of solids) that are not sectioned.

8 The layers all inherit the color of the **Layer1** layer.

9 Since the red solid is not sectioned, only the **Background_Layer2** layer is created. It inherits the color of the **Layer2** layer.

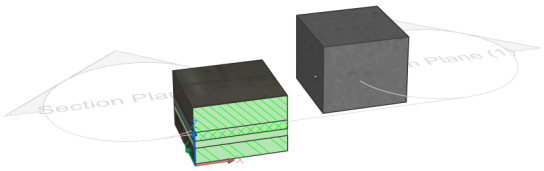
10 Drag the **Cavity Wall, Brick** composition from the **Compositions** panel onto the green box.

Nota: Set the LEVELOFDETAIL system variable to 1 or 2, to display the ply materials.

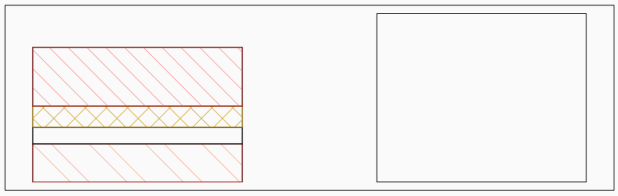
11 Double-click the **Cavity Wall, Brick** composition in the **Compositions** panel.

This composition consists of 4 plies, each with its own material. A physical material is associated with a hatch pattern, used for floor plans and sections. Another pattern is specified for elevations.

12 Assign the **Concrete, Reinforced** composition to the red box.



13 Update and open the section.



14 Go to your layers and execute the PURGE command to remove the unused layers.

	Current	Name	D..	On/Off	Freeze	Locked	Color
1		0					White
2		Background_Air					White
3		Background_Concrete_ Reinforced					White
4		Background_Facing Bricks_ Hand-form					White
5		Background_Insulation_ Polyurethane					White
6		Background_Layer1					Green
7		Background_Layer2					Red
8		Background_Supporting Wall_ Brick					White
9		BIM_SECTIONS_PREVIOUS					Magenta
10		BIM_VIEWPORTS					White
11		Boundary_Air					White
12		Boundary_Cavity Wall_ Brick					White
13		Boundary_Facing Bricks_ Hand-formed					14
14		Boundary_Insulation_ Polyurethane Fo					42
15		Boundary_Layer1					Green
16		Boundary_Supporting Wall_ Brick					14
17		Fill_Facing Bricks_ Hand-formed					21
18		Fill_Insulation_ Polyurethane Foam					42
19		Fill_Layer1					Green
20		Fill_Supporting Wall_ Brick					11
21		Title Block Frame					175
22		Title Block Labels					157
23		Title Block Text					251

You see the three categories (**Background**, **Boundary**, and **Fill**) again. Now the second part of the layer name is the name of the physical material used in the solid, instead of the layer name of the box:

- **Background_Facing Bricks_Hand-formed**, **Background_Insulation_Polyurethane Foam**, and **Background_Supporting Wall_Brick**.
- **Boundary_Facing Bricks_Hand-formed**, **Boundary_Insulation_Polyurethane Foam**, and **Boundary_Supporting Wall_Brick**.
- **Fill_Facing Bricks_Hand-formed**, **Fill_Insulation_Polyurethane Foam**, and **Fill_Supporting Wall_Brick**.

15 Set the **Generate elevation fill** property of the section **ON** to add an elevation hatch.

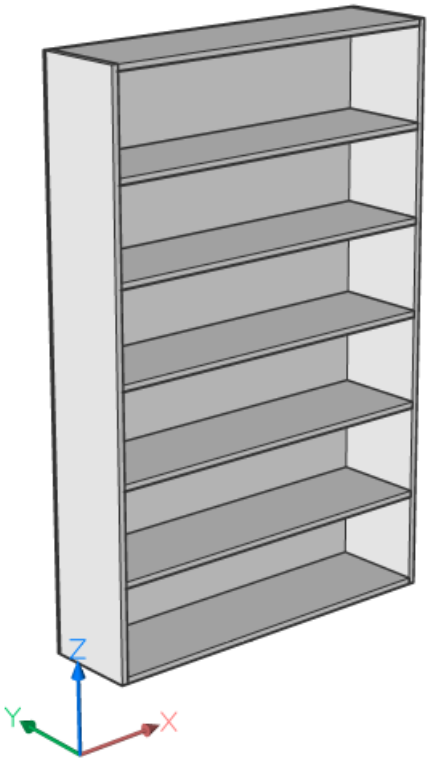
16 Update the section.

17 The outline of the second box is now hatched and the layer **Elevation_Concrete_Reinforced** is added.

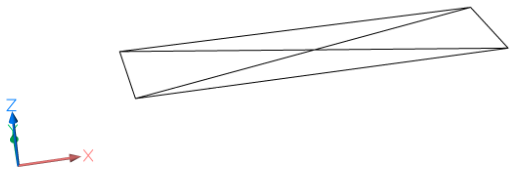
20.7.17.3 Procedure: understanding the BRX_2D layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

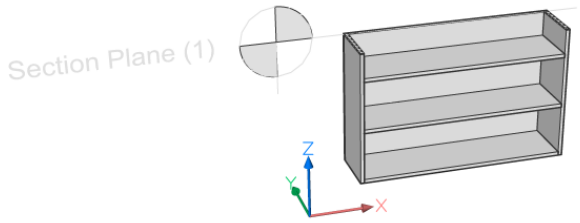
2 Drag the component **Furniture Bookshelf** into the drawing. You can find the component in the **Furnishing Elements** category in the **Library** panel (LIBRARYPANELOPEN command).



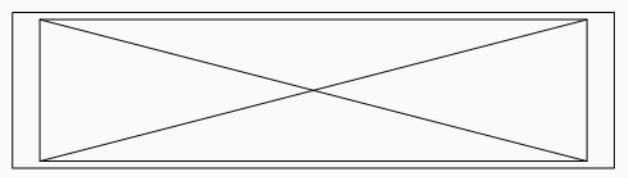
3 If you now show and isolate the **BRX_2D_Symbols** layer, you will see a symbol drawn at the insertion point of the bookshelf.



4 Now show all layers again (LAYUNISO command) and make a section through the bookshelf.



5 Generate and open the Plans drawing.



6 You will see (if the **Boundary_BRX_2D_Symbols** layer is on) that only the symbol is drawn onto the plan, and not the real section. It happens because this component had a 2D symbol on the BRX_2D layer attached to it.

7 Now return to your 3D model and Open a copy of the bookshelf.

8 Hide the bookshelf itself and make sure the **BRX_2D_Symbols** layer is on.

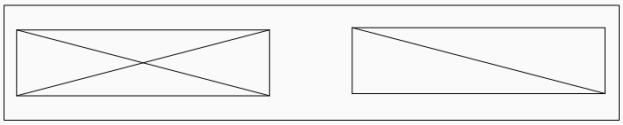
9 Delete one of the lines (of the cross).

10 **(Optional)** Draw extra 2D lines on the **BRX_2D_Symbols** layer.

11 Save the drawing.

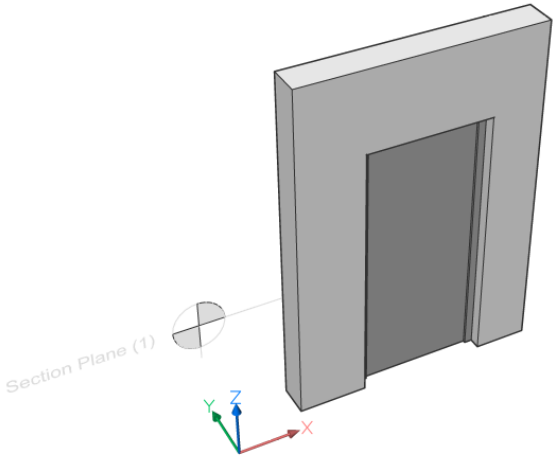
12 Go back to the original 3D model and type in BMINSERT in the Command line and go to the file path where you just saved the new bookshelf. Click **OK** and choose a point to insert the bookshelf.

13 Regenerate the section and open it. Widen the viewport if necessary, by dragging the existing viewport.



14 You will now see that the second bookshelf has the adapted symbol displayed.

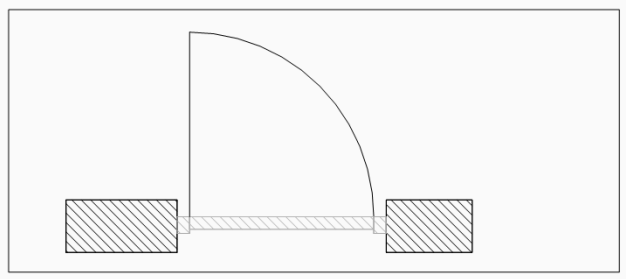
15 Draw a wall in your 3D model and insert the door **Door Ext Plain** into this wall.



16 Repeat steps 3 to 5 but now use the **BRX_2D+_Swing** layer instead of the **BRX_2D_Symbols** layer.

17 The generated plan displays the symbol of the door swing as well as the door section itself. The **BRX_2D+** layer of a component makes sure you can show the real section together with a symbol.

- 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.



20.7.17.4 Procedure: customizing the colors and line weights of hatch patterns and lines

1 Firstly, in the above procedure 'Understanding the created layers', you saw that for objects without a composition, the colors and line weights of the **Fill_**, **Background_**, and **Boundary_** layers are copied from the layer properties in the 3D model.

2 However, for objects with a composition, the default colors are set by a template found by typing in SUPPORTFOLDER command and browsing to *Bim/Sections/_SectionSettings.dwg*.

3 Open this drawing.

4 Go to the **Layers** panel, you will find a list of all the possible layers that can be created for every material present in the material library. They all have a color and line weight. Changing the color or line weight and saving the file will change the default color/line weight for this layer in your future generated drawings. You can fully customize how your drawing will look for every type of line (**Fill**,

Background, **Boundary**, or **Elevation**) per material.

195		Fill_Ceiling_ Plasterboard				253
196		Fill_Concrete_ Beam and Pot Floor				85
197		Fill_Concrete_ Plain				85
198		Fill_Concrete_ Precast				85
199		Fill_Concrete_ Reinforced				85
200		Fill_Copper				151
201		Fill_Facing Bricks_ Concrete Blocks				21
202		Fill_Facing Bricks_ Hand-formed				21
203		Fill_Facing Bricks_ Sand-lime				21
204		Fill_Floor Joist_ Wood				33
205		Fill_Floor_ Boarded				9
206		Fill_Floor_ Ceramic Tiles				251
207		Fill_Floor_ Concrete Tiles				251
208		Fill_Floor_ Laminate				251
209		Fill_Floor_ Parquet				251
210		Fill_Floor_ Stone Tiles				43
211		Fill_Freestone				43
212		Fill_Glass				141
213		Fill_Glass Brick				141
214		Fill_Gold				151
215		Fill_Granite				43
216		Fill_Gravel				251
217		Fill_Gypsum Board				251
218		Fill_Insulation_ Extruded Polystyrene				42
219		Fill_Insulation_ Glass Wool				42
220		Fill_Insulation_ Polyurethane Foam				42
221		Fill_Insulation_ Rock Wool				42
222		Fill_Iron				251
223		Fill_Lead				151
224		Fill_Marble				43

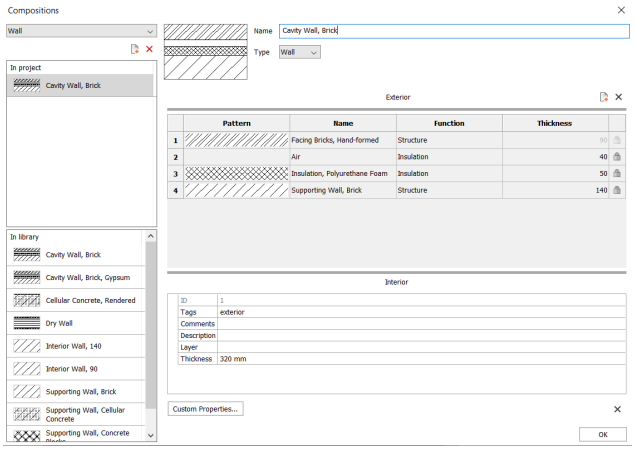
20.7.17.5 Procedure: customizing the hatch patterns

1 Open the drawing made in the above procedure 'Understanding the created layers'.

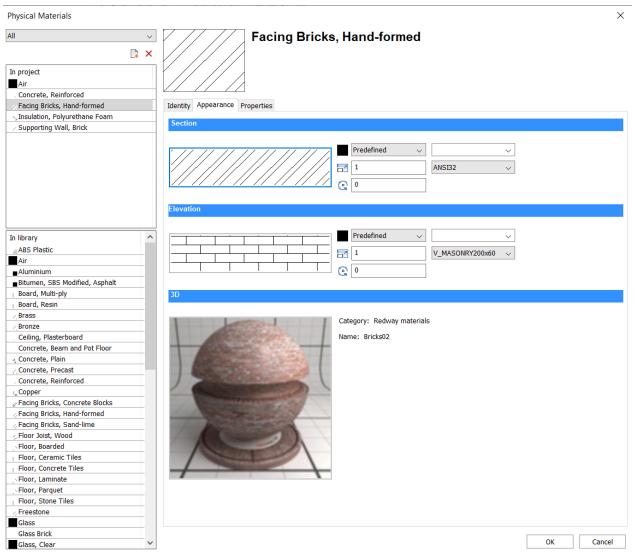
2 Open the **Compositions** panel.

3 Double-click the composition **Cavity Wall, Brick**.

4 In the procedure 'Understanding the created layers said that you can change the hatch patterns of the materials here. How you need to do this will follow in the next steps.



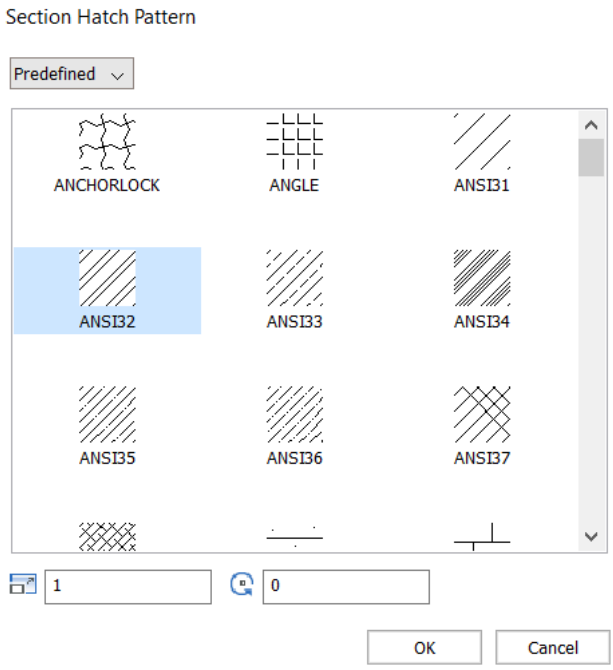
5 Double-click on the row of **Facing Bricks, Hand-formed**. The **Physical Materials** dialog will pop-up on the material **Facing Bricks, Hand-formed**. You could have entered this dialog box by typing in the command BLMATERIALS, after which you could have selected the **Facing Bricks, Hand-formed** material.



6 Go to the tab **Appearance**.

7 Under the title **Section**, double-click on the hatch pattern.

8 You can now choose your new standard section hatch representation for this particular material. In every composition with this material, the same hatch representation will be used.



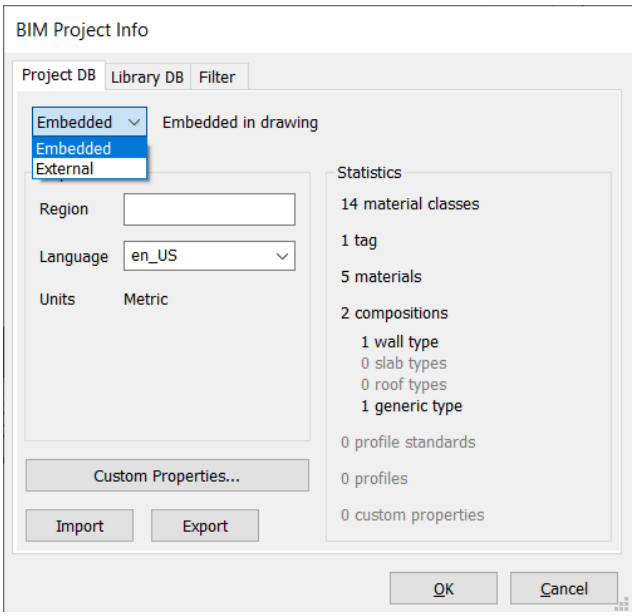
9 If you want to change the appearance of an elevation hatch, you can do so by double-clicking the hatch pattern under the **Elevation** title. The outer most visible material of a composition will be used to determine the elevation hatch.

10 Note that the representation of these hatches will not be copied into new drawings as the **Physical Materials** dialog is not a template, but rather a way to define the representation in a specific drawing.

If you would like to store this information, it is recommended to create a library of compositions. The best way to do so is to make a drawing where you attach the compositions you would like to store to little cubes. Make sure each composition has the hatch appearance you want by going through the process described above.

11 Once this is finished, type in BIMPROJECTINFO in the Command line. You will enter the **BIM Project Info** dialog box.

12 In the **Project DB** tab, switch the value for the database from **Embedded** to **External**. You will be asked to save the database on your computer and give it a name, for example, 'CompositionsTemplate.bsyslib'.



13 If you now want to use this database (Project DB) in a new drawing, switch the value for the database from **Embedded** to **External**. This time select the .bsyslib file you created.

20.7.17.6 Procedure: customizing tags per entity type for all section drawings

For this procedure, please visit the following procedures from the BIM Tags page:

- Create the source data for a BIM entity type.
- Customize tag blocks.
- Edit the source data to customize BIM Tags for all section drawings.

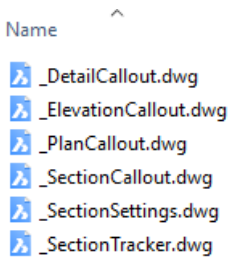
To change the tags in the current drawing, visit the procedure 'Customize tag blocks for the current drawing only'.

20.7.17.7 Procedure: customizing call-out blocks

1 Source files of the various call-out 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\Support**

\Bim\Sections. You can easily access this file by typing in SUPPORTFOLDER command and browsing to **Bim** > **Sections**.



2 You can edit these files to customize the layout of the call-outs and the section tracker blocks.

3 When a drawing is generated, the needed call-out blocks are created from the call-out blocks source file(s). When a BIM section is updated, the existing call-out blocks in the target drawing are used (type BEDIT to choose one to edit).

20.7.17.8 Procedure: customizing the sheet set template

1 The sheet set templates are stored in the **C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD \Vxx\en_US\Templates** folder.

2 If you want to edit an existing template, you have two options. Either you edit it directly, or you make a copy.
   - To edit it directly, you can go to **Open** under the BricsCAD icon, choose **.dwt** in the **Files of type** drop-down list and choose the file you want to edit. You can then make the modifications and save the file.
   - To make a copy, double-click on the .dwt file you want to copy. A new .dwg with this template will open. Make your modifications and **save your file as .dwt**. Save it in the templates folder to easily use the copied template in a new drawing.

3 If you want to start from scratch, using a wizard, follow the procedure 'To create a drawing using a wizard'. After the creation and modification of the dwg, save it as a .dwt file in your sheet set templates folder.

4 If you have saved the template .dwt to the sheet set templates folder, you will see it appear when you use the **New Wizard** to open a new drawing. You will be able to select it as the template for the creation of generated drawings in the sheet set set-up of the **Project Browser**.

5 What can you store into a template?

6 You can make a title block for your template, with sheet set properties, by following the procedure 'Using sheet set properties in a title block'. Do this for every layout in your template. You can also store the following things by using them in your .dwt file:
   - Layers
   - Colors
   - Line types
   - Hatch patterns

- Blocks
- Text styles
- Dimension styles
- Page set-ups

21. BricsCAD Mechanical

21.1 O Que há de novo

21.1.1 Visão Geral

O BricsCAD Mechanical V21 oferece todas as ferramentas novas e aprimoradas disponíveis no BricsCAD V21, além de aprimoramentos adicionais especificamente para fluxos de trabalho de mecânica.

Esta seção oferece uma visão geral de todos os novos recursos e melhorias nas **ferramentas do BricsCAD V21 Mechanical**:

Gerenciador de BOM na página 1533

Bibliotecas de peças na página 1536

Parâmetros e Restrições na página 1537

Peças padrão na página 1537

Chapa metálica na página 1537

Para obter informações sobre os novos recursos e melhorias nas **ferramentas do Núcleo e de Civil, no BricsCAD V21**, vá para a seção Novidades do BricsCAD V21. Esta contém os seguintes artigos:

Overview na página 31

User interface na página 32

Productivity na página 35

Drafting na página 49

Modeling na página 51

Civil tools na página 60

Performance and compatibility na página 65

Para obter informações sobre os novos recursos e melhorias no **BricsCAD V21 BIM**, vá para a seção Novidades no BricsCAD V21 BIM. Esta contém os seguintes artigos:

Overview na página 961

Modeling techniques na página 962

Building data na página 969

Project collaboration na página 978

Design documentation na página 979

Point clouds na página 988

21.1.1.1 Notas de Release

Clique aqui para obter uma visão geral das notas de release.

21.1.1.2 Diagnóstico e Coleta de Dados de Uso

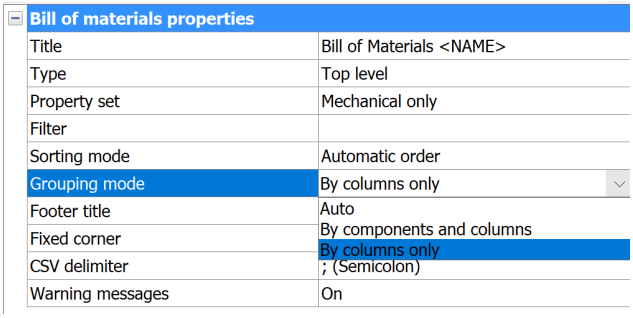
Muitas das melhorias foram inspiradas por solicitações e sugestões dos usuários BricsCAD. Na Bricsys, continuamos a expandir nossos métodos de coleta de feedback dos usuários, com novas ferramentas para coleta de dados. Quando você inicia o BricsCAD pela segunda vez, uma caixa de diálogo é exibida, convidando você a participar do Programa de Diagnóstico e Coleta de Dados de Uso. O processo de coleta de dados permite a você compartilhar dados anônimos de diagnóstico e dados de uso com a Bricsys, para ajudar a melhorar o BricsCAD. A qualquer momento no futuro você pode retirar seu consentimento, e ter seus dados removidos de nossos bancos de dados.

21.1.2 Gerenciador de BOM

Criar e editar listas de materiais está mais fácil do que nunca.

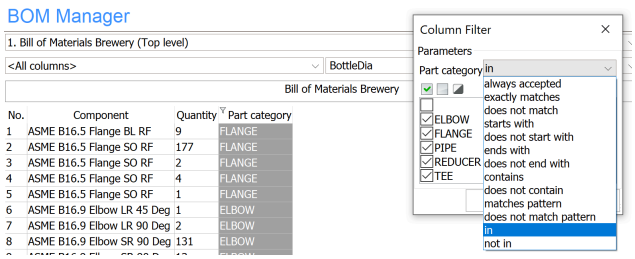
21.1.2.1 Agrupar

Com a adição da propriedade de agrupamento, agora você pode agrupar rapidamente o conteúdo da sua tabela BOM em linhas condensadas por propriedades comuns das colunas existentes.

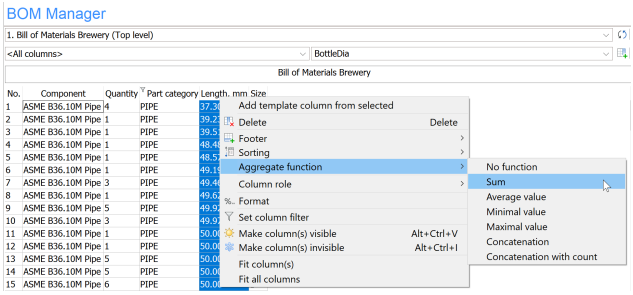


21.1.2.2 Colunas

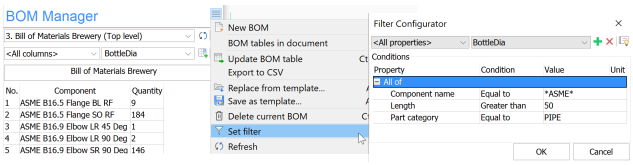
Agora é possível definir um dos vários filtros de coluna predefinidos, para qualquer coluna em uma BOM.



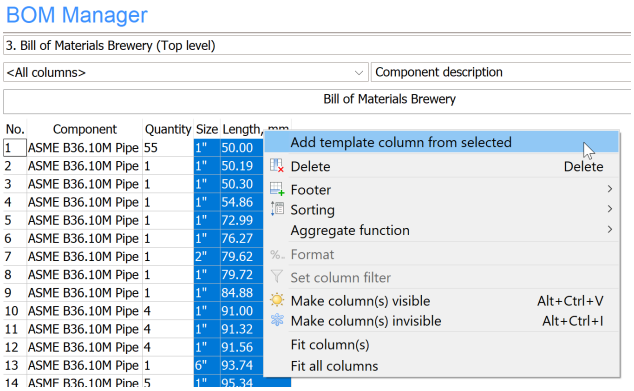
Colunas agregadas agora são suportadas em tabelas BOM de nível superior (top-level) e inferior (bottom-level) criadas com o comando BMBOM. Essas permitem calcular soma, valor mínimo, máximo ou da média de uma determinada propriedade, em relação ao agrupamento atual.



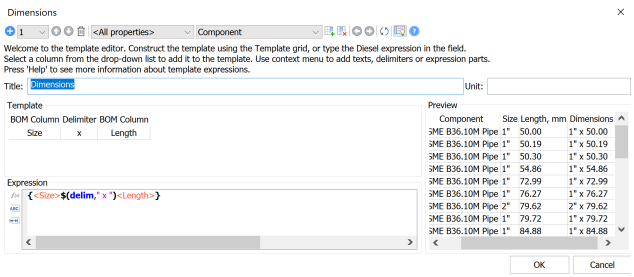
Uma opção no menu principal do Gerenciador de BOM abre a nova caixa de diálogo Configurator de Filtros. No Configurator de Filtros, você pode filtrar o conteúdo da tabela BOM; e expressões curinga são suportadas para propriedades de frases (strings) em um filtro BOM.



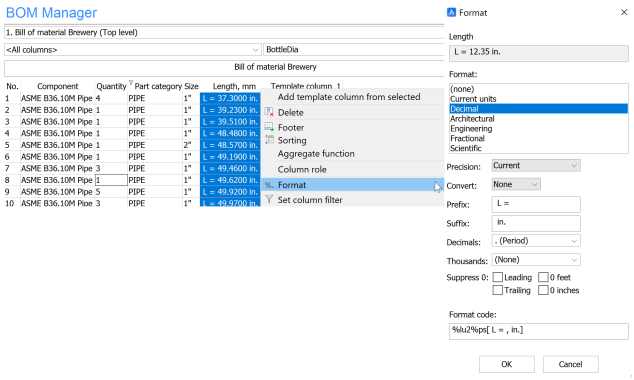
Colunas Template são um novo tipo de coluna avaliada, que permite combinar valores de várias propriedades em uma única coluna BOM, como texto, usando expressões DIESEL.



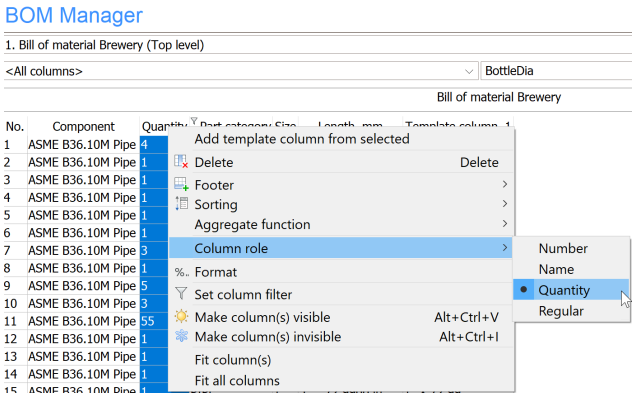
Você pode configurar e ter uma prévia da fórmula ou coluna de template usando a caixa de diálogo do editor de expressões.



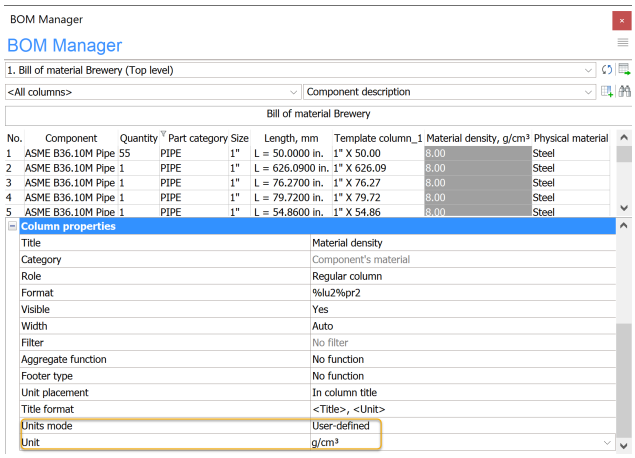
Com a adição da caixa de diálogo Formatar, agora você pode personalizar a expressão de seus dados em tabelas BOM com prefixo, sufixo e delimitadores.



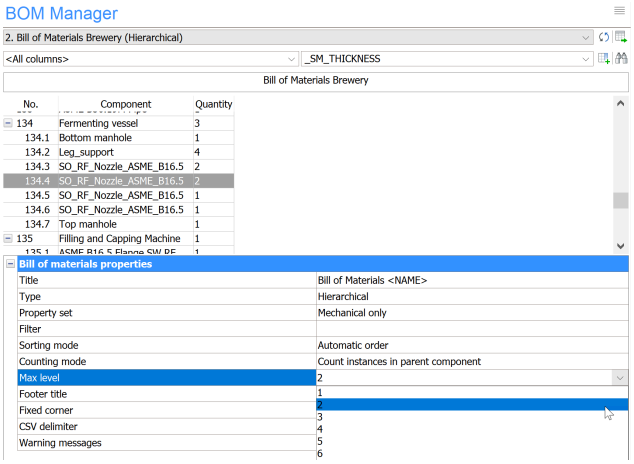
A nova opção Função de Coluna permite especificar qual coluna deve ser a origem dos balões para as vistas do desenho.



Agora é possível selecionar uma unidade para densidade do material, tanto na caixa de diálogo Materiais Físicos quanto em tabelas BOM na seção Propriedades da coluna. As unidades suportadas são: kg/m³, kg/dm³, g/cm³, lbs/ft³ and lbs/in³.

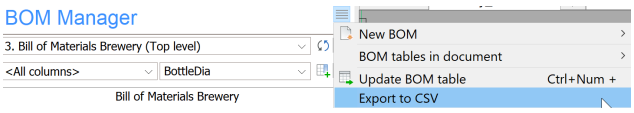


Foi introduzida uma opção para limitar o nível relatado de uma lista BOM hierárquica.



21.1.2.3 Exportar

Agora você pode exportar sua tabela BOM para um arquivo CSV.

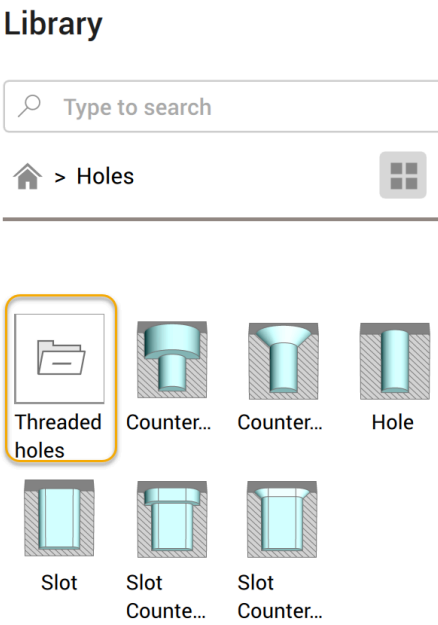


21.1.3 Bibliotecas de peças

O lançamento do BricsCAD V21 viu um aumento significativo no número de peças padronizadas disponíveis para você.

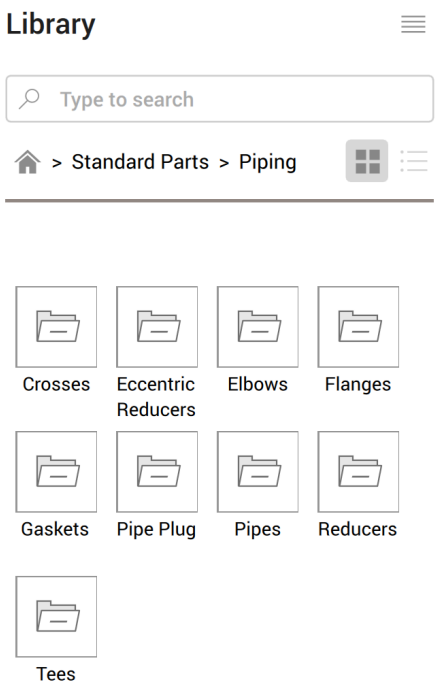
21.1.3.1 Furos

Uma biblioteca de componentes de furos roscados foi adicionada.



21.1.3.2 Tubos

Uma biblioteca de componentes de tubulação foi adicionada.



21.1.4 Parâmetros e Restrições

21.1.4.1 Tubulação

Junto com o lançamento das peças padronizadas de tubulação, um novo comando BMCONNECT vai restringir automaticamente as peças selecionadas, com base na sua proximidade.

21.1.5 Peças padrão

Restrições agora são restauradas automaticamente, depois de alterar os parâmetros de uma peça padrão.

21.1.6 Chapa metálica

As ferramentas de Sheet Metal no BricsCAD Mechanical V21 oferecem mais flexibilidade e fluxos de trabalho mais rápidos.

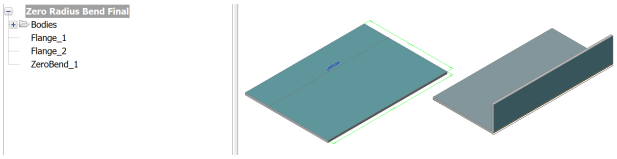
Por exemplo, uma tabela de dobra específica pode ser atribuída a cada sólido, e artefatos com espessura diferente podem existir no mesmo bloco.

21.1.6.1 Biséis

O recurso Bevel (bisel) recém-adicionado é suportado no comando SMCONVERT, no comando SMREPAIR, e no comando SMUNFOLD.

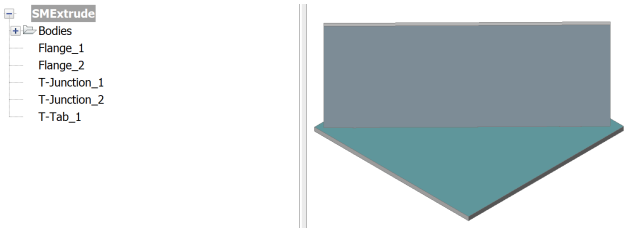
21.1.6.2 Curvas Raio Zero

Um novo recurso DobraZero permite a criação de uma dobra sem raio interno para componentes de Sheet Metal.



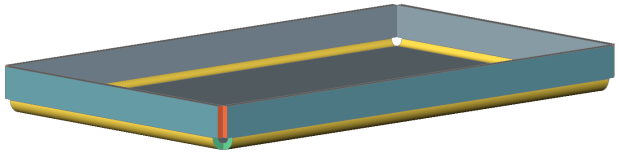
21.1.6.3 Junção-T

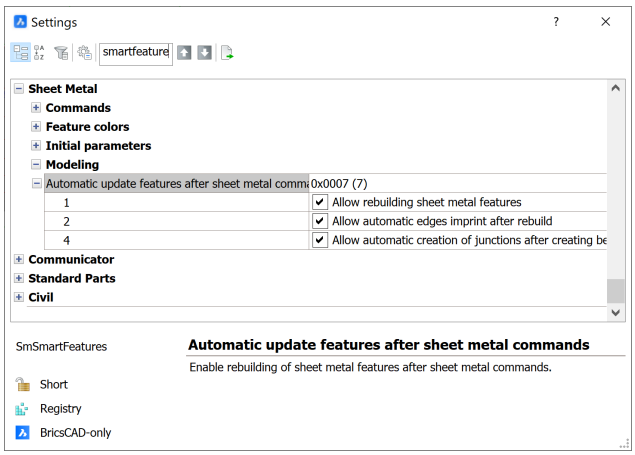
Um novo recurso Junção-T pode ser criado usando SMEXTRUDE em uma polilinha de segmento único, situada numa flange para criar uma nova flange ortogonal para a existente. Usando o recurso Junção-T, podemos então convertê-la em uma Aba-T que irá adicionar automaticamente abas à flange ortogonal e remover o material necessário da original.



21.1.6.4 Recursos Inteligentes Atualizados

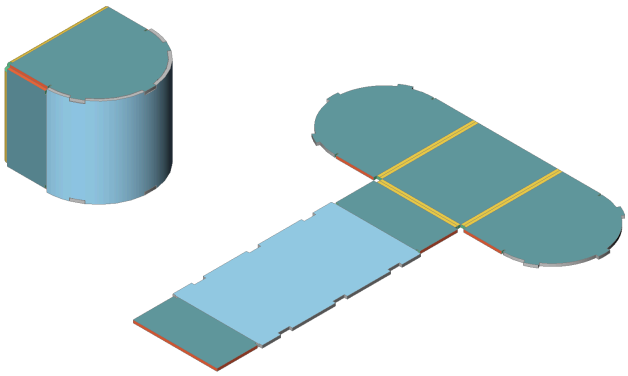
Agora você pode criar ainda mais rápido desenhos de Sheet Metal, utilizando o comando atualizado SMSMARTFEATURES, que permite criar automaticamente junções ao utilizar a ferramenta de dobra.



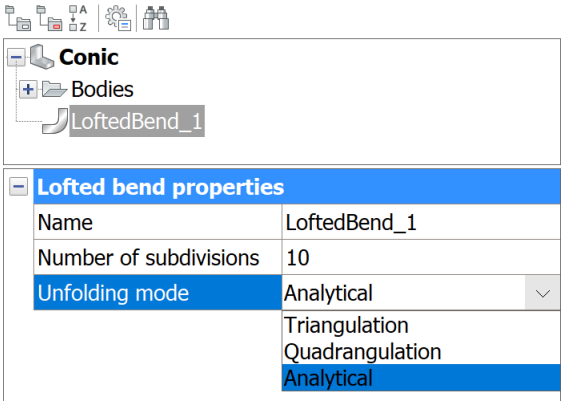


21.1.6.5 Abas

O comando SMTABCREATE agora tem uma opção para criar uma aba curvada em uma borda rígida curvada, conectando uma flange e uma dobra, incluindo dobras em loft.

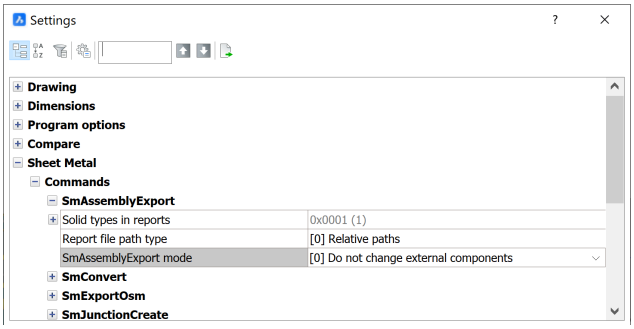


O comando SMUNFOLD oferece melhor comportamento para curvas em loft cônicas. Essas agora podem ser desdobradas analiticamente, para evitar dividi-las em um número especificado de dobras e flanges.



21.1.6.6 Exportar Montagem

A variável de sistema SMASSEMBLYEXPORTMODE permite que você mantenha recursos reconhecidos para Sheet Metal e peças de chapa metálica pobremente modeladas, ao usar o comando SMASSEMBLYEXPORT.



21.2 Começar

21.2.1 Area de trabalho: mecânica

21.2.1.1 Inicie um projeto de Projeto de Mecânica

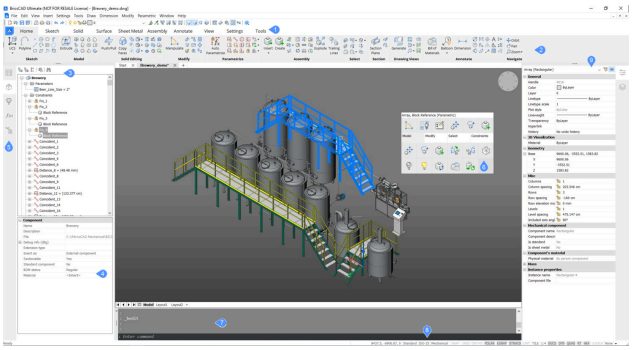
Você pode iniciar um Projeto de Mecânica:

1 Lançar o BricsCAD.

A janela *Lançador BricsCAD* é exibida.

2 Clique em **Iniciar** na seção **Projetos de Mecânica**.

21.2.1.2 Janela da aplicação BricsCAD mechanical



1 **Abas da Faixa de opções**: clique para alternar a uma aba específica.

2 **Faixa de opções**: cada aba contém vários painéis, que combinam ferramentas similares.

3 **Navegador de mecânica**: navegue por uma representação em forma de árvore do seu modelo, com a possibilidade de destacar um nó específico na área do modelo e editar suas propriedades mecânicas.

4 **Propriedades mecânicas** de um determinado nó: algumas são somente leitura. Verifique as propriedades do nó raiz na árvore.

5 **Painéis de ferramentas encaixáveis**: Navegador de mecânica e Biblioteca.

6 **Menu do cursor Quad**: aplicar uma ferramenta específica quando você passa o mouse sobre uma entidade afetada.

7 **Linha de Comando**: digite aqui o comando.

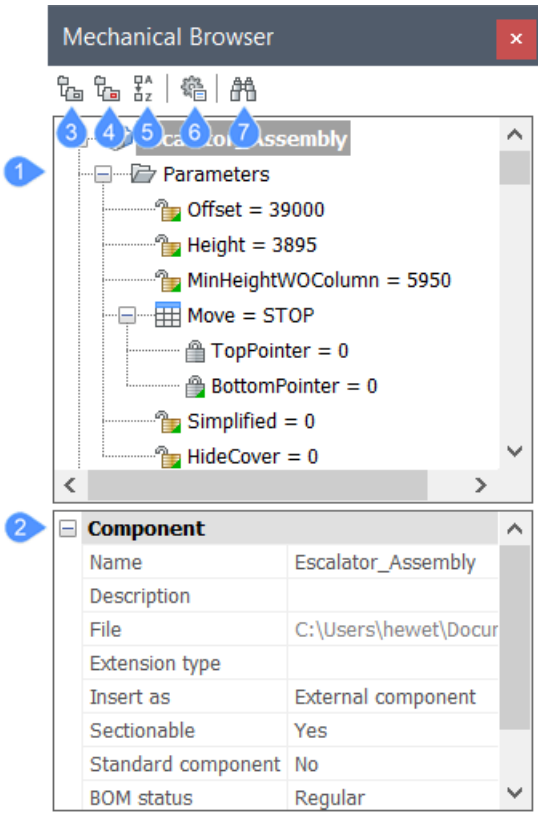
8 **Barra de Status**: a status e as opções de troca rápida.

9 **Painel Propriedades**: para ajustar propriedades não-mecânicas de entidades selecionadas; como cor ou camada.

Figura 4: Navegação no BricsCAD Mechanical

21.2.2 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

21.2.2.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

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 to all components in the assembly.

Insert standard hardware
Inserts a standard hardware part as a mechanical component in the current drawing.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create exploded view
Creates a block with an exploded representation of the current.

Dissolve
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.

Delete
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

Disable
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.

Right-clickmenusoffer additional toolsfor components:

Open
Opens the referenced drawing.

Open a copy
Opens a copy of a component insert as a new drawing.

Update
Reloads all referenced components from external files and updates BOM tables.

Replace…
Replaces a component insert.

Nota: Replacing a local insert turns it into an external insert.

Set material to component
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

BOM status
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Link to parameter
Links subcomponent parameter to the main level parameter.

21.2.2.2 Properties

Specifies the properties of the selected item.

21.2.2.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

21.2.2.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

21.2.2.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

21.2.2.6 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

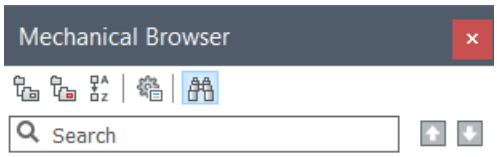
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

Sub-components of standard parts
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

21.2.2.7 Show search

Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

Search field
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.



21.2.3 Comparacao 3D

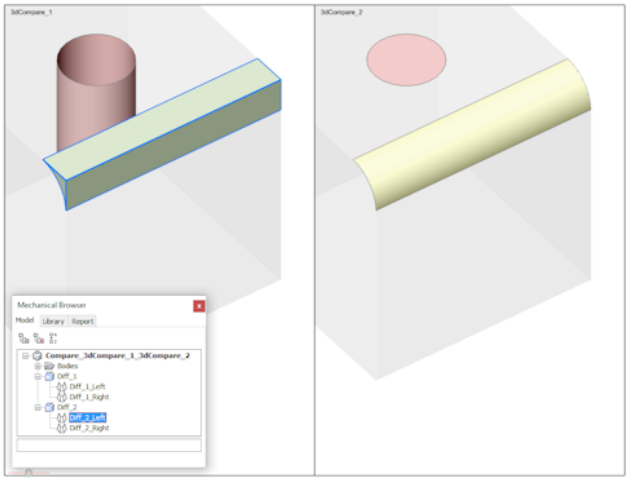
Comando: *COMPARAR3D*.

Você pode comparar dois modelos 3D, como por exemplo sólidos 3D e superfícies armazenados em 2 arquivos DWG, incluindo modelos importados no BricsCAD vindo de outros formatos de arquivo. A ferramenta pode processar grandes modelos e reporta as diferenças no **Navegador de Mecânica**. Vários modos de visualização são suportados.

21.2.3.1 Definições

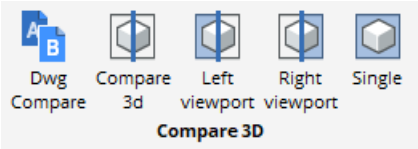
Um modelo 3D é um arquivo DWG com um ou mais sólidos 3D ou superfícies. Ao comparar dois modelos 3D, o BricsCAD primeiro tenta alinhá-los automaticamente no espaço 3D para obter o máximo de similaridade. Em seguida, todas as faces desses sólidos e superfícies 3D são comparadas geometricamente com uma dada tolerância. Se para cada face no primeiro arquivo existe uma face igual no segundo arquivo e vice-versa, esses modelos são considerados iguais. No entanto, se para uma face específica em um arquivo não houver uma face igual encontrada em outro arquivo, ou se apenas parte dessa face for igual a uma parte de uma face no outro arquivo, uma diferença é detectada. Portanto, uma diferença é uma face completa ou parte de uma face de um sólido 3D ou de uma superfície, para a qual não são encontradas partes iguais de faces completas no outro arquivo. Duas ou mais diferenças são agrupadas se forem geometricamente adjacentes umas às outras.

Um nó de Diferença é criado no **Navegador de Mecânica** para cada grupo de diferenças adjacentes. Se você expandir este nó, pode ver as diferenças específicas em que este consiste. Quando você seleciona a diferença no **Navegador de Mecânica**, o recurso correspondente é destacado também no desenho.



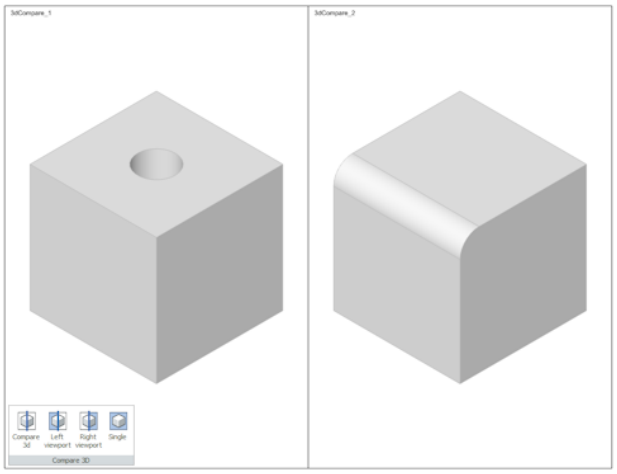
21.2.3.2 Barras

Você pode acessar o procedimento de Comparação 3D usando as ferramentas no painel **Comparar 3D** na aba **Ferramentas**.



21.2.3.3 Visualização

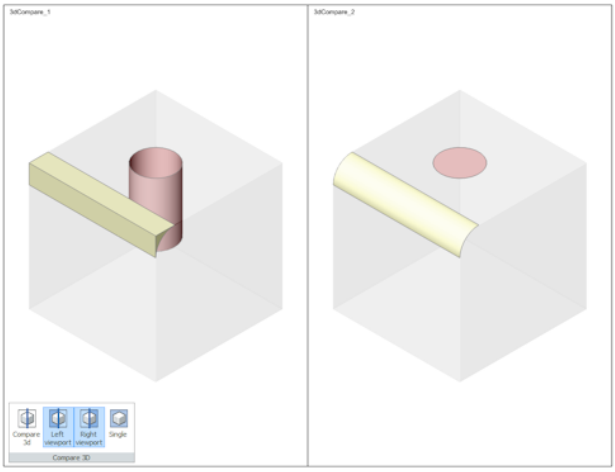
Use o comando *COMPARAR3D* para criar um novo documento DWG chamado **Compare_<Arquivo_1>_<Arquivo_2>.dwg**. Os modelos comparados são anexados como referências externas a este desenho, Um layout especial chamado **Comparação** é criado automaticamente com duas viewports, uma para cada desenho, ilustrando as diferenças entre os desenhos com uso de cores.



A exibição é controlada por meio da variável do sistema *3DCOMPAREMODE*. Se o valor de *3DCOMPAREMODE* for diferente de 0 (zero), o modelo correspondente se tornará semitransparente, enquanto as diferenças são exibidas usando cores diferentes. Cada diferença é pintada em sua cor individual. Você pode ampliar em Zoom uma diferença particular, utilizando o menu de contexto do Navegador de Mecânica.

Para exibir as diferenças:

1 Clique no botão da ferramenta **Viewport Esquerda** ( ) e/ou no botão **Viewport Direita** ( ).



2 Clique duas vezes em uma das viewports.

3 Use os atalhos do mouse/teclado para manipular a tela.

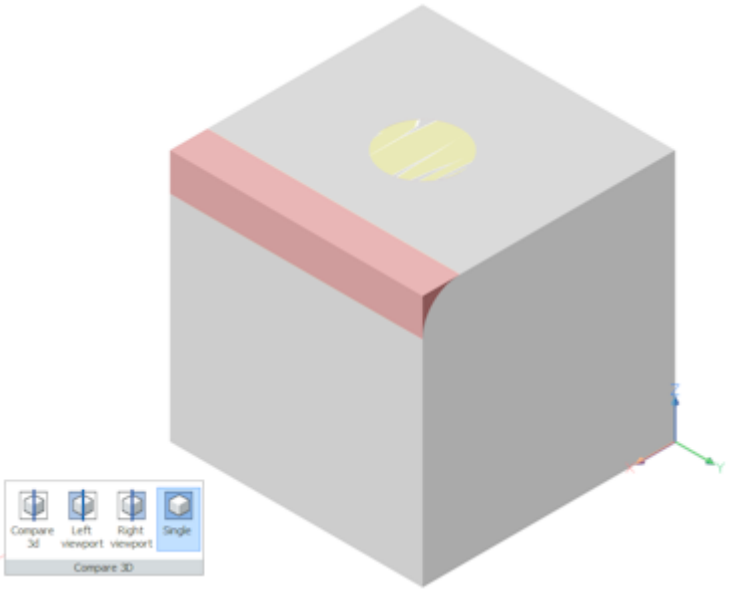
Exemplo: Fazer Zoom in/out ou rotacionar.

A exibição na outra janela de exibição é sincronizada automaticamente. Isso permite que você compare a mesma peça em ambos os modelos em detalhes.

Como alternativa, você pode:

Clique no botão **Unica** ( ).

As diferenças no Model Space são exibidas.

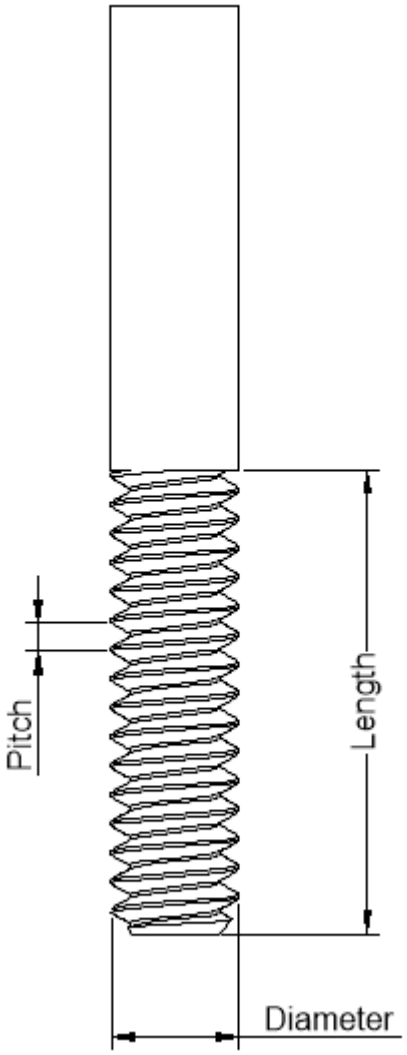


21.3 Projeto da Peças

21.3.1 Recurso de thread

Use o comando DMTHREAD para criar um recurso de rosca.

A imagem a seguir mostra a estrutura da linha:



Um recurso Rosca tem os seguintes parâmetros:

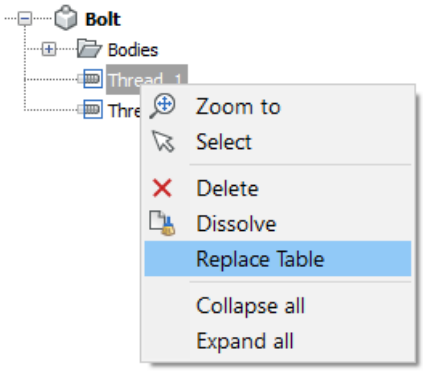
- **Padrão de rosca**

  É um conjunto de roscas padrão. Esta lista pode ser estendida com uma rosca personalizada. Ao alterar o padrão, a geometria será reconstruída automaticamente.
- **Passo** da rosca.
- **Comprimento** da rosca.
- **Diâmetro** da rosca.

- **Tipo** da rosca: macho ou fêmea. Se o tipo está definido como 'auto', o tipo será determinado automaticamente.

21.3.1.1 Adicionar uma configuração de rosca

1 Clique o botão-direito no recurso no **Navegador de Mecânica**.



2 Selecione **Substituir Tabela**.

3 Selecione um arquivo CSV com a seguinte configuração:

```
Size,Diameter,Pitch Name1, Diameter_value1, Pitch_value1 Name2,
 Diameter_value2, Pitch_value2 Name3, Diameter_value3, Pitch_value3
```

21.3.2 Metodos de Projeto de Peças

BricsCAD Mechanical permite criar e modificar modelos 3D.

Formas de criar um modelo 3D:

- Crie uma primitiva sólida 3D, como uma caixa e um cilindro.
- Esboçe um perfil 2D e extruda-o ou revolvendo este, para criar um sólido 3D.
- Modele uma superfície em 3D e dê a esta espessura para criar um sólido 3D, ou costurando entre si diversas superfícies, para criar um sólido 3D estanque/impermeável.
- Importando geometrias sólidas 3D e geometrias de superfície existentes, de outros programas CAD.

Formas de editar e adicionar mais detalhes aos seus modelos 3D:

- Empurrar/Puxar, mover e rotacionar faces.
- Criar Concordâncias ou Chanfros nas bordas do modelo.
- Aplicando Furos.
- Adicionar saliências e depressões e usar operações Booleanas, Edição Direta e ferramentas de Modelagem Deformável.

Sem dependências pai-filho = abordagem sem-histórico

O BricsCAD Mechanical não cria dependências pai-filho entre primitivas geométricas e os recursos dali gerados. Todas as alterações feitas à sua peça são locais; essas não forçam a regeneração da geometria. A abordagem sem-histórico permite que você se concentre em seu projeto, em vez de pensar em como este vai poder ser alterado no futuro.

Restrições 3D = Paramétricas

O BricsCAD Mechanical fornece um poderoso conjunto de ferramentas para tornar sua peça paramétrica, adicionando restrições 3D às suas faces e arestas, e vinculando-as a parâmetros globais usando expressões e tabelas do projeto.

21.3.3 Esboçar

O conjunto de ferramentas de esboço permite desenhar perfis 2D, que podem ser extrudados ou revolvidos para criar sólidos 3D e superfícies, ou adicionar/subtrair volume de/para entidades existentes. As ferramentas de esboço estão disponibilizadas na aba **Croqui** da Faixa de opções, da área de trabalho **Mecânica**:

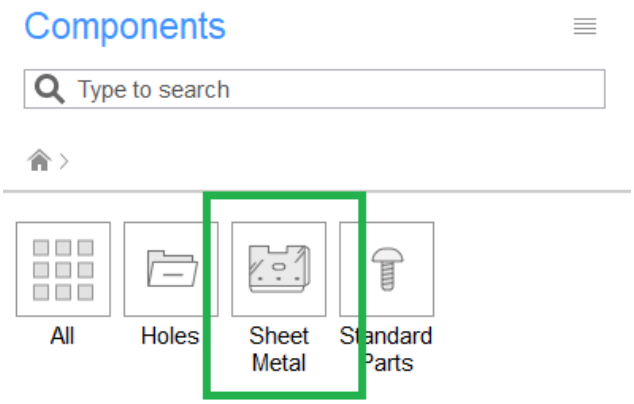


Figura 5: Faixa de opções na área de trabalho Mecânica

Os esboços são criados no plano XY do atual Sistema de Coordenadas do Usuário (UCS). Como o UCS dinâmico está habilitado por padrão, você pode criar entidades 2D na face sólida 3D sob o cursor. Use a ferramenta Alinhar UCS para posicionar o UCS de acordo com suas necessidades.

As ferramentas de desenho incluem primitivas 2D familiares, como um retângulo/polígono, um círculo/arco/elipse, uma polilinha e uma spline.

Use as ferramentas Mover, Copiar, Rotacionar 2D, Espelhar 2D e Matriz para manipular os esboços.

Use comandos *DESLOCAMENTO*, *ESTICAR*, *APARAR*, *ESTENDER*, *CONCORD/CHANFRO*, e *QUEBRAR* para editar os esboços.

Use restrições 2D para tornar os esboços paramétricos.

Nota: O valor da variável de sistema *DELOBJ* (Delete Object) controla se as entidades de esboço são excluídas quando usadas pelo *DMEXTRUDE* ou *DMREVOLVE* para criar sólidos ou superfícies 3D.

21.3.4 Modelagem Solida

O BricsCAD Mechanical fornece um rico conjunto de ferramentas para modelagem de sólidos, que inclui ferramentas tradicionais de modelagem 3D como Extrusão ou Subtração, mas também ferramentas

intuitivas de Edição Direta como; Empurrar/Puxar, Copiar Faces e poderosas Restrições 3D, que permitem ajustar a forma e o tamanho da sua peça 3D.

As ferramentas de sólidos 3D estão disponíveis na aba **Sólido** na área de trabalho **Mecânica** da Faixa de opções:



Figura 6: Solido

Tabela 1: Modelagem Solida

Ferramenta/conjuntos	Descrição
Extrusão	Criar sólidos 3D a partir de esboços.
Revolução	
Elevar	
Varredura	
Engrossar	
Ferramentas primitivas: Caixa, Cilindro, Cone, Pirâmide, Cunha, Esfera, e Toro	Criar sólidos 3D diretamente, sem esboços subjacentes.

Tabela 1: Modelagem Solida

Ferramenta/conjuntos	Descrição
Empurrar/Puxar	Editar sólidos 3D (aplicados a faces e/ou bordas).
Mover	
Rotacionar	
Copiar Faces	
Concord	
Chanfro	
Casca	Editar sólidos 3D.
Fatiar	
Ferramentas booleanas: Unir, Subtrair, Intersectar, e Separar	
Escalar	
Espelhar	
Matriz	
Apagar	Pode ser aplicado a todo o sólido 3D e a um conjunto de suas faces formando um recurso (por exemplo, uma concordância ou um furo).
Inspecionar	Encontre e corrija erros na geometria importada.
Simplificar	Reconheça automaticamente faces canônicas (planas, cilíndricas, cônicas, esféricas, toroidais) a partir de faces em spline do seu sólido 3D. Isso geralmente é necessário antes de aplicar restrições 3D.
Costurar	Costure superfícies e regiões que se tocam, que formam um limite estanque, em um sólido 3D inteiro.

Tabela 1: Modelagem Solida

Ferramenta/conjuntos	Descrição
Ferramentas de deformar: Torcer, Mover Ponto, Mover Borda, e Transformar Curva	Deformar seu sólido 3D uniformemente.
Seleção inteligente	Selecione faces e bordas de seus sólidos 3D com base em critérios geométricos e topológicos específicos, como comprimento, área e suavidade.
Parametrizar	Adicione restrições 3D a faces e bordas de sólidos 3D, para alinhá-las e redimensioná-las. Torne sua peça 3D paramétrica.
Intenção Projeto	Aplique automaticamente restrições geométricas 3D, como paralelo, perpendicular, coaxial, e tangente.

Nota: O BricsCAD é um modelador sem-histórico; este não cria dependências pai-filho entre os recursos, nem regenera a geometria na alteração dos parâmetros. Você pode incorporar controle paramétrico completo em qualquer peça 3D (incluindo as importadas) usando restrições 3D.

21.3.5 Modelagem de Superficie

O BricsCAD Mechanical fornece um conjunto poderoso de ferramentas para modelagem de superfície e conversão de sólidos 3D em superfícies, e vice-versa.

As ferramentas de Modelagem de Superfície estão disponíveis na aba **Superfície** da Faixa da área de trabalho **Mecânica**:

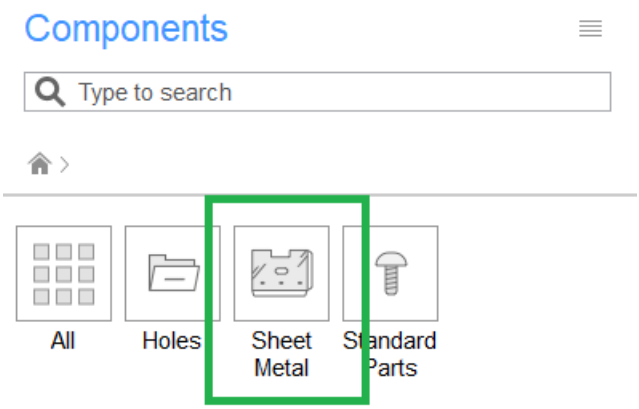


Figura 7: Superfície

Modelagem de Superfície e Modelagem de Sólidos são ferramentas semelhantes, relacionadas à edição e aplicação de restrições em 3D. A única diferença é que as ferramentas de superfície Extrusão/Revolucao/Loft/Varredura podem criar entidades de superfície a partir de perfis 2D fechados, e também aceitam perfis não-clonados.

A ferramenta Regiao pode criar uma entidade de região a partir de um perfil 2D fechado, e regiões podem ser costuradas entre si para criar uma superfície maior.

Outros métodos para criar uma superfície são por extrair uma ou mais faces adjacentes de um sólido 3D/superfície, e por explodir um sólido 3D em um conjunto de superfícies e regiões.

21.3.6 Modelagem de Superfície em Formas livres

21.3.6.1 Visão Geral

Adicione distorções complexas aos seus modelos sólidos ou de superfície usando ferramentas de deformação do BricsCAD Mechanical. Você pode criar formas suaves ou orgânicas. Modifique seus sólidos e superfícies movendo, rotacionando e torcendo suas bordas, para alterar a geometria de forma independente e sem-histórico de design.

A abordagem sem-histórico permite que você se concentre em seu design, em vez de pensar em como ele pode ser alterado no futuro.

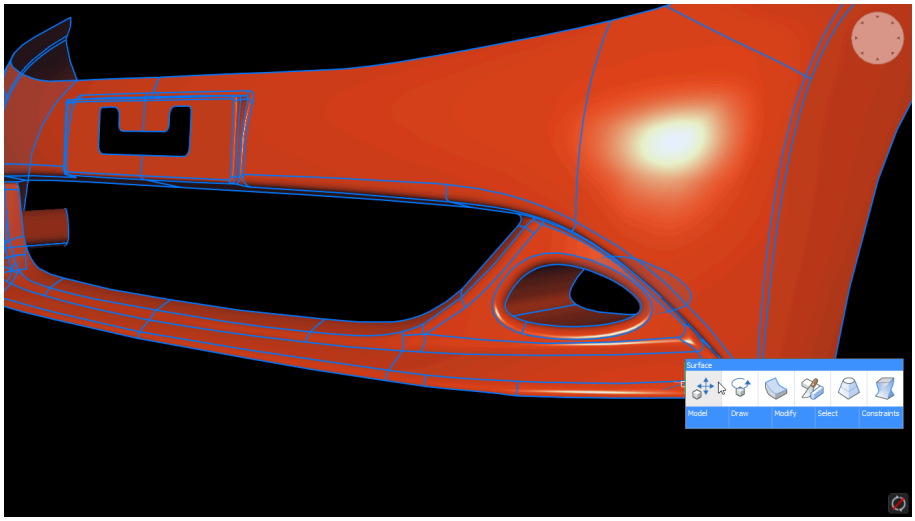


Figura 8: Modelagem de Superfície em Formas livres

21.3.6.2 Ferramentas de deformação

Você pode usar as seguintes ferramentas de deformação:



Figura 9: Deformar

Tabela 2: Ferramentas de deformação

Ferramenta	Comando	Descrição
Mover Borda	*DMDEFORMMOVE*	Deforma faces movendo e rotacionando suas arestas.
Mover Ponto	*DMDEFORMPOINT*	Deforme o conjunto de faces selecionadas movendo um ponto.
Transformar Curva	*DMDEFORMCURVE*	Deforma faces, alterando a geometria de suas bordas.
Torcer	*DMTWIST*	Deforme o objeto 3D torcendo uma porção, em torno de um determinado eixo.

Dica: Assista ao tutorial: Usar modelagem de superfície e modelagem deformável.

21.4 Projeto de Montagem

21.4.1 Projetos de Montagem Bottom-up e Top-down

21.4.1.1 Visão Geral

Projeto de montagem representa a estrutura de um produto projetado, e consiste em usar componentes para agrupar entidades geométricas em hierarquias. O projeto de montagem é comumente usado em aplicações de Projeto Assistido por Computador na Mecânica (MCAD), e simplifica o projeto de produtos complexos contendo milhares ou milhões de entidades geométricas.

Os seguintes termos são usados neste guia:

- Uma montagem é um componente que contém inserções de outros componentes
- Uma sub-montagem é uma montagem separada, que é inserida em outra montagem
- Uma peça é um componente que não contém inserções de outro componente.

Uma montagem com componentes inseridos (que por sua vez podem ser montados com outros componentes inseridos) forma uma árvore que é chamada de Estrutura Mecânica do modelo 3D. Esta árvore é exibida no Navegador de Mecânica.

Existem 2 maneiras principais de projetar produtos de mecânica: as abordagens **Top-down** (de cima para baixo) e **Bottom-up** (de baixo para cima). Você também pode adicionar estrutura mecânica a uma geometria existente, projetada no BricsCAD ou importada de outra fonte.

Use o comando *BMNEW* para criar um novo documento (desenho), para um novo componente mecânico.

Use o comando *BMMECH* para converter blocos existentes, e referências externas, para componentes.

Use o comando *BMUNMECH* para fazer o caminho inverso.

21.4.1.2 Projetar em Bottom-Up (de Baixo para Cima)

Com a abordagem Bottom-Up (de baixo-para-cima), o usuário começa a projetar um produto detalhando seu componente de nível mais básico. Esses componentes são então agrupados e vão se transformando em componentes de nível mais elevado. O processo é continuado até que o componente mais alto (o superior) seja então correspondente ao próprio produto.

O comando principal para o design de baixo-para-cima é o *BMINSERT*. Use este comando para inserir um componente existente no desenho atual.

21.4.1.3 Projetar em Top-Down (de Cima para Baixo)

Com a abordagem Top-Down (de cima-para-baixo), o usuário começa a projetar um produto definindo, mas não detalhando, seus componentes de primeiro nível (mais alto). A partir daí, então cada componente é refinado com mais detalhes, pela definição de seus subcomponentes. O processo é continuado até que todos os componentes de nível mais básico estejam definidos e detalhados.

Use o comando *BMFORM* para criar um novo subcomponente para um componente existente.

21.4.1.4 Adicionar Estrutura Mecânica a uma Geometria Existente

Você pode adicionar hierarquia de componentes a qualquer desenho existente, seja criado no BricsCAD ou importado de um formato CAD diferente.

Use o comando *BMFORM* para converter um conjunto selecionado de entidades geométricas em um subcomponente do desenho atual.

21.4.1.5 Alterar Estrutura Mecânica

Durante o processo de design, você pode perceber que um determinado componente deve ser movido para cima ou para baixo na hierarquia de componentes.

Use o comando *BMFORM* para formar um novo subcomponente para o desenho atual, que conterá um conjunto selecionado de um componente.

Use o comando *BMDISSOLVE* para excluir um componente e mover todos os seus subcomponentes para o nível do componente dissolvido.

21.4.1.6 Usar Componentes Padrão/Standard

Alguns componentes padrão são:

- parafusos
- porcas
- arruelas
- rolamentos
- furos.

O BricsCAD fornece uma enorme biblioteca de componentes padrão, mais de 30.000 elementos de detalhe em uma variedade de tamanhos. Você pode aumentar significativamente o desempenho do projeto usando peças padrão.

Use o painel Componentes para inserir componentes padrão em suas peças e montagens.

21.4.1.7 Restrições na Montagem

Restrições de montagem são Restrições 3D comuns, que vinculam faces e bordas de corpos para representar os componentes de uma montagem. Você pode vincular Restrições 3D tanto no nível superior (top-level) como no nível de subcomponentes.

21.4.1.8 Visibilidade do Componente

Um componente dentro de uma montagem complexa pode parecer invisível devido a outros componentes que o rodeiam.

Exemplo: um pistão é invisível dentro de um bloco de cilindros. Para tornar esse componente visível, talvez seja necessário ocultar os outros componentes ao seu redor.

O usuário pode ocultar qualquer subcomponente inserido no desenho atual ou quaisquer subcomponentes, em qualquer nível de aninhamento.

Você também pode tornar um componente transparente e/ou visualizá-lo como um modelo de em arame (wireframe).

Use o comando *BMHIDE* para ocultar um subcomponente em particular.

Use o comando *BMSSHOW* para tornar visível um subcomponente oculto.

21.4.2 Comandos de Projeto de Montagem, e Barras de Ferramentas

21.4.2.1 Comandos

Tabela 3: Comandos de Projeto de Montagem

Ícone	Comando	Descrição
	*BMNEW*	Cria um novo componente mecânico, como um novo arquivo de desenho.

Tabela 3: Comandos de Projeto de Montagem

Ícone	Comando	Descrição
	*BMMECH*	Converte o desenho atual em um componente mecânico principal.
	*BMINSERT*	Insere um componente mecânico existente, dentro do desenho atual
	*BMINSERT*	Insere um componente mecânico existente no desenho atual com redimensionamento automático da nova peça, conectando-a à peça 'pai'.
	*BMHARDWARE*	Abre o Painel de Componentes com os Componentes Mecânicos Padrão
	*BMFORM*	Cria um novo componente mecânico e insere este no desenho atual.
	*BMCONNECT*	Conecta duas peças padrão de Tubulação através da criação de restrições 3D entre suas entidades de conexão.
	*BMREPLACE*	Substitui um componente inserido.
	*BMLOCALIZE*	Substitui componentes externos por componentes locais.
	*BMEXTERNALIZE*	Substitui componentes locais por componentes externos.
	*BMOPEN*	Abre o desenho de origem de um componente mecânico externo.
	*BMOPENCOPY*	Abre uma cópia de um componente inserido, como um novo desenho.
	*BMDISSOLVE*	Dissolve um componente mecânico inserido no desenho atual.
	BMLINK	Altera sólidos 3D de destino de recursos baseados-em-componentes.

Tabela 3: Comandos de Projeto de Montagem

Ícone	Comando	Descrição
	*BMUNLINK*	Quebra a conexão entre um componente inserido (ex., uma janela) e um sólido 3D (ex., uma parede) mantendo a abertura.
	*BMHIDE*	Oculta um componente mecânico ou subcomponente, no desenho atual.
	*BMSHOW*	Mostra um componente mecânico anteriormente oculto ou um subcomponente, no desenho atual.
	*BMVSTYLE*	Aplica um estilo visual para um componente mecânico.
	*BMBALLOON*	Cria entidades associativas do tipo balão para componentes de montagem no Model Space e para suas vistas geradas num layout do Paper Space.
	*BMBALLOON*	Permite o posicionamento automático do balões associativos em todos os componentes, em uma vista de desenho específica, num layout do Paper Space.
	*BMBOM*	Insere a tabela Lista de Materiais (BOM) no desenho atual
	*BMMASSPROP*	Calcula propriedades de massa para o modelo atual, utilizando densidades atribuídas a seus componentes.
	*BMUPDATE*	Recarrega todos os componentes referenciados a partir de arquivos externos, e atualiza as tabelas BOM (Lista de Materiais).
	*BMUPDATE*	Recarrega todos os componentes referenciados a partir de arquivos externos, e atualiza as tabelas BOM (Lista de Materiais).
	*BMEXPLODE*	Cria uma representação de vista explodida para uma montagem, movendo componentes para torná-los todos visíveis, armazena esta vista em um novo bloco, e insere o bloco no Model Space.

Tabela 3: Comandos de Projeto de Montagem

Ícone	Comando	Descrição
	*BMEXPLODEMOVE*	Move as peças selecionadas para formar uma representação explodida em uma determinada direção, levando em consideração possíveis colisões físicas entre componentes (semelhante à opção Linear do comando *BMEXPLODE*).
	*BMTRAILINGLINES*	Cria todas as linhas de arraste necessárias para as peças selecionadas.
	*NAVEGADORMECANICAABRIR*	Exibe o Navegador de Mecânica.
	*PARAMETROSPAINELABRIR*	Exibe o Gerenciador de Parâmetros.
	*BMDEPENDENCIES*	Lista todos os arquivos na janela de comando, contendo definições dos componentes inseridos na montagem.
	*BMRECOVER*	Recupera uma estrutura mecânica fracionada.
	*BMUNMECH*	Converte o atual componente mecânico em um desenho regular.
	BMXCONVERT	Converte sólidos X-Hardware no desenho atual, em componentes mecânicos.

21.4.2.2 Ferram

Você pode usar as seguintes ferramentas para trabalhar com componentes mecânicos:

1 Nas barras de **Montagem**:

- **Montagem**

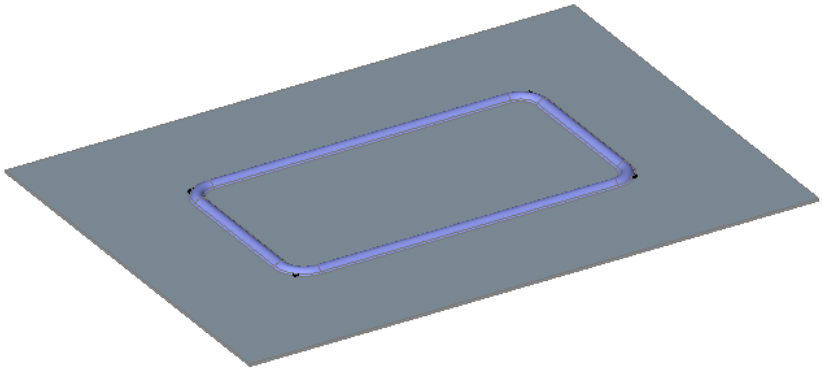


Figura 10: Montagem

- **Inserir Montagem**



Figura 11: Inserir Montagem

- **Explodir Montagem**

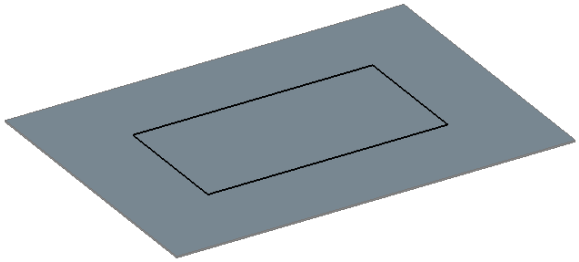


Figura 12: Montagem Explodir

- **Visualização da Montagem**

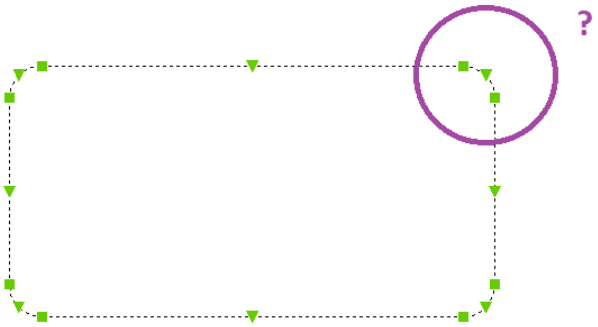


Figura 13: Visualização de Montagem

2 No menu **Montagem** (somente na Area de trabalho Mecânica):

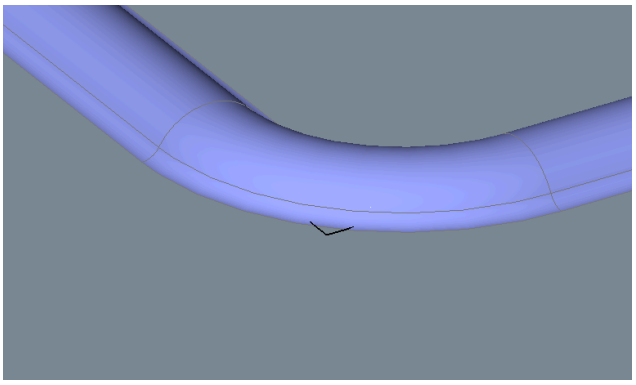


Figura 14: Menu de Montagem

3 Na aba **Montagem** da Faixa de opções na Area de trabalho Mecânica:

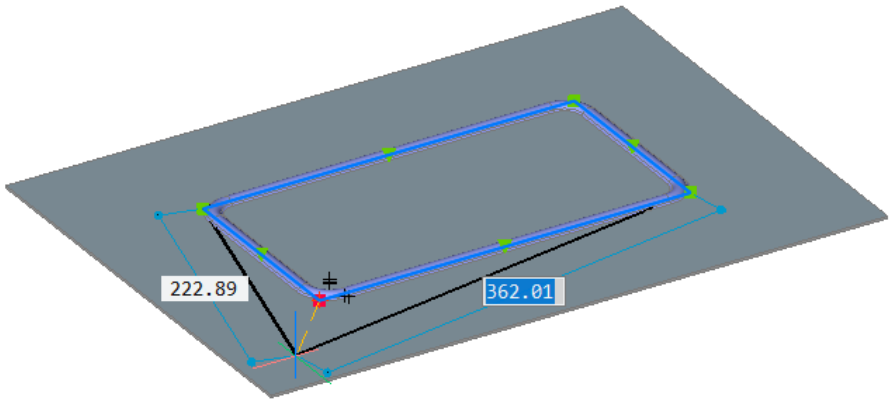


Figura 15: Montagem

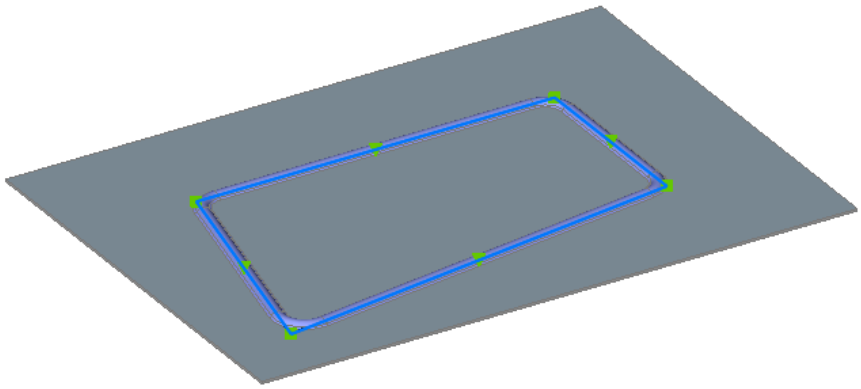


Figura 16: Ferram

21.4.3 Propriedades do Componente

As propriedades dos componentes e os menus de contexto estão localizados no painel **Navegador de Mecânica**.

Os menus de contexto de um componente são exibidos quando você clica o botão-direito com o cursor sobre este componente. As propriedades do componente são exibidas automaticamente ao selecionar um ou mais componentes no Navegador de Mecânica.



Figura 17: Navegador de Mecânica

Componentes podem ser: **Principal**, **Inserção** e **Subcomponente**.

Tabela 4: Componentes

Tipo de componente	Estilo visual Renderizado	Estilo visual em Arame	oCultas
Componente sem subcomponentes	*	*	*
Componente com subcomponentes	*	*	*

Tabela 4: Componentes

Tipo de componente	Estilo visual Renderizado	Estilo visual em Arame	oCultas
Componente peça padrão			

* Componentes externos contêm uma seta verde:  ou  .

Tabela 5: Propriedades do Componente

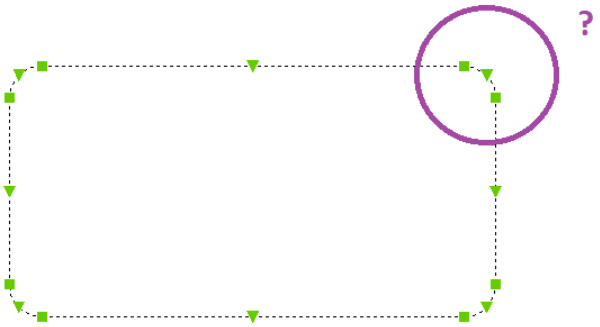
Menus de contexto	
Menu de contexto do Componente Principal	Clique o botão-direito no nome do componente principal:  **Figura 18:** Menu de contexto do Componente Principal • **Atualizar**: atualiza a hierarquia dos componentes mecânicos para o desenho atual, no caso em que arquivos de desenho referenciados de subcomponentes tenham sido modificados. • **Estilo Visual**: **Tudo por Viewport**: aplique o atual **Estilo Visual** a todos os componentes da montagem (consulte o comando *MODOAPLICARCOR*). • **Inserir hardware padrão**: insere uma peça de hardware padrão como um componente mecânico no desenho atual. • **Trocar tudo para local**: troca todos os componentes para componentes internos (consulte o comando *BMLOCALIZE*). • **Trocar tudo para externo**: troca todos os componentes para componentes externos (consulte o comando *BMEXTERNALIZE*). • **Adicionar novo parâmetro**: cria um novo parâmetro na montagem. • **Selecionar tudo**: seleciona todos os componentes com a mesma definição. • **Selecionar o mesmo**: seleciona todos os componentes com o mesmo nome, e os mesmos valores de parâmetro. • **Destacar tudo**: destaca todos os componentes com a mesma definição. • **Destacar mesmo**: destaca todos os componentes com o mesmo nome e os mesmos valores de parâmetro. • **Criar vista explodida**: cria um bloco com uma representação explodida da montagem atual. • **Recolher tudo**: recolhe o componente principal e todos os componentes e subcomponentes. • **Expandir tudo**: expande o componente principal e todos os componentes e subcomponentes.

Tabela 5: Propriedades do Componente

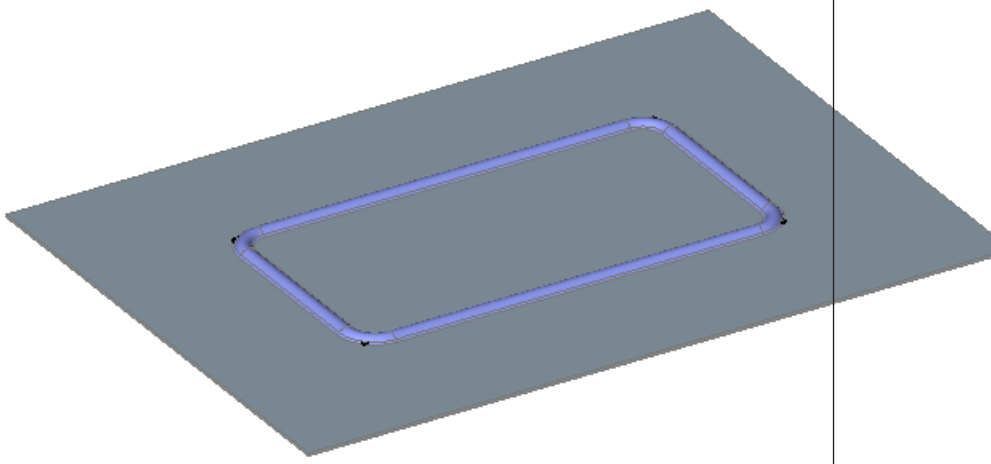
Menus de contexto	
Menu de contexto dos componentes	Clique o botão-direito num nome de componente. Menu de contexto dos componentes externos (esq), menu de contexto dos componentes locais (dir):  • **Abrir**: abre o desenho referenciado (ver comando *BMOPEN*). • **Abrir uma cópia**: abre uma cópia de um componente inserido, como um novo desenho (ver comando *BMOPENCOPY*). • **Atualizar**: recarrega todos os componentes referenciados de arquivos externos e atualiza as tabelas BOM (consulte o comando *BMUPDATE*). • **Substituir**: substitui uma inserção de componente (consulte o comando *BMREPLACE*). Ao substituir uma inserção local transforma esta em uma inserção externa. • **Substituir todas inserções**: substitui todas as inserções que se referem à mesma origem (consulte o comando *BMREPLACE*). Ao substituir inserções locais transforma essas em inserções externas. • **Trocar para local**: troca um componente externo por um componente interno (ver comando *BMLOCALIZE*). • **Trocar para externo**: troca um componente interno para um componente externo (consulte o comando *BMEXTERNALIZE*). • **Definir material para componente**: atribui um material físico a um componente local. • **Remover material do componente**: remove a definição de material físico de um componente local. • **Status BOM**: parâmetro para controlar a aparência do componente nas Tabelas BOM. • **Dissolver**: dissolve um componente mecânico inserido no desenho atual (ver comando *BMDISSOLVE*). • **Desvincular**: quebra a conexão entre uma inserção de componente (ex., uma Janela) e um sólido 3D (ex., uma Parede) mantendo a abertura. • **Ocultar** ( ) / **Mostrar** ( ): oculta ou exibe o componente selecionado. • **Excluir todas as inserções da seção**: defina a propriedade **Seccionável** de todas as inserções semelhantes para *NÃO*. Defina se uma inserção é afetada pelo comando *VISTACORTE*. • **Estilo Visual**: exibe o menu **Estilo Visual**. Todos os Estilos Visuais que estão salvos no desenho atual são disponíveis. - **Por componente pai**: renderiza o componente selecionado de acordo com o modo de sombreamento do componente pai (consulte o comando *MODOAPLICARCOR*).

Tabela 5: Propriedades do Componente

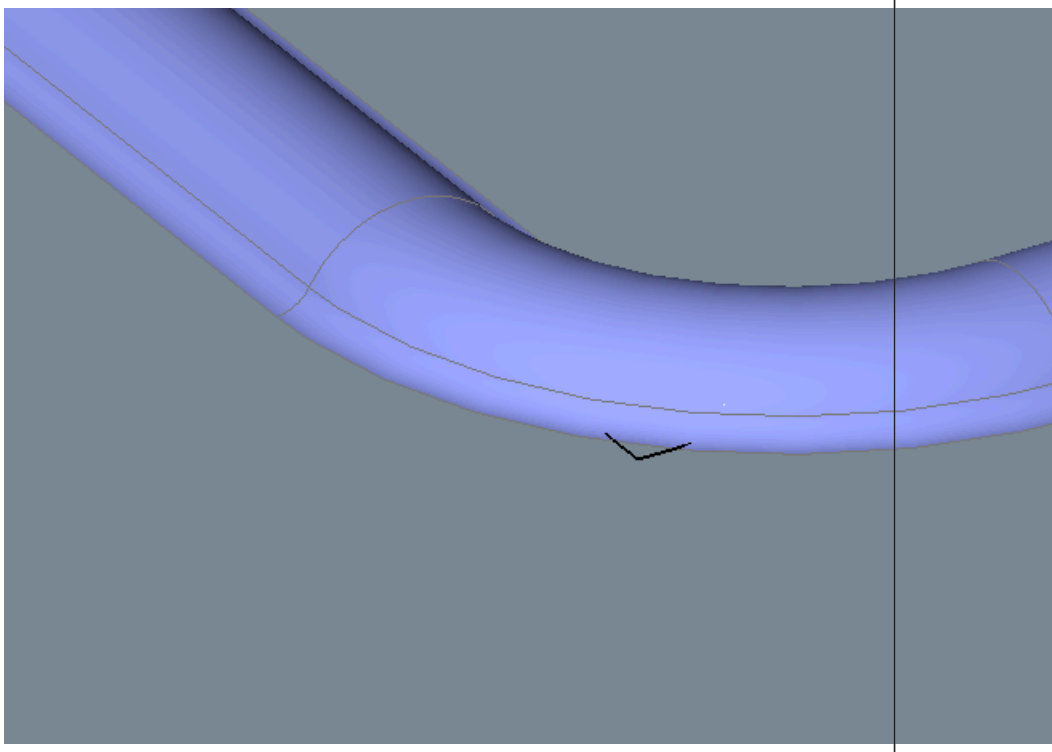
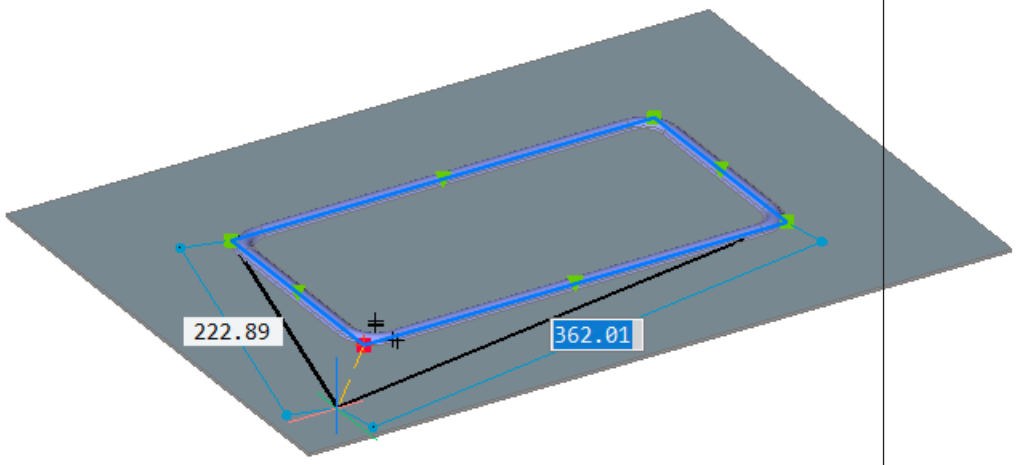
Menus de contexto	
Menu de contexto do subcomponente	Menu de contexto dos subcomponentes externos (esq), menu de contexto dos componentes locais (dir):   • **Abrir**: abre o desenho referenciado (ver comando *BMOPEN*) • **Abrir uma cópia**: abre uma cópia de um componente inserido, como um novo desenho (ver comando *BMOPENCOPY*). • **Ocultar** ( ) / **Mostrar** ( ): oculta ou exibe o subcomponente selecionado. • **Excluir todas as inserções da seção**: define a propriedade **Seccionável** de todas as inserções semelhantes para *NÃO*. Define se uma inserção é afetada pelo comando *VISTACORTE*. • **Estilo Visual**: ver Estilo Visual do Componente. • **Zoom para**: faz Zoom no conjunto de seleção. • **Selecionar tudo**: seleciona todos os componentes com a mesma definição. • **Selecionar o mesmo**: seleciona todos os componentes com a mesma definição e os mesmos valores de parâmetro. • **Destacar tudo**: destaca todos os componentes com a mesma definição. • **Destacar o mesmo**: destaca todos os componentes com a mesma

Tabela 5: Propriedades do Componente

Menus de contexto	
Menu de contexto do componente parâmetro	Botão-direito sobre um parâmetro: • **Link para parâmetro**: cria uma ligação entre um ou mais parâmetros de componente e parâmetros de montagem. • **Recolher tudo**: recolhe o componente principal e todos os componentes e subcomponentes. • **Expandir tudo**: expande o componente principal e todos os componentes e subcomponentes.
Propriedades	

Tabela 5: Propriedades do Componente

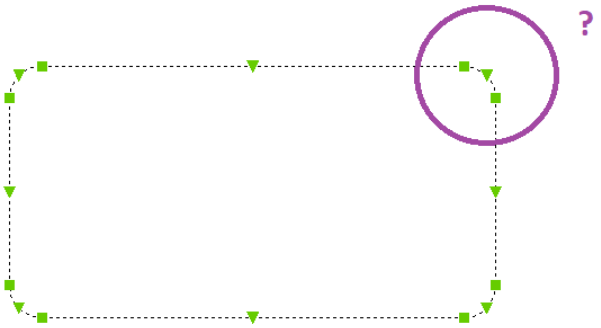
Menus de contexto	
Propriedades do Componente Principal	Clique o nome do componente principal:  **Figura 19:** Propriedades do Componente Principal • **Nome**: digite um nome para o componente principal. Por padrão, o nome do componente principal é igual ao nome do arquivo. • **Descrição**: descrição opcional do componente principal. • **Arquivo**: mostra o caminho e o nome do desenho. • **Tipo de extensão**: identifica o componente como um Componente BIM ou um recurso de Sheet Metal. • **Inserir como**: selecione **Componente interno** ou **Componente externo**. • **Seccionável**: defina se o componente é afetado pelo comando *VISTACORTE*. • **Componente padrão**: define se este componente é um componente padrão ou não. • **Status da BOM**: define o comportamento desse componente em uma tabela BOM. • **Material**: material do componente principal. O **Nome**, a **Densidade** e a opcional **Descrição** são definidos na caixa de diálogo Materiais Físicos. Selecione o nó **Material** e execute um dos seguintes procedimentos: - Clique no botão **Procurar** (…) para abrir a caixa de diálogo Materiais Físicos e escolher um material na lista do Projeto ou da Biblioteca. A propriedade Densidade do material selecionado é usada pelo comando *BMMASSPROP*. - Clique no botão **Excluir** (x). O campo **Material** agora lê <*Herdar*>. Quando o componente é inserido em uma montagem, a definição de **Material** é copiada do componente principal da montagem.

Tabela 5: Propriedades do Componente

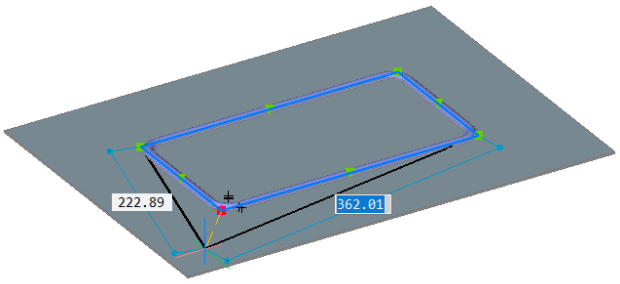
Menus de contexto	
Propriedades do componente inserido	Clique o nome do componente:  **Figura 20:** Propriedades do componente inserido • **Nome**: digite um nome para o componente. Por padrão, o nome do componente principal é igual ao nome do arquivo de origem. • **Nome do componente**: nome do componente conforme definido no arquivo de origem. • **Descrição**: descrição do componente conforme definido no arquivo de origem. • **Visível**: mostra ou oculta o componente na montagem. Escolha entre: - Clique no campo **Visível**, clique no botão de seta para baixo e escolha *Sim* ou *Não*. - Clique-duplo no campo **Visível** para trocar a visibilidade do componente. • **Seccionável**: defina se o componente é afetado pelo comando *VISTACORTE*. • **Componente padrão**: define se este componente é padrão ou não. • **Status da BOM**: define o comportamento desse componente em uma tabela BOM. • **Estilo visual**: permite controlar o estilo visual para o componente. Por padrão, o estilo visual é herdado da janela de exibição. Clique no campo **Estilo visual**, clique no botão de seta para baixo e escolha um estilo visual na lista. • **Arquivo**: mostra o caminho e o nome do arquivo referenciado. • **Material**: mostra o material do componente. Se definido para *<Herdar>*, o material é copiado do componente principal. Se o material for definido explicitamente no arquivo de origem, esse material será usado.

Tabela 5: Propriedades do Componente

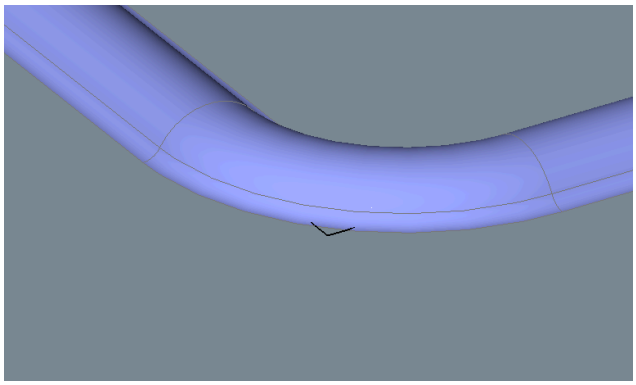
Menus de contexto	
Propriedades do subcomponente	Clique o nome do subcomponente:  **Figura 21:** Propriedades do subcomponente • **Nome**: nome do componente conforme definido no arquivo de origem do componente pai. • **Nome do componente**: nome do componente conforme definido no arquivo de origem. • **Descrição**: descrição do componente conforme definido no arquivo de origem. • **Visível**: mostre ou oculte o componente na montagem. Escolher entre: - Clique no campo **Visível**, clique no botão de seta para baixo e escolha *Sim* ou *Não*. - Clique-duplo no campo **Visível** para trocar a visibilidade do componente. • **Seccionável**: defina se o componente é afetado pelo comando *VISTACORTE*. • **Componente padrão**: define se este componente é padrão ou não. • **Status da BOM**: define o comportamento desse componente em uma tabela BOM. • **Estilo visual**: controla o estilo visual do componente. Por padrão, o estilo visual é herdado da viewport. Clique no campo **Estilo visual**, clique no botão de seta para baixo e escolha um estilo visual na lista. • **Arquivo**: mostra o caminho e o nome do arquivo referenciado. • **Material**: mostra o material do subcomponente. Se definido para *<Herdar>*, o material é copiado do componente principal. Se o material for definido explicitamente no arquivo de origem, esse material será usado.

Tabela 5: Propriedades do Componente

Menus de contexto	
Métodos de seleção	
Selecione o(s) componente no modelo	O(s) componente se destaca no Navegador de Mecânica.
Clique em um componente no Navegador de Mecânica	O componente se destaca no desenho. Pressione a barra de espaço para selecionar o componente.

21.4.4 Propriedades de massa da montagem

21.4.4.1 Visão Geral

Usar o comando *BMMASSPROP*.

Uma montagem pode consistir em várias peças feitas de diferentes materiais. Para calcular corretamente a massa, centro de gravidade (centróide) e a inércia de tais montagens, o BricsCAD suporta densidades individuais para componentes.

Um material físico para um componente mecânico é definido na propriedade Material no **Navegador de Mecânica**.

Por padrão, um componente herda a densidade da montagem na qual este foi inserido, ou a partir da qual foi formado.

21.4.4.2 Definir as unidades de massa

1 Abra a caixa de diálogo *Configurações*.

2 Expanda as configurações sobre **Unidades de propriedade** que se encontra sob **Desenho > Desenho > Unid. no desenho**.

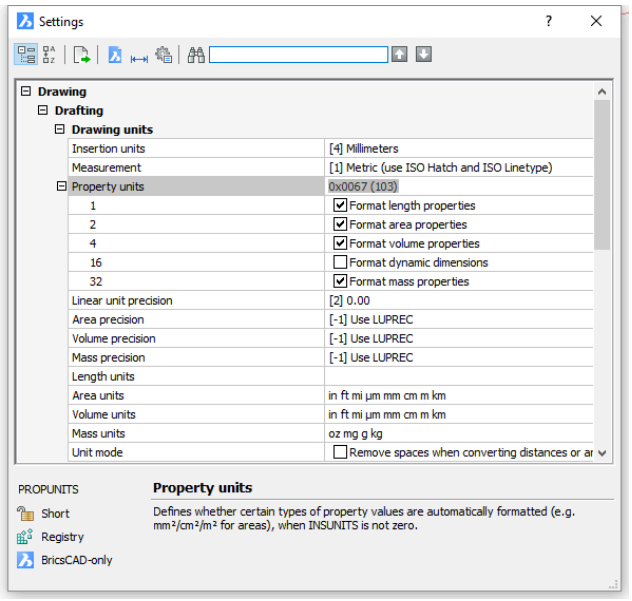


Figura 22: Configurações: Unidades de propriedade

3 Marque a caixa **Formatar propriedades de massa** para habilitar a formatação dos valores de massa com as unidades adequadas.

4 Selecione o campo **Unid. massa**.

5 Clique no botão **Procurar** ....

6 Especifique as unidades de massa desejadas, na caixa de diálogo *MASSUNITS*.

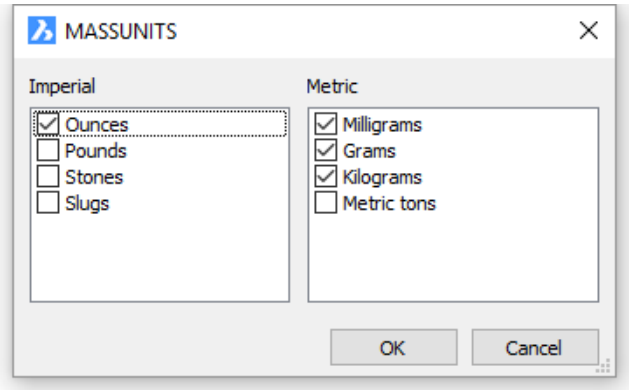


Figura 23: MASSUNITS

Nota: A configuração **Unidades de massa** afeta apenas os valores de massa. Outras propriedades de massa, como densidade ou momentos de inércia, serão formatadas em unidades SI para o sistema métrico e em unidades imperiais para o sistema imperial, independentemente do valor da configuração de **unidades de massa**.

7 Especificar a precisão desejada no campo **Precisão da massa**.

8 Abra a caixa de diálogo *Configurações*.

21.4.4.3 Calcular as propriedades de massa

Usar o comando *BMMASSPROP*.

Você pode calcular propriedades de massa para todo o modelo ou para uma seleção de sólidos 3D e/ou subcomponentes. Os valores de massa, volume, centróide, momentos de inércia, produtos de inércia, raios de giração, momentos principais, e direções X-Y-Z sobre o centróide são relatados na barra de Comando.

Opcionalmente, você pode alinhar o Sistema de Coordenadas do Usuário com os eixos de inércia. Como resultado, o centro de massa ou centroide do componente fica na origem do UCS, o qual pode ser útil para investigar o comportamento de um produto sob a força de gravidade.

21.4.5 Componentes Mecanicos Padrao

21.4.5.1 Visão Geral

Use o comando *COMPONENTESPAINELABRIR*.

O BricsCAD vem com uma biblioteca de mais de 1.000 peças mecânicas padrão disponíveis em mais de 170.000 tamanhos diferentes. Eles estão disponíveis no painel Biblioteca e podem ser arrastados da biblioteca e soltos em sua montagem. Você pode ajustar os parâmetros no painel Propriedades, que lista temporariamente os parâmetros do componente que está sendo inserido. Após, parâmetros de inserção, podem ser editados no Navegador de Mecânica e no painel Propriedades.

Na área de trabalho Mecânica, o painel Biblioteca contém 3 categorias:

- Peças Padrão
- Furos
- Sheet Metal

21.4.5.2 Peças Padrão (Hardware)

A categoria Peças Padrão contém hardware padrão paramétrico, agrupados em 4 subcategorias:

- Fixadores, como parafusos, porcas e arruelas.
- Peças de Máquina, como rolamentos, e rodas dentadas.
- Tubulação, como tubos, cotovelos, e flanges.
- Formas Estruturais, como vigas, canais, e barras.

O conteúdo de cada subcategoria é estruturado por tipo, subtipo, e padrões.

Exemplo: Standard Parts / Fasteners / Bolts / Hex Head Bolts / ISO / ISO 4017.

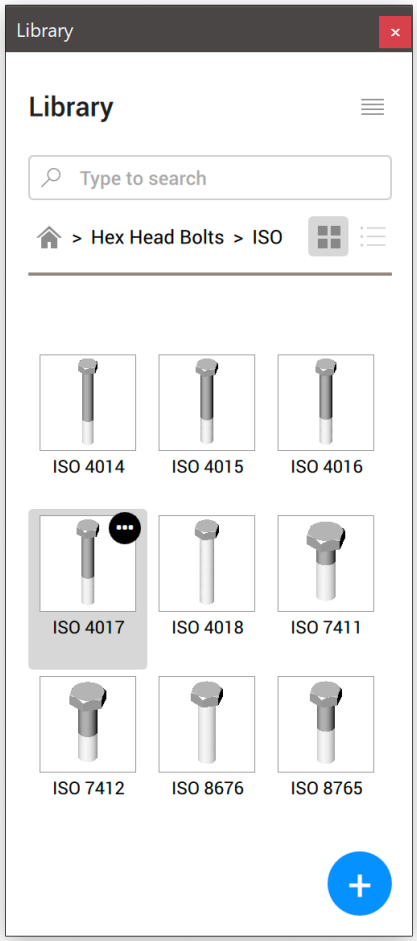


Figura 24: Biblioteca

Nota: Existem duas configurações específicas que você pode configurar para Peças Padrão (Standard) na caixa de diálogo *Configurações*:

- **Representação de rosca (THREADDISPLAY)** é usada para representar (ou não) uma rosca 3D para parafusos e porcas. Essa configuração permite diminuir o consumo de memória e aumentar o desempenho de grandes montagens
- **Número máximo de dentes da roda dentada (GEARTEETHNUMBER)** é usado para limitar o número de dentes para rodas dentadas (disponível em **Standard Parts /SPROCKET)**. Essa configuração permite diminuir o consumo de memória e aumentar o desempenho.

Por padrão, essas configurações são definidas como *Desligado*, a fim de otimizar o desempenho do modelo. Quando alteradas, as configurações se aplicam a todos os novos documentos.

Representação padrão de rosca nas vistas do desenho

Vistas do desenho de suas montagens, geradas com o comando *VISTABASE*, contêm anotações de rosca para peças padrão como linhas finas para vistas laterais, e como um arco fino de 270-graus para vistas superior/inferior.

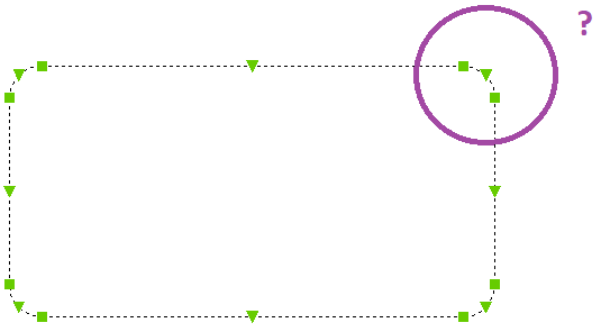


Figura 25: Exemplo

21.4.5.3 Furos

A categoria Furos contém uma biblioteca paramétrica de furos padrão, como fendas, rebaixado, escareado, etc.

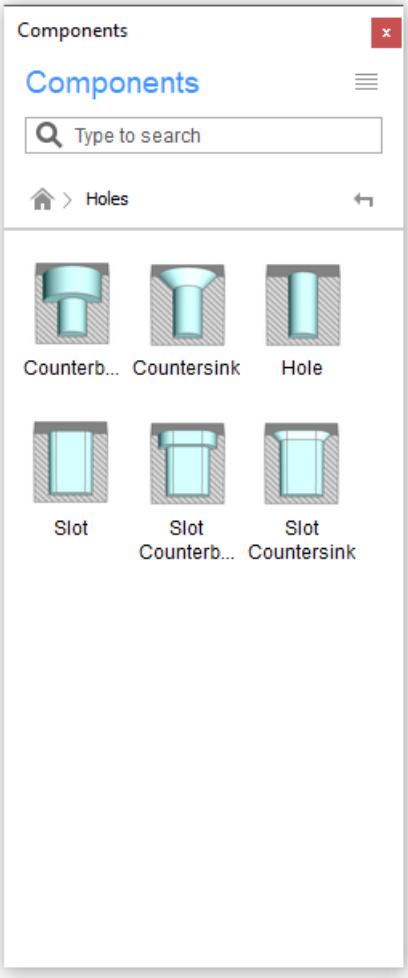


Figura 26: Furos

Há também uma subcategoria de furos roscados, representados em vistas do desenho.

21.4.5.4 Recursos de Forma para projetos em Sheet Metal

A categoria Sheet Metal contém uma biblioteca paramétrica de recursos de forma para sheet metal, como pontes, relevos, nervuras, estamparias, etc.

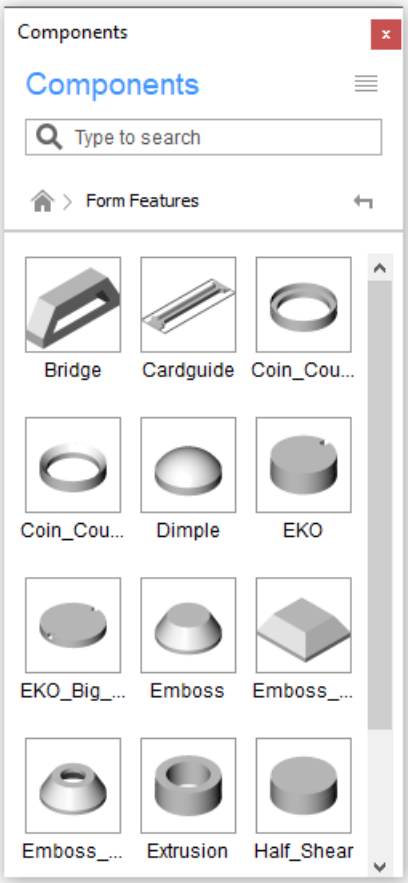


Figura 27: Recursos de Forma

21.4.5.5 Para inserir um Componente padrão no desenho atual:

1 Escolha uma categoria.

2 Escolha um padrão.

3 Selecione um tipo.

4 Clique no componente padrão desejado.

5 Edite os parâmetros no painel Propriedades.

6 Clique na área de desenho para inserir o componente selecionado.

21.4.5.6 Crie seu próprio componente padrão

Use o comando *BMCREATECOMPONENT* para criar suas próprias peças padrão e adicioná-las ao painel Componentes.

21.4.5.7 Controle a exibição de roscas

1 Abra a caixa de diálogo *Configurações*.

2 Encontre a variável de *Representação com rosca* (**THREADDISPLAY**).

3 Habilite a configuração.

4 Se aplicável, atualize o documento para que os segmentos apareçam em peças existentes:

- Defina temporariamente o **modo de atualização de componentes da Montagem(BMUPDATEMODE)** para **[1] Atualizar todos os componentes**.
- Use o comando *BMUPDATE* para atualizar o documento.
- Se aplicável, retorne o **Modo de atualização de componentes da montagem (BMUPDATEMODE)** para o valor anterior.

21.4.5.8 Controle o número de dentes da roda dentada

1 Abra a caixa de diálogo *Configurações*.

2 Encontre a variável *Número máximo de dentes da roda dentada* (**GEARTEETHNUMBER**).

3 Defina o valor a ser suficiente para as rodas dentadas usadas (*500* é recomendado).

Nota: Valores maiores retardam a atualização das rodas dentadas padrão. Se o valor for menor do que o número necessário, apenas esse número de dentes de rodas dentadas é exibido.

4 Se for aplicável, atualize o documento para que os dentes da roda dentada apareçam nas peças existentes:

- Defina temporariamente o **modo de atualização de componentes da Montagem(BMUPDATEMODE)** para **[1] Atualizar todos os componentes**.
- Use o comando *BMUPDATE* para atualizar o documento.
- Se aplicável, retorne o **Modo de atualização de componentes da montagem (BMUPDATEMODE)** para o valor anterior.

21.4.6 Tubulação no Projeto de Montagem

21.4.6.1 Visão Geral

Usar comandos: *BMINSERT*, *BMCONNECT*, e *BMBOM*.

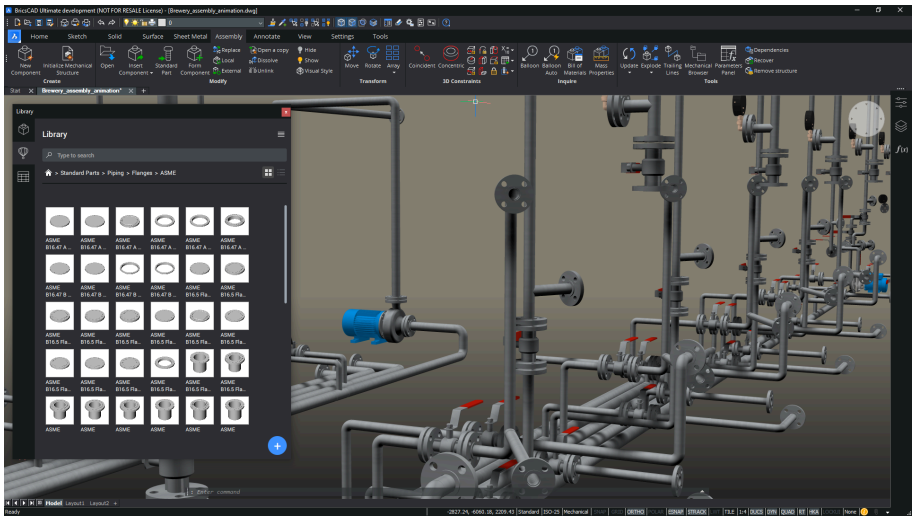


Figura 28: Tubulação no Projeto de Montagem

Tubos sem costura e soldados, cotovelos, tees, cruzetas, redutores e flanges são amplamente utilizados nas indústrias de maquinaria, automotiva, e aviação. Estes também são encontrados em petróleo e gás, indústria química, e abastecimento de água. O BricsCAD possui uma biblioteca de peças paramétricas de tubulação de diferentes padrões, como ASME, DIN e ISO. Essas peças padrão são paramétricas. Você pode ajustar facilmente seus parâmetros, vinculá-los a diferentes parâmetros globais, e usar restrições 3D para conectar corretamente as peças. O BricsCAD permite que você simplifique essas operações de rotina usando o **modo Smart** de comando *BMINSERT* e um comando dedicado *BMCONNECT*.

21.4.6.2 Inserir peças de tubulação em sua montagem

Arraste e solte qualquer peça padrão de tubulação do painel **Biblioteca** em sua montagem.

Por padrão, todas as partes da categoria Tubulação são inseridas no modo Inteligente (Smart mode), o que significa que, se você passar o mouse sobre uma peça de tubulação inserida anteriormente, o BricsCAD:

- Copia expressões de parâmetro daquela peça, para os parâmetros correspondentes da peça que está sendo inserida.
- Adiciona um conjunto de restrições 3D para conectar as peças.

Nota: Pressione a tecla **Ctrl** para trocar entre o modo Smart e o modo Regular, no qual as restrições 3D com outras peças não são geradas.

21.4.6.3 Criar manualmente restrições 3D entre peças padrão de tubulação

As peças standard (padrão) de tubulação têm um conjunto de entidades em arame (pontos e linhas) na camada STD_PARTS_PIPING_CONNECTION, que por padrão está Desligada.

Restringir uma conexão

Para restringir uma conexão (cotovelo, redutor, flange, ...) com outra conexão ou com um tubo, crie 2 restrições Coincidente 3D entre:

- 2 pontos das conexões, que deverão coincidir,

- 2 linhas das conexões, que devem estar colineares.

Restringir um tubo

Restringir tubos é mais complexo porque um tubo pode ter qualquer comprimento. Se um tubo já estiver restrito com uma conexão em um lado, utilizando 2 restrições Coincidente 3D , então restringi-lo a partir do outro lado deve ser diferente.

Você deve criar:

- Uma restrição Coincidente 3D entre a linha, que representa o eixo do tubo, e a linha correspondente da outra conexão
- Uma restrição de Distância 3D conduzida-por-geometria entre a face planar do tubo no lado oposto (já restrito) e o ponto correspondente da conexão, o qual está sendo conectado.

Você deve então digitar o nome da restrição de Distância no campo de expressão para o parâmetro Comprimento do tubo. Como as restrições conduzidas-por-geometria permitem controlar uma determinada dimensão, o tubo sempre terá o comprimento correto, e essa parametrização não limitará nenhuma edição adicional de sua montagem usando manipulação direta de restrições dimensionais adicionais não conduzidas-por-geometria.

21.4.6.4 Conectar automaticamente peças de tubulação

O conjunto de restrições é criado automaticamente utilizando o modo Smart do comando *BMINSERT* ou o comando dedicado *BMCONNECT* para conectar 2 peças padrão.

21.4.6.5 Criar uma Lista de Materiais (BOM) para uma montagem de tubulação

Em uma lista BOM você pode querer todos os tubos de mesmo tipo e tamanho (PartNumber), mas com comprimentos diferentes, estejam listados como a mesma peça.

Para fazer isso, use a função Agregar, do Gerenciador de BOM, para combinar todas essas peças em uma única linha, com seus comprimentos totais.

21.4.7 Matrizes de Componentes

21.4.7.1 Visão Geral

Usar os comandos *MATRIZ*, *MATRIZRETANG*, *MATRIZPOLAR*, *MATRIZCAMINHO*, e *BMINSERT*.

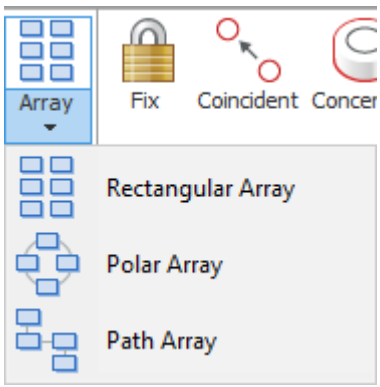
O BricsCAD permite que você insira o mesmo componente várias vezes de forma regular. Todas as inserções podem ser distribuídas uniformemente (à mesma distância) ao longo de uma linha, um círculo ou uma curva arbitrária. Padrões de ordem-superior (por exemplo: retangular ou cilíndrico) também são suportados. Isto é conseguido usando a tecnologia de matrizes associativas: retangular, polar, e por caminho.

Matrizes associativas são entidades persistentes, que podem ser editadas (manual ou automaticamente) a qualquer momento após sua criação - para alterar o número de elementos ou a distância entre esses. Todas as matrizes no seu modelo são listadas no Navegador de Mecânica, onde podem ser editadas.

Todos os componentes pode ser organizados em matriz - incluindo peças, sub-montagens, peças padrão, e furos.

21.4.7.2 Criar uma matriz de componentes

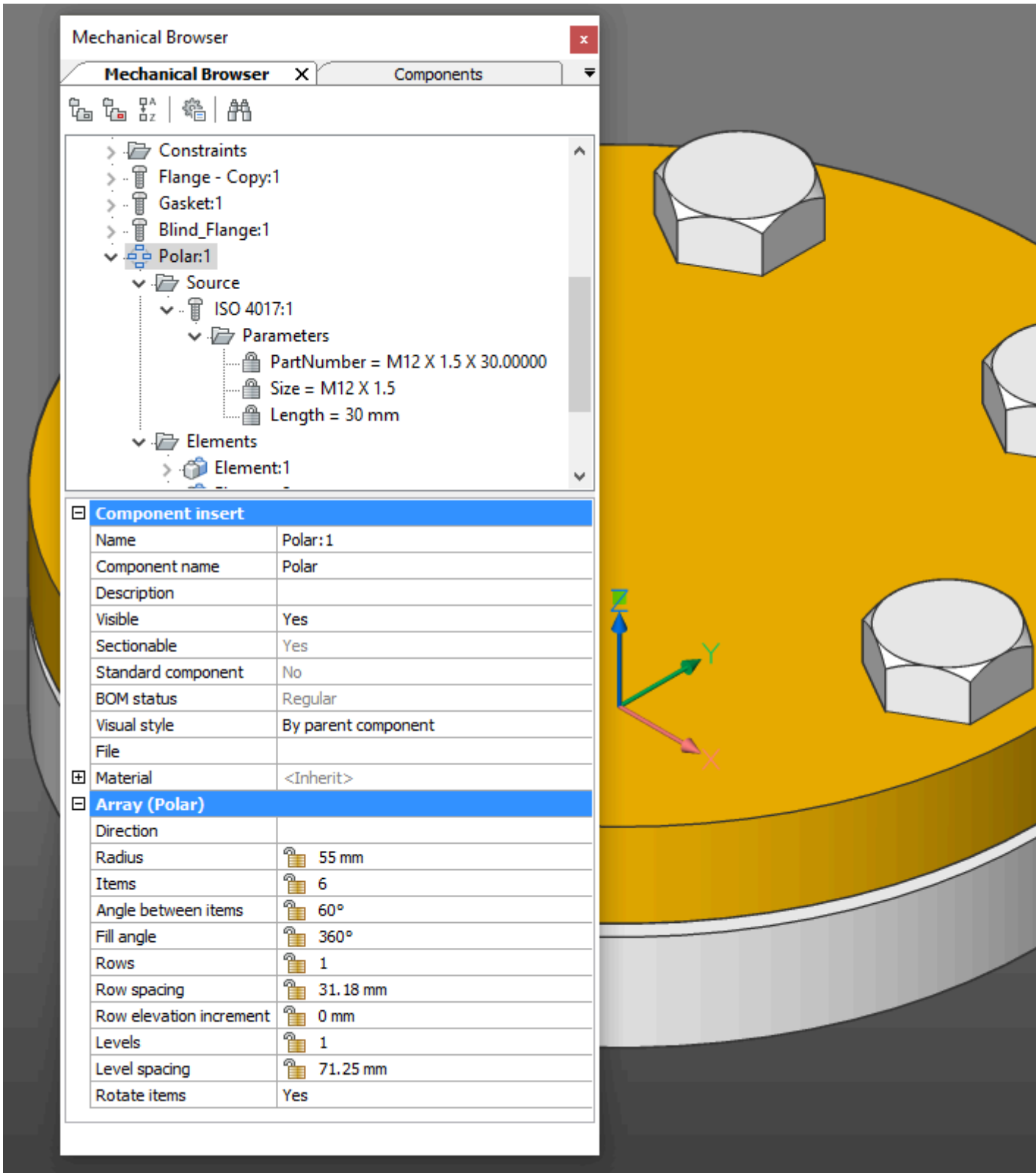
Use uma das ferramentas de matriz no painel **Montagem** do perfil Mecânica:



- **Matriz Retangular**
- **Matriz Polar**
- **Matriz Caminho**.

Outra maneira de criar uma matriz de componentes é usar a opção Múltplas/Matriz da ferramenta Inserir Componente .

21.4.7.3 Matrizes de componentes no Navegador de Mecânica



As matrizes de componentes são representadas como nós especiais no Navegador de Mecânica. Eles são nomeados após o tipo de matriz (Retangular, Polar ou Caminho) seguido de ponto e vírgula, e o índice da matriz (por exemplo, Retangular:1, Polar: 2, Caminho:3). Esses nós têm as mesmas propriedades que matrizes associativas regulares. As propriedades podem ser usadas para alterar o número de elementos na matriz, distância/ângulo entre estes, e outros parâmetros da matriz. Você também pode atribuir uma expressão persistente a qualquer propriedade. Ele será usado para recalcular automaticamente o valor da propriedade ao alterar seus parâmetros de condução.

Cada nó de matriz tem dois nós afiliados: Origem e Elementos.

- Sob o nó Origem, você pode encontrar o componente inserido, e seus parâmetros.
- O nó Elementos pode ser expandido para uma lista de elementos de matriz, com a possibilidade de ocultar ou alterar o estilo visual de um determinado elemento dentro da matriz.

21.4.7.4 Matrizes de componentes e Lista de Materiais

Matrizes de componentes são transparentes para a estrutura do produto. Componentes dentro de matrizes são considerados inseridos diretamente no mesmo conjunto. Portanto, em uma Lista de materiais (BOM) Top-level, componentes inseridos em matrizes de nível superior são listados como componentes de nível superior.

A coluna **Quantidade** na Lista BOM é preenchida com o número total de inserções do mesmo componente, incluindo componentes que estejam dentro de matrizes.

21.4.7.5 Matrizes de componentes paramétricos, peças padrão, e furos

Se um componente paramétrico for inserido como uma matriz, seus parâmetros ainda estarão disponíveis para edição e vinculação com parâmetros globais. Tecnicamente, os parâmetros estão associados ao elemento de origem de uma matriz. Quando esse elemento de origem é editado parametricamente, todos os outros elementos da matriz são alterados após este.

Peças Padrão e Furos são componentes paramétricos, e você pode editar seus parâmetros dentro das matrizes. Você pode facilmente criar uma peça com muitos furos e você pode controlar parametricamente o número do furo, posição, e seu tamanho, e ainda inserir um parafuso paramétrico. Essa poderosa tecnologia permite que você crie uma montagem de flange paramétrica com múltiplas configurações

21.4.8 Propriedades da Matriz

21.4.8.1 Visão Geral

Matriz de componentes é uma matriz associativa criada a partir de um componente paramétrico. As matrizes de componentes têm uma representação especial no **Navegador de Mecânica**.

Matriz de componentes tem dois sub-nós principais:

- Origem: nó que contém objeto de origem e seus parâmetros. O objeto de origem pode ser apresentado como um componente ou submontagem com propriedades específicas.
- Elementos: nó que contém todas as instâncias do objeto de origem

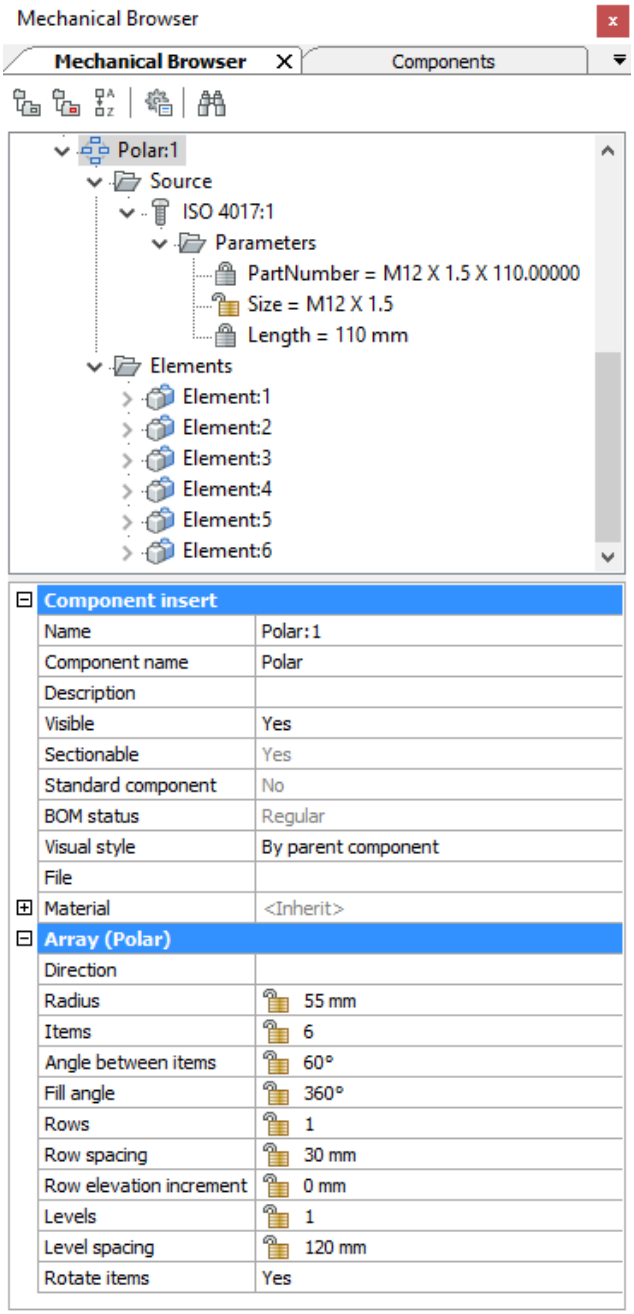
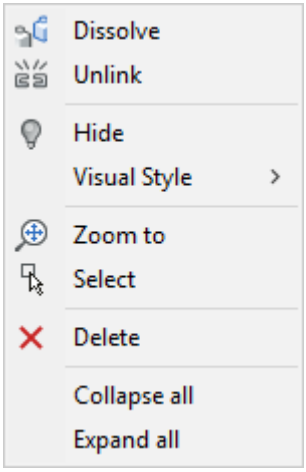
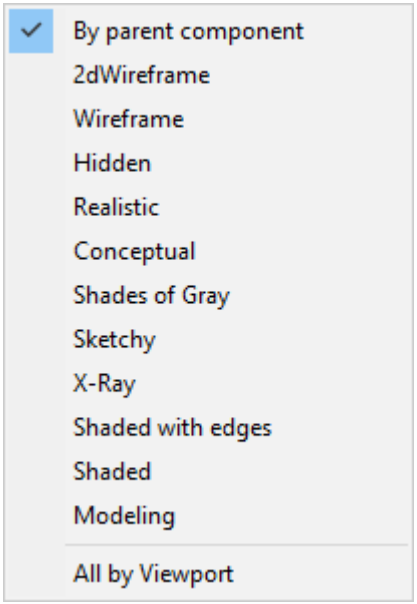


Figura 29: Navegador de Mecânica

O menu de contexto da matriz de Componentes é exibido clicando o botão-direito do mouse no nó da matriz enquanto as propriedades da matriz são exibidas automaticamente ao selecionar uma matriz no Navegador de Mecânica.

Tabela 6: Menu de Contexto

Menu de contexto	Descrição
Menu de contexto da matriz de componentes	Clique o botão-direito sobre uma matriz:  • **Dissolver**: Dissolve uma matriz criada no desenho atual (consulte o comando *BMDISSOLVE*). • **Desvincular**: Desvincula uma matriz de um sólido 3D, dissolvendo o recurso subjacente baseado-em-componente • **Ocultar** / **Mostrar** : Ocultar ou mostrar a matriz selecionada. • **Excluir todas as inserções do corte**: Define a *Seccionável* propriedade de todas as inserções semelhantes para *NÃO*. Defina se uma inserção é afetada pelo comando *VISTACORTE*. • **Estilo Visual**: exibe o menu **Estilo Visual**. Estilos visuais que estão salvos no desenho atual estão disponíveis.  • **Por Viewport**: renderize a matriz selecionada de acordo com o modo aplicar cor atual da viewport (veja o comando *MODOAPLICARCOR*). • **Tudo por Viewport**: Renderiza todas as matrizes de acordo com o modo de aplicar cor para a viewport atual (ver comando *MODOAPLICARCOR*). • **Zoom para**: faz Zoom no conjunto de seleção. • **Selecionar**: selecione a matriz. • **Excluir**: remove da montagem a matriz selecionada. • **Recolher tudo**: recolhe o componente principal e todos os componentes e subcomponentes. • **Expandir tudo**: expande o componente principal e todos os componentes e subcomponentes.

21.4.8.2 Comandos Relacionados

Tabela 7: Comandos Relacionados

Comando	Descrição
MATRIZ3D	Criar matrizes 3D.
MATRIZ	Cria uma matriz polar ou retangular de entidades; exibe avisos no barra de Comandos.
MATRIZCLASSICA	Cria uma matriz estática polar ou retangular de entidades, por meio de uma caixa de diálogo.
EDITARMATRIZ	Edita entidades de matriz associativa e suas entidades de origem.
MATRIZCAMINHO	Distribui associativamente cópias da entidade de forma uniforme, ao longo de um caminho, em várias filas e níveis.
MATRIZPOLAR	Distribui associativamente cópias de entidades, uniformemente em um padrão circular, ao redor de um ponto central ou eixo de rotação, usando múltiplas linhas e níveis.
MATRIZRETANG	Distribui associativamente cópias da entidade em qualquer número de linhas, colunas, e níveis.
INSERIRM	Insere matrizes de blocos.

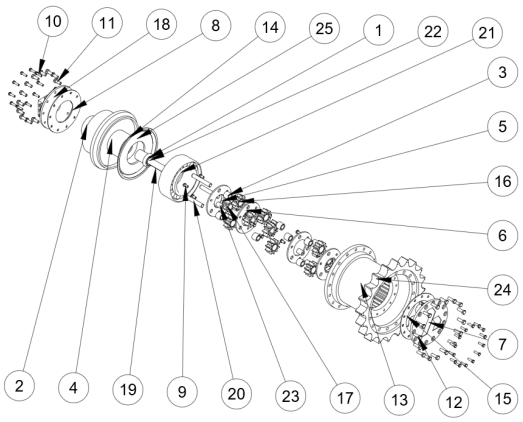
21.4.9 Representacoes Explodidas

21.4.9.1 Visão Geral

Use os comandos *BMEXPLODE*, *BMEXPLODEMOVE*, *BMTRAILINGLINES*, e *VISTABASE*.

A funcionalidade de representação explodida cria representações explodidas associativas de montagens, e não modifica a própria montagem. Representações explodidas são armazenadas em blocos dedicados, que podem ser editados e inseridos de acordo com suas necessidades. Você também pode gerar vistas de desenho a partir das representações explodidas.

Representações explodidas são associativas. Cada peça em uma representação explodida está ligada à peça correspondente na montagem. Use o comando *BMBALLOON* para atualizar representações explodidas para o estado atual da montagem, bem como colocar anotações de balão na(s) vista de desenho correspondente.



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

Figura 30: Exemplo de Representação Explodida

21.4.9.2 Níveis da representação explodida

Representações explodidas contêm referências aos componentes.

2 níveis de representações explodidas são suportados: Top (superior) e Bottom (inferior).

- **Top level** (nível-superior):

  A representação explodida contém referências apenas aos componentes de nível-superior. Os componentes de nível-superior são componentes inseridos diretamente no conjunto principal. Um componente top-level, em uma representação explodida, não é explodido em mais níveis - todas as suas peças e sub-montagens permanecem montados. Uma representação explodida Top-Level (nível-superior) permite que você visualize a composição básica de sua montagem.

- **Bottom-level (nível-inferior)**:

  A representação explodida contém referências apenas aos componentes de nível-inferior. Os componentes de nível-inferior são as peças terminais, que não contêm outras peças ou sub-montagens. As representações explodidas em Bottom Level (nível-inferior) permitem que você visualize todas as peças isoladamente.

Nota: Se **Bottom level** (nível-inferior) estiver selecionado, e houver alguns componentes de nível médio, contendo não apenas inserções de componentes, mas também outras entidades, essas entidades estarão perdidas na representação explodida, porque não é possível referenciá-las corretamente. Aquelas entidades devem ser convertidas primeiro nos correspondentes componentes de nível-inferior, bottom-level.

21.4.9.3 Algoritmos

O BricsCAD fornece 3 algoritmos para criar automaticamente representações explodidas e um algoritmo manual.

Tabela 8: Algoritmos

Algoritmo	Descrição
Tabela por Tipo	Crie uma explosão semelhante a uma tabela, onde componentes do mesmo tipo estão agrupados em linhas.
Linear	Encontre a sequência de desmontagem dos componentes em uma determinada direção e ordene os componentes. ***Nota:*** Isso considera possíveis colisões físicas entre componentes. Os componentes somente podem ser movidos pelo algoritmo, se não houver outros componentes (ainda não movidos) que os bloqueiam.
Automático	Encontre a sequência de desmontagem de componentes em relação à hierarquia de montagem. ***Nota:*** Isso considera todas as possíveis colisões físicas. Para cada peça ou submontagem, a direção do movimento é automaticamente identificada.
Manual	Cria uma cópia exata da montagem, pronta para edição personalizada. Neste modo, você pode criar uma representação explodida personalizada sem que haja alteração na montagem principal.

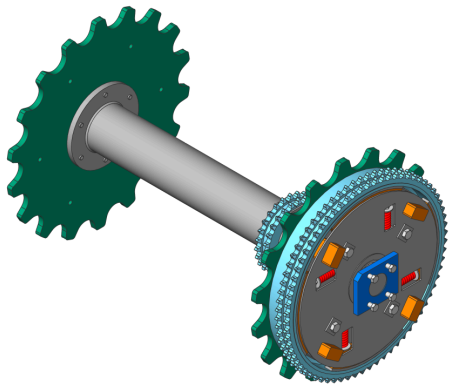
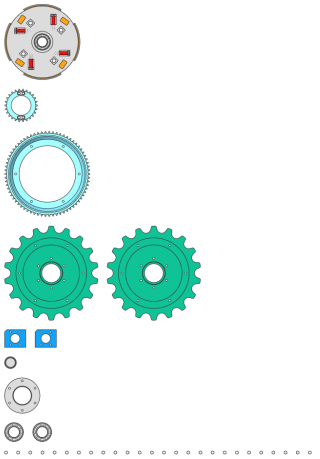
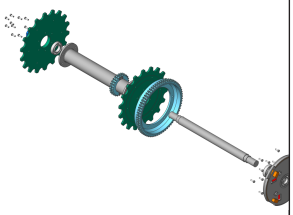
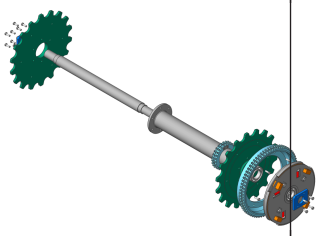
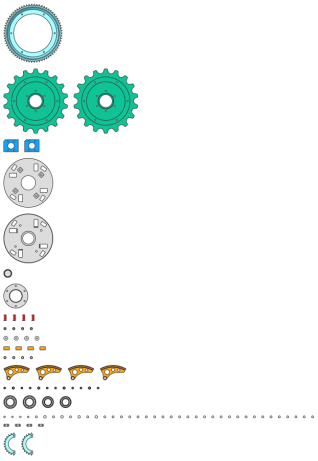
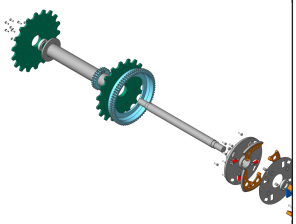
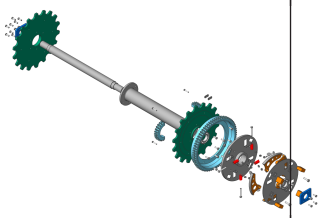


Figura 31: Exemplo: O conjunto do eixo de acionamento principal

Tabela 9: Exemplo: conjunto do eixo de acionamento principal

	Tabela por Tipo	Linear	Automático
Top level			
Bottom level			

21.4.9.4 Etapas e animações

Vistas explodidas podem conter uma ou várias etapas, que descrevem uma sequência específica de montagem ou desmontagem. Cada etapa corresponde a um conjunto de componentes que devem ser movidos nesta etapa em particular, para montar ou desmontar a montagem principal. Todos os algoritmos de representação explodidos criam automaticamente todas as etapas necessárias. Você também pode excluir, mesclar, dividir, mover ou adicionar etapas extras.

Cada etapa tem um nome único que pode ser usado para descrever essa etapa.

É possível animar uma determinada etapa, bem como toda a sequência de etapas, tanto em direções indiretas quanto inversas (para obter animações de desmontagem e montagem). Para que a animação funcione corretamente, uma etapa inicial que define o estado inicial da montagem precisa estar presente

na representação explodida. Use a propriedade **Auto ocultar** da representação explodida para ocultar automaticamente componentes que não são importantes para uma etapa específica durante a animação.

21.4.9.5 Editar representações explodidas

Cada representação explodida é armazenada como um bloco. Isso permite que você edite a representação explodida com o Editor de Blocos.

Importante: Não use o comando *EDITARREF* para editar representações explodidas. Este comando é incompatível com etapas de representação explodida.

Para abrir a representação explodida para edição:

1 Clique o botão-direito na representação explodida necessária no Navegador de Mecânica.

2 Selecione **Editar** no menu de contexto.

O editor de blocos abrirá automaticamente.

Dica: Como alternativa, você pode abrir a representação explodida para editar, no final do comando *BMEXPLODE*.

Quando a representação explodida está aberta para edição, você pode alterar o estado da etapa atual. Por padrão, esta é a última etapa feita pelo algoritmo. No entanto, você pode alterar esta de duas maneiras:

- Usar a propriedade **Etapa atual** da representação explodida no navegador.
- Clicar o botão-direito na etapa desejada no Navegador de Mecânica e selecionar **Aplicar** no menu de contexto.

Nota: Se você alterar a etapa atual, você vai descartar todas as modificações não salvas, bem como qualquer operação nas etapas. Certifique-se de que todas as modificações estejam salvas, antes de alterar a etapa atual, ou executar qualquer outra operação.

Use os comandos *DMMOVE*, *MOVER*, *DMROTATE*, e *BMEXPLODEMOVE* para mover componentes, dentro das representações explodidas, para as posições necessárias. O comando *BMEXPLODEMOVE* adiciona automaticamente todas as etapas necessárias após a etapa atual. Para outras operações, salve a etapa atual após a conclusão da edição. Somente a diferença entre a etapa atual e a anterior é armazenada.

Também é possível alterar a representação explodida executando operações em etapas.

Exemplo: se você remover uma etapa, todos os componentes movidos nesta etapa serão retornados às respectivas posições antes dessa etapa.

Salve a etapa atual depois de terminar a edição. Caso contrário, as alterações serão descartadas.

21.4.9.6 Linhas de arraste

Linhas de arraste ajudam a explicar a relação entre peças. Uma polilinha 3D representa uma trajetória, e exibe o movimento de uma peça durante o processo de desmontagem.

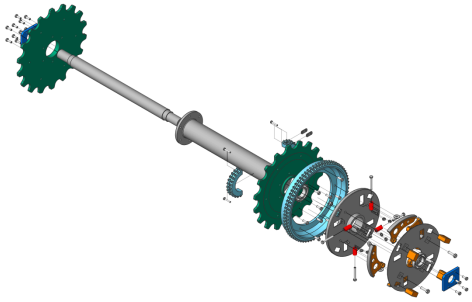


Figura 32: Exemplo de linhas de arraste

Use o comando *BMTRAILINGLINES* para adicionar linhas de arraste a uma representação explodida que estiver aberta para edição. Essa ferramenta cria todas as linhas de arraste. A informação vinda das etapas de vista explodida é usada para fazer isso.

Você pode gerar linhas de arraste para todas as peças, ou para um subconjunto de peças. Para um subconjunto, linhas de arraste serão construídas apenas para peças daquele subconjunto, embora apenas linhas de arraste necessárias sejam adicionadas. Algumas peças podem não ter nenhuma linha de arraste. Você também pode selecionar manualmente duas peças para adicionar uma linha de arraste entre essas.

Quando uma linha de arraste é computada, o algoritmo determina automaticamente o local da peça e o local de onde a peça foi removida, para calcular a trajetória. Este considera os movimentos de ambas as peças.

Cada linha de arraste é exibida no Navegador de Mecânica. Com o Navegador de Mecânica, você pode ver todas as linhas de arraste no modelo; destacar, selecionar e ampliar em zoom para as peças conectadas com a linha selecionada; renomear ou remover a linha de arraste.

Você pode editar as linhas de arraste com as ferramentas de edição padrão para polilinha 3D.

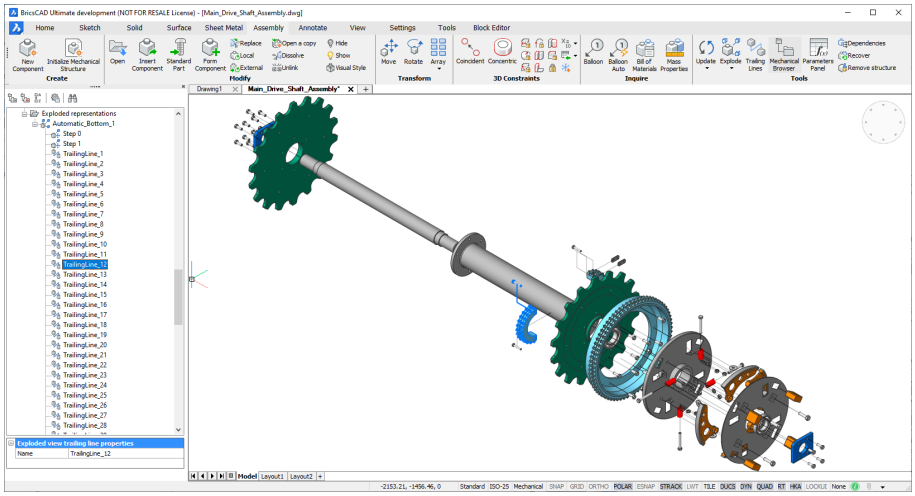


Figura 33: Propriedades de linhas de arraste na vista explodida

21.4.9.7 Cria uma representação explodida

Use o comando *BMEXPLODE* para criar uma representação explodida da montagem atual.

Nota: Você deve ter pelo menos uma inserção de uma peça na montagem, para que o comando funcione.

1 Escolha uma das seguintes ações para iniciar o comando *BMEXPLODE*:

- Clique no botão **Explodir**  no painel **Ferramentas** da aba **Montagem**.
- Clique no botão **Explodir**  na barra **Montagem**.
- Escolha **Explodir** no menu **Montagem**.
- Digite **BmExplode** na linha de Comando.

Você é solicitado: Selecionar algoritmo de explosão [Tabela por tipos/Linear/Manual/Configurações]:

2 Se for aplicável, defina o *Nível* e o *Nome* da representação explodida, usando a opção **Configurações**.

3 Selecione o algoritmo.

Você é solicitado: Selecione o comportamento da vista explodida [Editar/Gerar vistas do desenho/Finalizar]:

4 Execute uma das seguintes ações:

- Selecione **Editar** para editar a representação explodida.
- Selecione **Gerar vistas do desenho** para gerar vistas do desenho da representação explodida.
- Selecione **Finalizar**para concluir o comando.

21.4.9.8 Gerar vistas do desenho da representação explodida

1 Inicie o comando *VISTABASE*.

Você é solicitado: Selecionar objetos ou [Modelo inteiro/Presets/Vistas especiais] <Modelo inteiro>:

2 Selecione a opção **Vistas especiais**.

Você é solicitado: Selecione a vista [Vista explodida/Voltar] <Voltar>:

3 Selecione a opção **Vista explodida**.

Uma caixa de diálogo para selecionar a representação explodida é exibida.

4 Selecione a representação explodida.

Você é solicitado: Entre nome novo ou existente do layout para tornar o atual <Layout1>:

5 Se for aplicável, insira o *Nome do layout* do layout para a representação explodida.

6 Posicionar as vistas de desenho geradas da representação explodida no layout de Paper Space.

Nota: Para usar balões (veja o comando *BMBALLOON*) com vistas de desenho de representações explodidas, a *GENERATEASSOCVIEWS* (variável de sistema) deve estar *Ligada*.

21.4.9.9 Atualizar uma representação explodida

Execute o comando *BMUPDATE* para sincronizar representações explodidas com o estado atual da montagem.

Se for aplicável, ajuste a representação explodida. Esses ajustes podem ser necessários devido à adição e/ou exclusão de algumas peças.

Nota: A posição das peças em representações explodidas criadas nos modos **Manual** e **Linear** não são atualizadas automaticamente.

21.4.9.10 Gerenciar representações explodidas

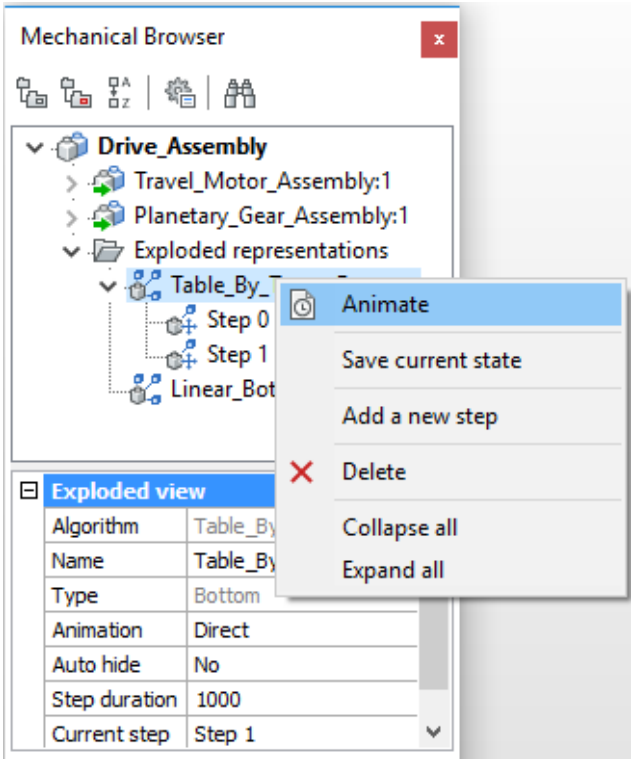


Figura 34: Gerenciar representações explodidas

Abra o Navegador de Mecânica para ver todas as representações explodidas no documento. Todas as representações explodidas serão listadas sob o grupo **Representações explodidas**. Abra o menu de contexto para as representações explodidas selecionadas, para ver todas as operações suportadas neste contexto.

Propriedades de representações explodidas

Cada representação explodida tem um conjunto de propriedades. Você pode editar algumas das propriedades

Tabela 10: Propriedades de representações explodidas

Propriedade	Descrição
Algoritmo	O nome do algoritmo usado para criar essa representação explodida.
Nome	Nome da representação explodida. Esse deve ser exclusivo para todas as representações explodidas, bem como para todos os blocos no modelo.
Tipo	Tipo de representação explodida: *Superior* ou *Inferior*.

Tabela 10: Propriedades de representações explodidas

Propriedade	Descrição
Animação	A direção da animação. Escolha 1 de 2 valores: *Direto* (do início ao final) ou *Inverso* (do final ao início).
Auto ocultar	Se for definido como *Sim*, todas as peças sem importância para uma determinada etapa serão ocultas durante a animação.
Duração da etapa	Duração padrão de cada etapa durante a animação (em milissegundos). O valor padrão é 1000, que é igual à duração 1s.
Etapa atual	A etapa atual que corresponde ao estado da representação explodida.

Operações em representações explodidas

As operações suportadas dependem da abertura ou não da representação explodida para edição.

Se a representação explodida **não estiver aberta para edição**, as seguintes operações são possíveis:

Tabela 11: Operações em representações explodidas

Operação	Descrição
Editar	Abre a representação explodida para edição. ***Dica:*** Você também pode editar a representação explodida com o comando *EDITAR-BLOCO*.
Excluir	Remova a representação explodida e o bloco associado do documento.

Se a representação explodida estiver **aberta para edição**, as seguintes operações são possíveis:

Tabela 12: Operações em representações explodidas

Operação	Descrição
Animar	Anime toda a sequência de etapas.
Salvar o estado atual	Salve as posições atuais das peças na etapa atual.

Tabela 12: Operações em representações explodidas

Operação	Descrição
Adicionar uma nova etapa	Adicione uma nova etapa após a última. ***Nota:*** A nova etapa será definida automaticamente como *atual*. Todas as modificações não salvas da representação explodida serão perdidas.
Excluir	Remova a representação explodida e o bloco associado do documento.

Além disso, todas as etapas serão listadas no Navegador de Mecânica para essa representação explodida.

21.4.9.11 Gerenciar etapas de representações explodidas

Abra o Navegador de Mecânica e abra a representação explodida necessária para edição. Todas as etapas serão listadas sob o nó correspondente, na árvore.

Propriedades das etapas

Tabela 13: Propriedades das etapas

Propriedade	Descrição
Nome	O nome da etapa. Deve ser único para esta representação explodida.
Duração, ms	A duração da etapa. Por padrão, isso é igual à Duração da etapa definida para a representação explodida.
Tipo de duração	Especifique se a duração deve ser retirada da representação explodida ou se ela é específica para essa etapa em particular.
Valor de duração (somente para o tipo de duração Absoluto)	A duração desta etapa em milissegundos.

Operações nas etapas

Tabela 14: Operações nas etapas

Operação	Descrição
Aplicar	Define a etapa como atual, e atualiza as peças na representação explodida de acordo com esta etapa.

Tabela 14: Operações nas etapas

Operação	Descrição
Animar	Anima a etapa selecionada.
Definir como anterior	Desmarca a etapa, e assim as posições de todas as peças serão as mesmas que na etapa anterior.
Mesclar com a anterior	Mescle esta e a etapa anterior. A etapa anterior é removida.
Dividir	Divide a etapa em uma sequência de etapas, cada uma delas corresponde exatamente a uma etapa.
Mover uma etapa acima	Mova a etapa uma posição para cima, na árvore.
Mover uma etapa abaixo	Mova a etapa uma posição para baixo, na árvore.
Excluir	Remova completamente a etapa, bem como os movimentos associados das peças. Depois disso, as posições das peças são atualizadas na representação explodida.

21.4.9.12 Editar representações explodidas

1 Selecione a representação explodida no Navegador de Mecânica.

2 Clique o botão-direito na representação explodida, e selecione **Editar** no menu de contexto.

Nota: Como alternativa, você pode executar o comando *EDITARBLOCO* e selecionar o bloco com a representação explodida.

3 Realize as operações necessárias para obter a representação explodida desejada.

4 Salve a etapa atual.

5 Salve as alterações usando o comando *FECHARBLOCO* com a opção *Salvar*.

Nota: Não use a opção *Descartar* do comando *FECHARBLOCO*, pois ele pode levar a um estado inconsistente do bloco e das etapas de representação explodidas. Usar *Desfazer* em vez disso.

21.4.9.13 Adicionar linhas de arraste

1 Abre a representação explodida para edição.

2 Execute o comando *BMTRAILINGLINES*.

3 Selecione o subconjunto de peças para o qual deve criar linhas de arraste, ou selecione o modelo inteiro.

4 Selecione os pontos das peças a ser usadas, para criar a trajetória: origem ou centro.

5 Se aplicável, edite as linhas de arraste resultantes.

6 Salvar mudanças.

21.4.10 Gerenciador de Lista de Materiais (BOM)

21.4.10.1 Visão Geral

Usar os comandos *BMBOMPANELOPEN*, *BMBOMPANELCLOSE* e *BMBOM*.

O Gerenciador da Lista de Materiais permite que você procure, crie e edite listas de materiais (BOMs) no desenho atual. Use o gerenciador para configurar interativamente listas de materiais, tanto as novas como as existentes. Você pode salvar a configuração de uma lista de materiais como um arquivo template e reutilizá-la posteriormente em outros desenhos. Você também pode adicionar uma lista de materiais que não está vinculada a uma tabela no desenho.

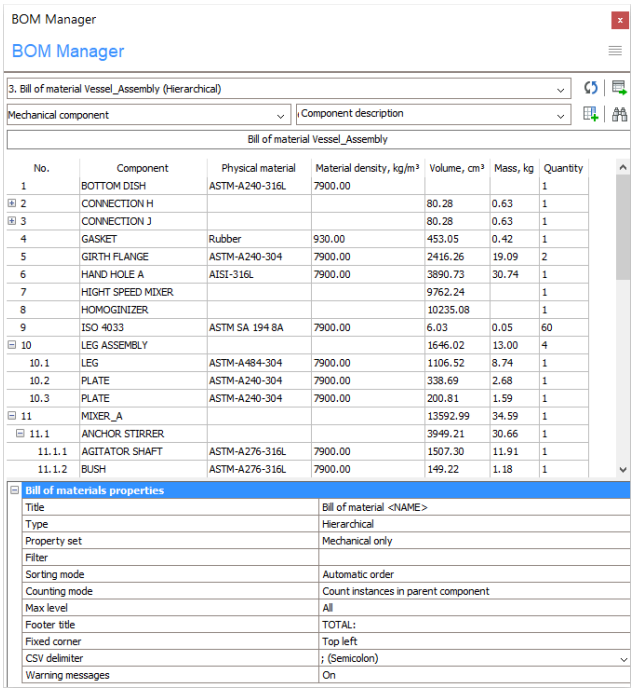


Figura 35: Gerenciador de BOM

Com o Gerenciador da Lista de Materiais, você pode:

- Procurar listas de materiais existentes no desenho.
- Criar uma nova lista de materiais usando tanto um tipo padrão (como top-level, bottom-level ou hierárquico) como usar um modelo de lista BOM existente.
- Editar listas de materiais:
  - Adicionar ou remover colunas (inclui colunas de fórmula).
  - Mover colunas selecionadas para novas posições.
  - Renomear as colunas.
  - Configurar como a tabela da BOM é ordenada.

  - Configurar o valor do rodapé para cada coluna individual.
  - Configurar as unidades e como elas são exibidas para cada coluna.
  - Configurar a largura de cada coluna individual usando as propriedades da coluna.
  - Configurar a visibilidade de cada coluna individual.
  - Filtrar os componentes com uma expressão de filtro.
  - Alterar o título das linhas de título e de rodapé.
- Pesquisar por componentes na tabela.
- Destacar e selecionar instâncias de componentes.
- Salvar a configuração da BOM como um arquivo template.
- Carregar uma configuração de tabela BOM a partir de um arquivo template.
- Colocar a tabela BOM em um desenho.
- Excluir tabelas BOM.

Uma licença Mecânica ou Ultimate é necessária para modificar ou criar listas de materiais. No entanto, o gerenciador está disponível com uma licença Pro.

21.4.10.2 Abrir o Gerenciador de Lista de Materiais

Siga um dos procedimentos:

- Posicione o cursor sobre uma barra, clique o botão-direito, escolha Paineis, e a seguir escolha **Gerenciador de BOM** no sub-menu.
- Inicie o comando *BMBOMPANELOPEN*.

21.4.10.3 Ferram

A maioria das operações de configuração no gerenciador estão disponíveis no menu principal. Um menu de contexto está disponível para colunas ou linhas selecionadas. Você também pode alterar as propriedades das colunas selecionadas e da própria BOM, na seção de propriedades do gerenciador. Para trabalhar com uma coluna específica, selecione-a na grade BOM. Várias colunas podem ser selecionadas simultaneamente.

Menu principal

Clique em **Menu** (☰) no canto superior direito do painel Gerenciador de BOM.

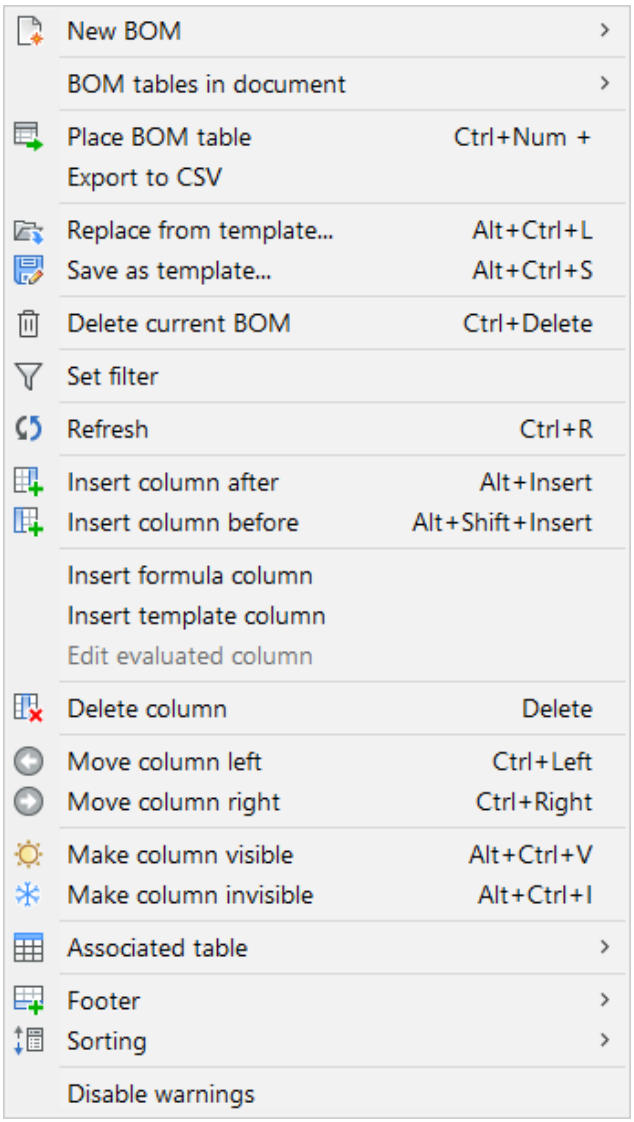


Figura 36: Menu principal

Tabela 15: Menu principal

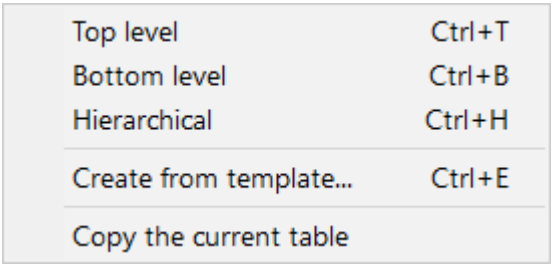
Ícone	Ferramenta	Descrição
	**Nova Lista BOM**	Abre um submenu:  **Figura 37:** Sub-menu • **Top-level**: cria uma nova lista BOM, top-level. • **Bottom-level**: cria uma nova lista BOM, bottom-level. • **Hierárquica**: cria uma nova lista BOM, hierárquica. • **Criar a partir do template...**: abre uma caixa de diálogo para selecionar um arquivo template para configuração da BOM. • **Copiar a tabela atual**: cria uma nova BOM com base na tabela atual.
	Tabelas BOM no documento	Abre um submenu:  **Figura 38:** Sub-menu • **Ir para a primeira tabela**: muda para a primeira tabela BOM no documento. • **Ir para a tabela anterior**: muda para a tabela BOM anterior no documento. • **Ir para a próxima tabela**: muda para a tabela BOM seguinte no documento. • **Ir para a última tabela**: muda para a última tabela BOM no documento. • **<Nome da BOM>**: muda para a tabela BOM correspondente no documento.

Tabela 15: Menu principal

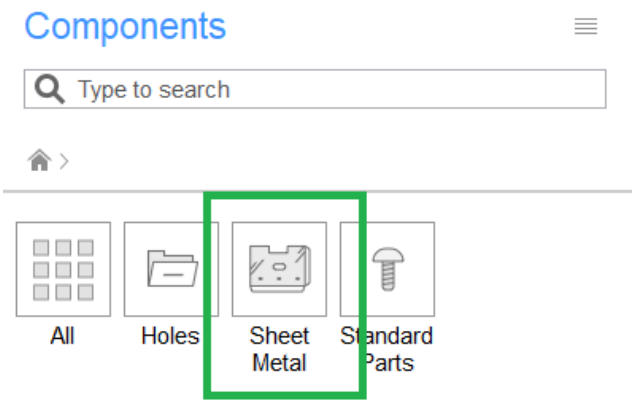
Ícone	Ferramenta	Descrição
	Inserir/atualizar tabela BOM	Inicia o posicionamento interativo de uma tabela no desenho. Se a tabela já estiver colocada no desenho, esta vai atualizar a tabela. ***Nota:*** Todas as operações de edição atualizam automaticamente a tabela associada no desenho.
	Exportar para CSV	Salva o conteúdo da tabela atual em um arquivo CSV. Abre uma caixa de diálogo para selecionar o local para o arquivo CSV:  **Figura 39:** Entrar o nome do arquivo para exportar a tabela BOM ***Nota:*** Você pode ajustar o separador para valores CSV usando as propriedades da tabela.
	Substituir a partir do template	Substitui a configuração da tabela BOM atual pelo template selecionado. Abre uma caixa de diálogo para selecionar o arquivo a usar como template.
	Salvar como template	Salva a configuração da tabela BOM atual como um template. Abre uma caixa de diálogo para selecionar o local para armazenar o novo arquivo template.
	Excluir BOM atual	Exclui a atual tabela de BOM do documento.

Tabela 15: Menu principal

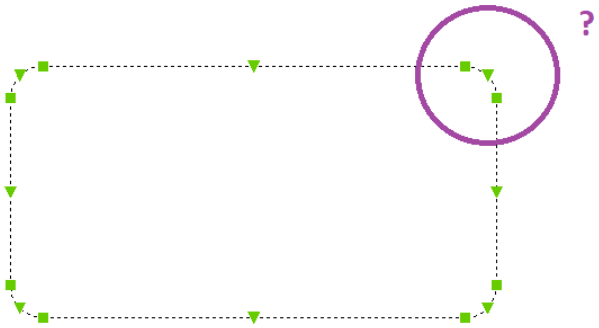
Ícone	Ferramenta	Descrição
	Definir filtro	Abre uma caixa de diálogo para configurar o filtro principal:  **Figura 40:** Configurador de Filtros
	Atualizar	Atualiza a atual lista BOM, e lista todas as tabelas de BOM no desenho.
	Inserir coluna após	Se uma coluna for selecionada: insere uma coluna a partir do **Seletor de coluna** após a coluna selecionada na tabela. Se nenhuma coluna for selecionada: adiciona uma nova coluna depois da última coluna na tabela.
	Inserir coluna antes	Se uma coluna for selecionada: insere uma coluna a partir do **Seletor de coluna** antes da coluna selecionada na tabela. Se nenhuma coluna for selecionada: adiciona uma nova coluna como a primeira coluna na tabela.

Tabela 15: Menu principal

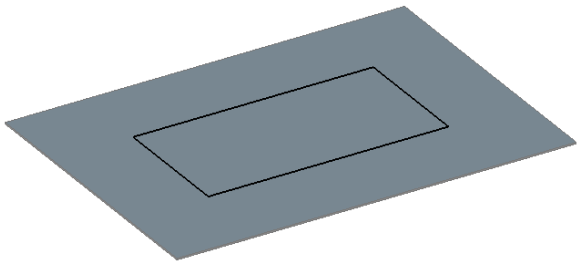
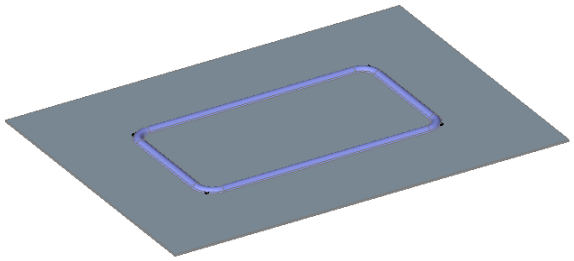
Ícone	Ferramenta	Descrição
	Adicionar uma coluna de fórmula	Abre uma caixa de diálogo para configurar uma coluna de fórmula:  **Figura 41:** Coluna de fórmula Clique em **OK** para adicionar uma nova coluna de fórmula após a selecionada. Se nenhuma coluna for selecionada, uma nova coluna de fórmula será adicionada depois da última coluna na tabela.
	Inserir coluna de template	Abre uma caixa de diálogo para configurar uma coluna do template:  **Figura 42:** Coluna de template Clique em **OK** para adicionar uma nova coluna de template após aquela selecionada. Se nenhuma coluna for selecionada, uma nova coluna de template será adicionada depois da última coluna na tabela.
	Editar coluna avaliada	Abre uma caixa de diálogo para editar a fórmula, ou coluna de template, selecionada.

Tabela 15: Menu principal

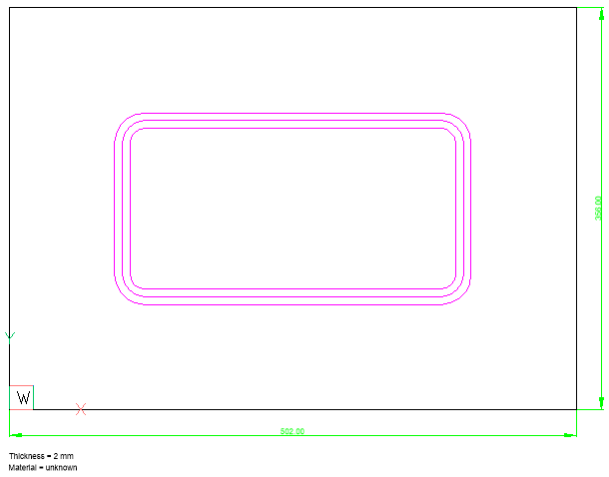
Ícone	Ferramenta	Descrição
	Excluir coluna	Remove a coluna selecionada da tabela. Você também pode selecionar uma coluna na **Grade BOM** e pressionar a tecla **Delete** para excluir uma coluna.
	Mover coluna p/ esquerda	Move a(s) coluna selecionada, uma coluna para a esquerda.
	Mover coluna p/ direita	Move a(s) coluna selecionada, uma coluna para a direita.
	Tornar coluna visível	Exibe a coluna selecionada, na tabela associada no desenho.
	Tornar coluna invisível	Remove a coluna selecionada da tabela associada. Colunas invisíveis permanecem visíveis no gerenciador de BOM, mas são marcadas com um padrão de hachura diagonal em segundo plano.
	Tabela associada	Abre um submenu:  **Figura 43:** Sub-menu • **Zoom para**: amplia a tabela. Se necessário, define o layout da tabela como o atual. • **Selecionar**: seleciona a tabela. Se necessário, define o layout da tabela como o atual. • **Excluir**: exclui a tabela, desse documento. • **Desanexar**: desconecta a tabela. A tabela no documento não será mais atualizada, quando a tabela BOM for modificada.

Tabela 15: Menu principal

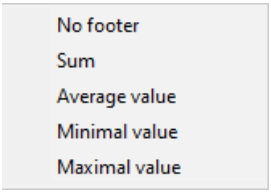
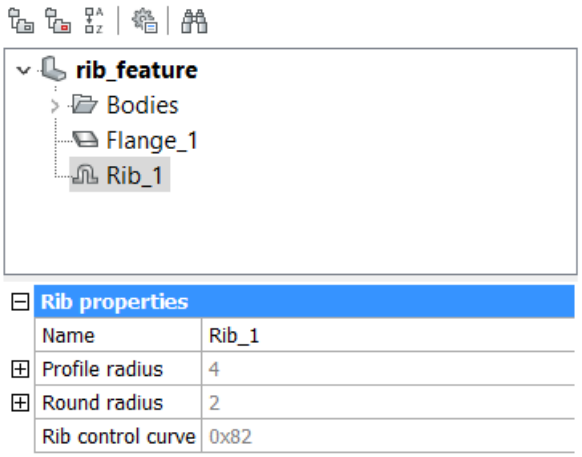
Ícone	Ferramenta	Descrição
	**Rodapé**	Abre um submenu:  **Figura 44:** Sub-menu Para cada uma das colunas selecionadas: • **Sem rodapé**: remove o rodapé. • **Soma**: exibe a soma no rodapé. • **Valor da média**: exibe a média no rodapé. • **Valor mínimo**: exibe o valor mínimo no rodapé. • **Valor máximo**: exibe o valor máximo no rodapé. Para obter mais detalhes, consulte a seção Rodapé em Lista de Materiais na página 1635.
	**Ordenar**	Abre um submenu:  **Figura 45:** Sub-menu • **Ordem automática**: ordena a tabela automaticamente. • **Ordem natural**: classifica a tabela pela ordem natural das instâncias de componentes no documento. • **Ordem ascendente**: define a ordem crescente para as colunas selecionadas. • **Ordem decrescente**: define a ordem decrescente para as colunas selecionadas. Para obter mais detalhes, consulte a seção Classificação em Lista de Materiais na página 1635.

Tabela 15: Menu principal

Ícone	Ferramenta	Descrição
	Desativar/Ativar avisos	Controla a exibição de mensagens de aviso durante a criação ou atualização de tabelas BOM.

Barras

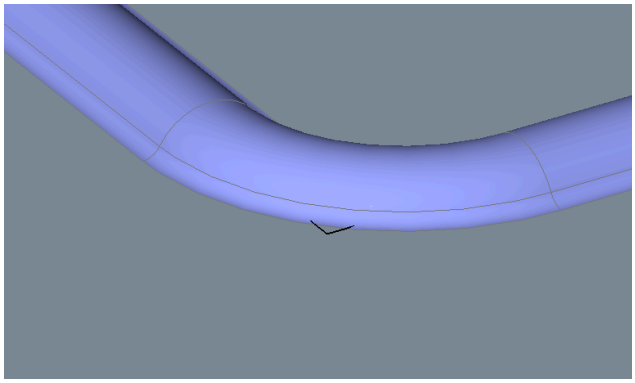


Figura 46: Gerenciador de BOM

Tabela 16: Gerenciador de BOM

Ícone	Ferramenta	Descrição
	Menu principal	Abre o menu principal.
	Seletor de Listas BOM	Seleciona a tabela BOM atual. A seleção é armazenada no desenho. Quando o desenho é fechado e reaberto, a BOM selecionada é restaurada no gerenciador. Quando selecionada, a tabela é atualizada para o estado atual do desenho.
	Atualizar	Atualiza a Lista BOM atual, e a lista de todas as tabelas BOM existentes no desenho.
	Colocar/atualizar tabela	Inicia o posicionamento interativo de uma tabela num desenho. Atualiza a tabela, se essa já existir. ***Nota:*** Todas as operações de edição atualizam automaticamente a tabela associada no desenho.

Tabela 16: Gerenciador de BOM

Ícone	Ferramenta	Descrição
	Seletor de categoria	Seleciona uma categoria de propriedades.
	**Seletor de coluna**	Seleciona a coluna a ser adicionada à lista BOM atual. Você também pode usar isso para inserir uma fórmula para um campo de fórmula. Para adicionar uma fórmula, digite uma expressão começando com "=", como "=<Massa>*<Quantidade>". A fórmula pode se referir a outras colunas. Use seus nomes de exibição incluídos entre chaves <>.
	**Inserir coluna após**	Se uma coluna for selecionada: insere a coluna a partir do **Seletor de coluna** após a coluna selecionada, na tabela. Se nenhuma coluna for selecionada: adiciona a nova coluna depois da última coluna na tabela.
	**Localizar**	Ativa e desativa o campo de pesquisa. Você também pode usar **Ctrl+F**.
	**Campo de pesquisa**	Digite uma string de pesquisa; curingas são suportados.
	**Localizar anterior**	Seleciona a correspondência anterior na tabela BOM. Você também pode usar **Ctrl+F3**.
	**Localizar próxima**	Seleciona a próxima correspondência na tabela BOM. Você também pode usar **F3**.
	**Selecionar todos os encontrados**	Seleciona todas as correspondências na tabela BOM.

Menu de contexto da coluna

Clique o botão-direito em um rótulo de coluna, para abrir o menu de contexto. A coluna é adicionada à seleção atual.

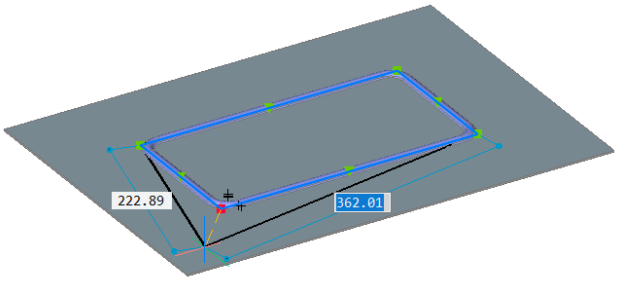


Figura 47: Menu de contexto da coluna

Tabela 17: Menu de contexto da coluna

Ícone	Ferramenta	Descrição
	Adicionar coluna template a partir das selecionadas	Abre uma caixa de diálogo para configurar uma nova coluna de template. As colunas selecionadas serão adicionadas automaticamente à expressão do template.
	Excluir coluna	Remove a(s) coluna selecionada da tabela. Você também pode selecionar uma coluna na **grade BOM** e pressionar a tecla **Delete**.
	Rodapé	Abre um submenu para a(s) coluna selecionada: • **Sem rodapé**: remove o rodapé. • **Soma**: exibe a soma no rodapé. • **Valor da média**: exibe a média dos valores no rodapé. • **Valor mínimo**: exibe o valor mínimo no rodapé. • **Valor máximo**: exibe o valor máximo no rodapé.
	Ordenar	Abre um submenu: • **Ordem automática**: classifica a tabela automaticamente**.** • **Ordem natural**: classifica a tabela pela ordem natural das instâncias de componentes no documento. • **Ordem ascendente**: define a ordem crescente para as colunas selecionadas. • **Ordem decrescente**: define a ordem decrescente para as colunas selecionadas.

Tabela 17: Menu de contexto da coluna

Ícone	Ferramenta	Descrição
	Função de agregar	Abre um submenu para a(s) coluna selecionada: • **Nenhuma função**: remove a função agregada. • **Soma**: exibe a soma de todos os valores no grupo. • **Valor da média**: exibe a média de todos os valores no grupo. • **Valor mínimo**: exibe o menor de todos os valores no grupo. • **Valor máximo**: exibe o maior de todos os valores no grupo. • **Concatenação**: exibe uma concatenação de todos os valores do grupo em ordem crescente, separados por um delimitador especificado (';' por padrão) • **Concatenação com contagem**: exibe uma concatenação de todos os valores no grupo com o número correspondente de encontros em ordem crescente. Os pares de valor e contagem são separados por um delimitador especificado (';' por padrão)
	Função de coluna	Abre um submenu para a coluna selecionada: • **Número**: a coluna é usada como fonte de números pelo comando *BMBALLOON*. • **Nome**: a coluna é usada como fonte de nomes pelo comando *BMBALLOON*. • **Quantidade**: a coluna é usada como fonte de quantidades pelo comando *BMBALLOON*. • **Regular**: a coluna não tem função específica.

Tabela 17: Menu de contexto da coluna

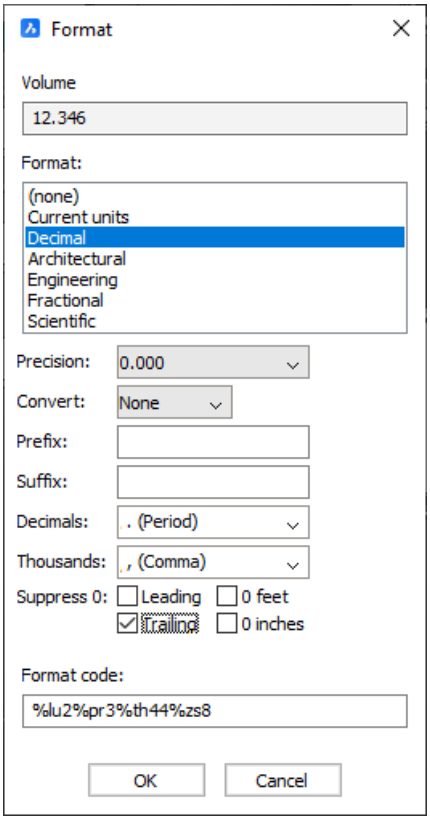
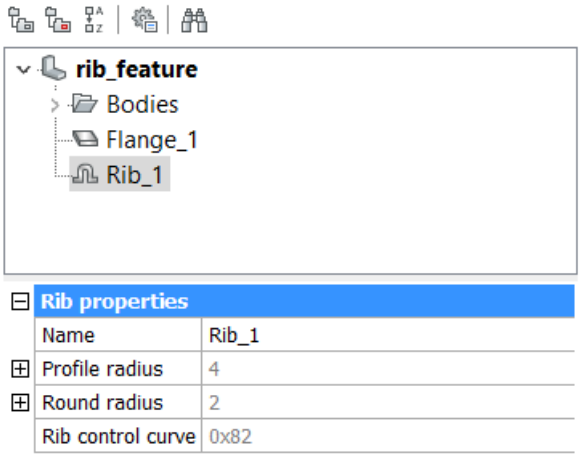
Ícone	Ferramenta	Descrição
%..	**Formato**	Define o formato especificado para a(s) coluna selecionada. Abrir uma caixa de diálogo:  **Figura 48:** Formato

Tabela 17: Menu de contexto da coluna

Ícone	Ferramenta	Descrição
	Definir filtro de coluna	Abre uma caixa de diálogo para configurar o filtro de coluna:  **Figura 49:** Filtro de Coluna
	Tornar coluna visível	Mostra a(s) coluna selecionada na tabela associada, no desenho.
	Tornar coluna invisível	Remove a coluna selecionada da tabela associada. Colunas invisíveis permanecem visíveis no gerenciador de BOM, mas são marcadas com um padrão de hachura diagonal em segundo plano.
	Ajustar coluna(s)	Ajusta a largura da(s) coluna selecionada ao seu conteúdo. A largura da coluna ajusta automaticamente o conteúdo, a menos que a largura tenha sido alterada manualmente.
	Ajustar todas colunas	Ajusta a largura de todas as colunas ao seu conteúdo. A largura da coluna ajusta automaticamente o conteúdo, a menos que a largura tenha sido alterada manualmente.

Menu de contexto da linha

Clique o botão-direito do mouse em qualquer célula para abrir o menu de contexto.

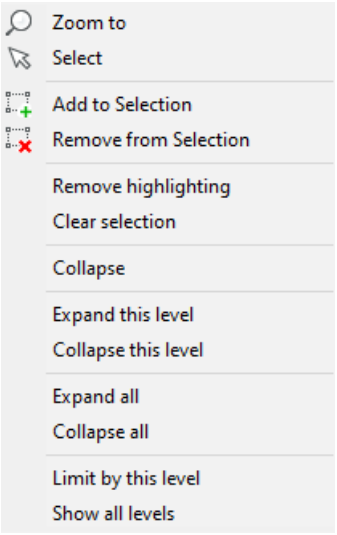


Figura 50: Menu de contexto

Tabela 18: Menu de contexto da linha

Ícone	Ferramenta	Descrição
	Zoom para	Amplia a(s) peça selecionada, no modelo
	Selecionar	Seleciona a(s) peça. Substitui uma seleção existente.
	Adicionar à Seleção	Adiciona peça(s) à seleção.
	Remover da Seleção	Remove as peças da seleção.
	Remover destaque	Remove os itens realçados.
	Limpar seleção	Limpa uma seleção.
	Recolher	Recolhe o grupo de sub-montagem na grade da BOM.
	Expandir	Expande o grupo de sub-montagem na grade da BOM.
	Expandir este nível	Expande todos os grupos de sub-montagens no mesmo nível da árvore da montagem, como o atual.
	Recolher este nível	Recolhe todos os grupos de sub-montagens no mesmo nível da árvore de montagem, como o atual.
	Expandir tudo	Expande todos os grupos de sub-montagens na lista BOM atual.

Tabela 18: Menu de contexto da linha

Ícone	Ferramenta	Descrição
	Recolher tudo	Recolhe todos os grupos de sub-montagens na lista BOM atual.
	Limitar por este nível	Define o nível atual como o nível máximo para tabelas hierárquicas de BOM.
	Mostrar todos os níveis	Remove o limite de nível máximo para tabelas hierárquicas de BOM.

Título da BOM



Figura 51: Título da BOM

Exibe o título da lista BOM atual, conforme exibido no desenho. Clique o botão esquerdo sobre este controle para editar o desenho. Neste modo, o controle de **Título BOM** mostra o **template do título**, em vez disso. O **template do título** é uma string que pode conter um <*NOME*> espaço reservado, que será substituído pelo nome da montagem.

Grade

No.	Component ▲₂	Physical material	Material density	Is standard	Quantity ▼₁
1	ISO 4033			Yes	60
2	MM-THD			Yes	30
3	LEG			No	4
4	FLANGE			No	2
5	CONNECTION H	ASTM-A276-316L	7900.00	No	1
6	CONNECTION J	ASTM-A276-316L	7900.00	No	1
7	GASKET			No	1
8	HAND HOLE			No	1
9	HIGHT SPEED MIXER			No	1
10	HOMOGINIZER			No	1
11	MIXER_A			No	1
12	NOZZLE B			No	1
13	NOZZLE D			No	1
14	NOZZLE F	AISI-316L	7900.00	No	1
15	NOZZLE L			No	1

Figura 52: Grade

A grade exibe o conteúdo da tabela BOM selecionada. Cada linha é vinculada com suas instâncias correspondentes. Quando uma linha é selecionada, as peças correspondentes são destacadas no desenho. Use o menu de contexto, para selecionar peças correspondentes, adicioná-las ou removê-las da seleção atual, e para ampliar em zoom essas peças. Clique duas vezes em qualquer célula na linha para ampliar em zoom as peças correspondentes.

Movendo uma coluna

Para mover uma coluna na tabela BOM, arraste e solte-a em sua nova posição.

Colunas visíveis e invisíveis



Figura 53: Colunas visíveis e invisíveis

Você pode tornar invisível qualquer coluna na tabela BOM, para que ela não seja exibida na tabela resultante. Colunas invisíveis permanecem visíveis na grade BOM com um plano de fundo hachurado. Você pode alterar a visibilidade das colunas no menu principal, no menu de contexto da coluna, ou editando a propriedade **Visível** de uma coluna na **grade Propriedades**.

Ajuste de largura

Você pode ajustar a largura de qualquer coluna na grade BOM, que é independente da largura na tabela associada. Se a largura da coluna for menor que o necessário para exibir o texto da célula, a altura da linha correspondente aumentará, para que caiba o texto.

Quando a largura da coluna é ajustada manualmente, a largura permanece a mesma, independentemente do conteúdo da tabela. A largura também é armazenada na configuração da BOM. Todas as larguras de coluna ajustadas permanecem as mesmas mesmo quando o desenho é fechado e reaberto.

Para ajustar automaticamente a largura da coluna ao conteúdo, siga um destes procedimentos:

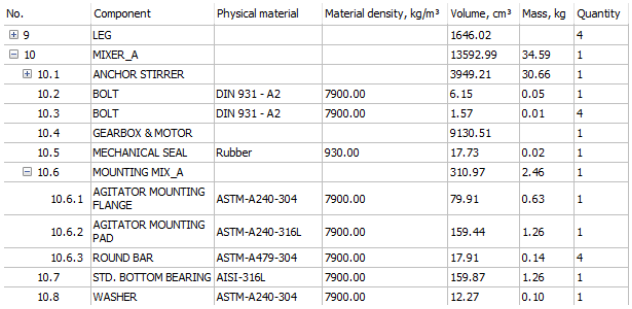
- Selecione a(s) coluna, clique o botão-direito, e no menu de contexto selecione **Ajustar coluna(s)**.
- Clique-duplo no separador, no cabeçalho da coluna.

Ordenando uma BOM

Use a grade para controlar como a tabela de conteúdo deve ser ordenada. Clique duas vezes em um rótulo de coluna para alterar a ordem entre ascendente, descendente e não ordenada. Se uma coluna for usada para ordenar a tabela, seu rótulo tem um símbolo adicionado (▲ para ascendente e ? para decrescente). Clique duas vezes no rótulo para reverter a ordem de classificação.

Você pode usar várias colunas para classificar. Os símbolos de classificação dessas colunas têm um subscrito para denotar a ordem de classificação. Por exemplo, se você classificar por quantidade em ordem decrescente, então pelo nome do componente em ordem crescente, a coluna Quantidade tem um símbolo ?? e a coluna Componente tem um símbolo ▲?.

Expandir/recolher grupos



No.	Component	Physical material	Material density, kg/m³	Volume, cm³	Mass, kg	Quantity
⊞ 9	LEG			1646.02		4
⊟ 10	MIXER_A			13592.99	34.59	1
⊞ 10.1	ANCHOR STIRRER			3949.21	30.66	1
10.2	BOLT	DIN 931 - A2	7900.00	6.15	0.05	1
10.3	BOLT	DIN 931 - A2	7900.00	1.57	0.01	4
10.4	GEARBOX & MOTOR			9130.51		1
10.5	MECHANICAL SEAL	Rubber	930.00	17.73	0.02	1
⊟ 10.6	MOUNTING MIX_A			310.97	2.46	1
10.6.1	AGITATOR MOUNTING FLANGE	ASTM-A240-304	7900.00	79.91	0.63	1
10.6.2	AGITATOR MOUNTING PAD	ASTM-A240-316L	7900.00	159.44	1.26	1
10.6.3	ROUND BAR	ASTM-A479-304	7900.00	17.91	0.14	4
10.7	STD. BOTTOM BEARING	AISI-316L	7900.00	159.87	1.26	1
10.8	WASHER	ASTM-A240-304	7900.00	12.27	0.10	1

Figura 54: Expandir/recolher grupos

Em listas BOM hierárquicas, cada sub-montagem forma um grupo que pode ser recolhido ou expandido na grade BOM.

Para recolher ou expandir um grupo, siga um destes procedimentos:

- Clique no ícone do nó na primeira coluna da grade da lista BOM.
- Clique o botão-direito do mouse na linha, e selecione **Recolher** ou **Expandir** no menu de contexto.

Você também pode recolher ou expandir todos os grupos ou todos os grupos correspondentes ao mesmo nível de montagem, com o menu de contexto.

Nota: O status do grupo de sub-montagem (recolhido ou expandido) não é armazenado no documento. Quando a tabela BOM é fechada e reaberta, todos os grupos de sub-montagem são expandidos por padrão.

Propriedades

Bill of materials properties	
Title	Bill of material <NAME>
Type	Top level
Property set	Mechanical only
Filter	"Is standard" == "Yes"
Sorting mode	Automatic order
Grouping mode	By columns only
Footer title	TOTAL:
Fixed corner	Top left
CSV delimiter	; (Semicolon)
Warning messages	On
Column properties	
Title	Mass
Category	Mass
Role	Regular column
Format	%lu2%pr2
Visible	Yes
Width	Auto
Filter	In [10, 50]
Aggregate function	Average value
Footer type	Sum
Unit placement	In column title
Title format	<Title>, <Unit>
Units mode	User-defined
Unit	kg

Figura 55: Propriedades de lista de materiais

A grade de propriedades está localizada na parte inferior do painel do gerenciador. Para ajustar seu tamanho, use o controle do divisor, localizado entre a grade e a grade de propriedades.

A grade de propriedades contém as propriedades da tabela atual; título, tipo, filtro, modo de ordenamento e título do rodapé. Se alguma coluna for selecionada, a grade também exibirá suas propriedades comuns. Você pode alterar qualquer propriedade comum para todas as colunas selecionadas de uma só vez. As propriedades da coluna alteram o título, a visibilidade e a largura da coluna.

Para valores numéricos, você pode definir o tipo de rodapé, o posicionamento da unidade e o modo de unidades (se a coluna tiver unidades que possam ser alteradas).

O **Posicionamento da unidade** define onde as unidades são exibidas: no título da coluna, na mesma célula como o próprio valor, ou em sua própria coluna.

O **Modo de unidades** define como a unidade exibida deve ser obtida; a melhor unidade para todos os valores, a melhor unidade para exibir um valor específico na célula, ou uma unidade especificada. Você pode escolher o **Modo de unidades** para distância, área, volume ou propriedades de massa.

21.4.10.4 Filtrar

É possível adicionar um filtro personalizado à BOM atual. Para aplicar um filtro, siga um destes procedimentos:

- Usar a propriedade **Filtrar** na seção **Propriedades da Lista de Materiais** da **grade de Propriedades**.
- Usar o comando **Definir filtro** no **menu Principal**.

Em ambos os casos, a caixa de diálogo *Configurador de Filtros* é exibida:

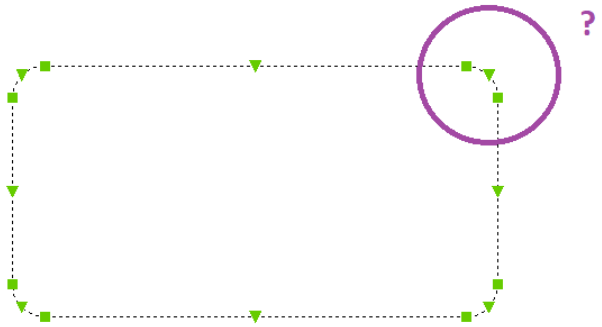


Figura 56: Configurador de Filtros

Cada linha nesta caixa de diálogo representa uma condição elementar. Também é possível adicionar grupos para combinar várias condições e/ou grupos elementares. Grupos permitem criar filtros complexos.

Adicionar uma nova condição elementar

Faça o seguinte:

1 Se for aplicável, escolha a categoria exigida, para limitar a lista de propriedades a essa categoria específica.

2 Selecione a propriedade necessária na lista de propriedades. Quando selecionada, uma condição para a propriedade é adicionada automaticamente.

3 Se for aplicável, escolha o tipo de condição necessária:
- Não igual a
- Menor que
- Maior que
- Não inferior a
- Não maior que

4 Defina o valor de destino. Para valores de texto, você pode selecionar o valor de destino na lista suspensa de todos os valores possíveis. Para valores numéricos, você pode especificar a unidade do valor de destino. O valor será automaticamente convertido para a unidade do documento.

Remoção de uma ou mais condições

Faça o seguinte:

1 Selecione as condições para remover. Você pode usar a tecla Ctrl para selecionar várias condições, e a tecla Shift para selecionar um intervalo de condições.

2 Clique na ferramenta **Excluir** (✕)

ou clique o botão-direito na linha que deseja excluir, e selecione **Remover condição** no menu de contexto.

Adicionar um novo grupo de condições

Faça o seguinte:

1 Clique na ferramenta **Adicionar** (✚).

2 Selecione **Adicionar grupo "Todos de"** para adicionar um novo grupo "Todos de"
ou selecione **Adicionar grupo "Qualquer de"** para adicionar um novo grupo "Qualquer de".

3 Adicione as condições necessárias.

Agrupar as condições selecionadas

Faça o seguinte:

1 Selecione as condições para remover. Você pode usar a tecla Ctrl para selecionar várias condições, e a tecla Shift para selecionar um intervalo de condições.

2 Clique o botão-direito sobre a linha que deseja agrupar.

3 Selecione **Formar o grupo "Tudo de" dos selecionados**, para formar um novo grupo "Tudo de",
ou selecione **Formar grupo "Qualquer de" dos selecionados**, para formar um novo grupo "Qualquer de".

Para aplicar o filtro, clique em **OK**.

21.4.10.5 Filtro de coluna

Os filtros de coluna permitem que você defina as condições sob as quais as linhas serão incluídas na tabela. Cada filtro de coluna define uma condição que os valores da coluna devem atender, para ser incluídos. Por exemplo, você pode definir um filtro de coluna para incluir apenas peças com quantidades maiores que um determinado número, ou peças cujos nomes contenham a palavra **Nozzle**. Uma coluna não pode ter mais do que um filtro. No entanto, é possível definir simultaneamente filtros para quantas colunas você precisar. Colunas com filtros ativos são marcadas com um ícone de filtro (▽) em seus títulos.

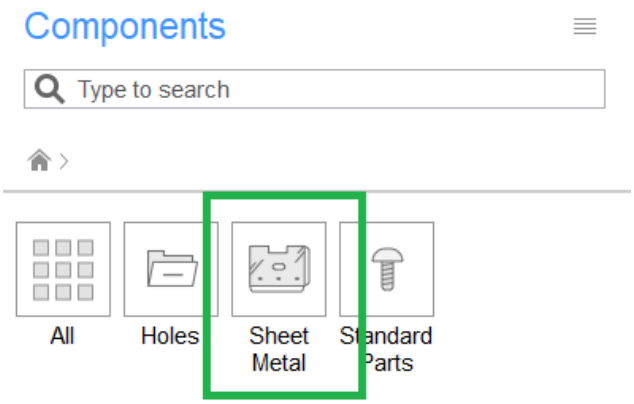


Figura 57: Exemplo de colunas com filtros ativos

Para adicionar um filtro de coluna, clique o botão-direito no título da coluna e selecione **Definir filtro de coluna** no menu de contexto.

Uma caixa de diálogo *Filtro de coluna* é exibida.

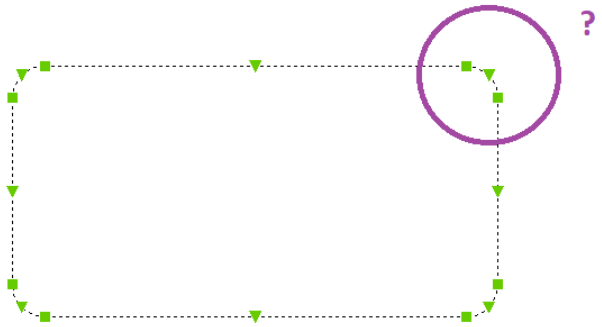


Figura 58: Filtro de coluna

Selecione o tipo de filtro necessário. Em seguida, você precisará definir o(s) valor de destino para o filtro, dependendo do tipo de filtro e do conteúdo da coluna.

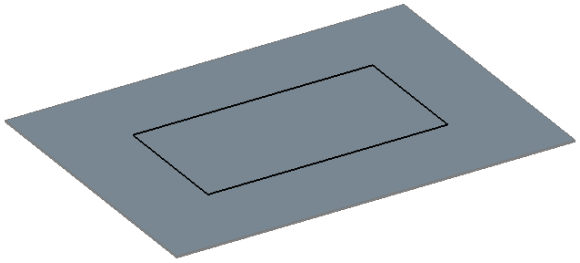


Figura 59: Valores de destino para o filtro

Para colunas com conteúdo numérico, você pode definir a unidade para o valor de destino. O valor de destino será automaticamente convertido para a unidade do documento internamente.

Para filtros de texto, você pode escolher entre as opções **ignorar caixa** (maiúscula/minúscula) e **igualar correspondência da caixa** (maiúscula/minúscula). Os filtros **Em** e **Não em** mostram a lista de todos os valores possíveis para esta coluna (em relação aos filtros já aplicados). Verifique os valores que você deseja ser incluídos ou excluídos de forma correspondente. Você também pode verificar todos os valores, desmarcar todos os valores, ou inverter a seleção atual usando os botões correspondentes nas barras de ferramentas, ou comandos no menu de contexto.

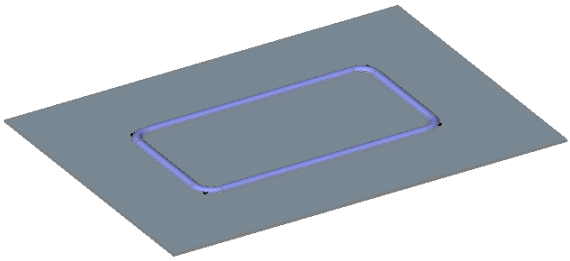


Figura 60: Filtro de Coluna

Para aplicar o filtro de coluna, clique em **OK**.

Para remover um filtro de coluna, clique o botão-direito no título da coluna e selecione **Remover filtro da coluna** no menu de contexto.

Para obter mais detalhes, consulte a seção Filtragem em Lista de Materiais.

21.4.10.6 Ordenar

As tabelas da BOM suportam vários modos de classificação (para obter mais detalhes, consulte a seção Classificação em Lista de Materiais).

Para alternar entre esses modos, use a propriedade **Modo de ordenamento** na seção **Propriedades da Lista de materiais** da **grade Propriedades**. Use o menu suspenso **Modo de ordenamento** ou clique duas vezes nos rótulos de coluna na **Grade** para alternar entre ordem crescente, decrescente, ou natural, para a respectiva coluna.

21.4.10.7 Unidades da coluna

É possível alterar o **Posicionamento da unidade** e o **Modo de unidades** para a maioria das propriedades numéricas com unidades. Selecione as colunas necessárias na **Grade** e use a seção **Propriedades da coluna** da **grade Propriedades** para alterar o **Posicionamento da unidade** e o **modo Unidades**.

Para obter mais detalhes, consulte a seção Unidades de coluna em Lista de Materiais.

Veja também: Alterar as unidades e os modos de posicionamento da unidade para a coluna.

21.4.10.8 Colunas avaliadas

Uma Lista BOM suporta colunas avaliadas. Existem dois tipos de colunas avaliadas: colunas de fórmula e colunas template. Para obter mais detalhes sobre campos de fórmula, como sintaxe de expressão, consulte a seção Colunas avaliadas em Lista de Materiais.

Veja também: Adicionar uma coluna avaliada na página 1628.

Colunas de fórmula

Para adicionar uma coluna de fórmula, faça o seguinte:

1 Selecione a categoria **da coluna Avaliada** usando o **seletor Categoria**.

2 Selecione a **coluna Fórmula** no **Seletor de coluna**.

3 Use as ferramentas **Inserir coluna após** ou **Inserir coluna antes**.

Como alternativa, você pode usar a opção **Inserir coluna de fórmula** do **menu Principal**.

Em ambos os casos, uma caixa de diálogo *Configurador de expressões* é exibida:

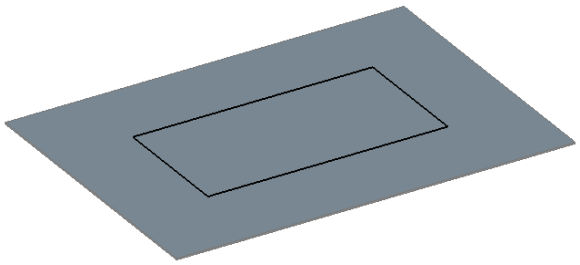


Figura 61: Configurador de expressões

Esta caixa de diálogo permite a você configurar o título, a unidade, e a expressão para a coluna da fórmula. Esta também exibe uma vista prévia dos resultados da fórmula. O campo de expressão suporta preenchimento automático.

Para adicionar uma referência a uma coluna ou uma propriedade à expressão na posição do cursor, selecione a propriedade necessária na lista de propriedades. Você também pode clicar na ferramenta **Adicionar referência de coluna** (ABC) e selecionar a propriedade no menu, ou clicar na ferramenta **Adicionar função** ( ) para selecionar a função exigida.

Para adicionar uma coluna de fórmula à tabela BOM, clique em **OK**.

Colunas de template

Para adicionar uma coluna de template, faça o seguinte:

1 Selecionar *categoria <Coluna avaliada>* usando o **seletor de Categoria**.

2 Selecionar *<Coluna de modelo>* no **seletor de Coluna**.

3 Use as ferramentas **Inserir coluna após** ou **Inserir coluna antes**.

Como alternativa, você pode usar a opção **Inserir coluna template** no **menu Principal**.

Em ambos os casos, uma caixa de diálogo *Configurador de template* é exibida.

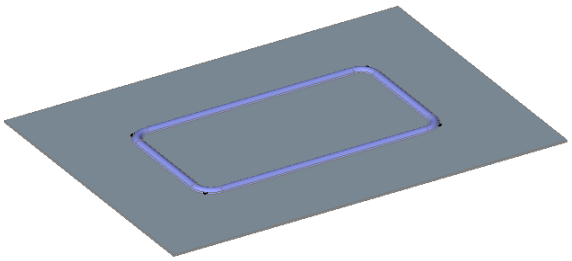


Figura 62: Configurador de template

Esta caixa de diálogo permite a você configurar o título, a expressão e, se necessário, a unidade, para a coluna do template. Este também exibe uma prévia dos resultados. O campo de expressão suporta preenchimento automático.

As colunas de template podem conter mais de uma sub-expressão de template. Nesse caso, essas são avaliados do primeiro ao último, e a primeira sub-expressão que ocasiona resultados não-vazios é usada. O **Configurador de template** permite adicionar, excluir, e alternar entre sub-expressões de template usando a barra de ferramentas. Este também permite alterar a ordem das sub-expressões de template.

Nota: Observe que, no campo de expressão, todas as sub-expressões são mostradas de uma só vez, onde cada sub-expressão está entre {} colchetes.

Para adicionar uma sub-expressão, clique em  .

Para excluir a sub-expressão atual, clique em  .

Nota: Quando a última sub-expressão é removida, esta será substituída por uma vazia.

Para alterar a sub-expressão atual, selecione o número da sub-expressão necessária na lista suspensa.

Use  e  para alterar a ordem das sub-expressões.

A sub-expressão atualmente selecionada pode ser configurada usando a seção **Template**. Cada coluna nesta seção representa um item da expressão do modelo, que você pode alterar.

Os tipos de itens de template são:

- **Referência de coluna**: representa uma referência a uma coluna BOM ou uma propriedade. Para adicionar uma referência de coluna, selecione a coluna necessária na lista de propriedades. Esta será adicionada automaticamente após a última coluna selecionada, ou, se nenhuma coluna for selecionada, após a última coluna. Se necessário, um **Delimitador** será adicionado automaticamente na frente da **Referência da coluna**.
- **Delimitador**: representa um separador entre dois itens. O texto do delimitador vai aparecer no resultado final somente se o texto em ambos os lados deste não estiver vazio. Por exemplo, se você tiver referências de coluna PartNumber e Length com o delimitador "X" entre estas, o "X" vai ser adicionado ao texto final somente se ambos, PartNumber e Length, não estiverem vazios. Na maioria dos casos, os delimitadores são adicionados automaticamente. No entanto, você pode adicionar delimitadores usando o botão  e selecionando **Adicionar delimitador** na lista suspensa. Como alternativa, você pode adicionar um delimitador a partir do menu de contexto.
- **Texto**: representa um texto personalizado. Você pode adicionar texto personalizado clicando  e selecionando **Adicionar texto** na lista suspensa. Como alternativa, você pode adicionar um texto personalizado no menu de contexto.
- **Texto avaliado**: representa um item que deve ser adicionado à expressão final, como está. O texto avaliado pode conter funções ou outros símbolos especiais que irão modificar o resultado. Você pode adicionar um texto avaliado usando o botão  e selecionando **Adicionar expressão** a partir da lista suspensa. Como alternativa, você pode adicionar um texto avaliado a partir do menu de contexto.

A qualquer momento, você pode alterar o conteúdo de qualquer item no template. Você também pode excluir itens selecionando-os e usando o botão  , ou escolhendo **Excluir coluna** no menu de contexto.

Para mover um item, selecione uma coluna no template, arraste-a e solte esta na posição desejada. Quaisquer modificações do template serão refletidas no campo de expressão. Da mesma forma, quaisquer alterações no campo de expressão serão refletidas na área do template.

Para adicionar uma coluna de template à tabela BOM, clique em **OK**.

21.4.10.9 Linha de rodapé

Uma tabela BOM pode incluir uma linha de rodapé. Para adicionar uma linha de rodapé, especifique o tipo de rodapé para uma ou mais colunas. Por padrão, o título da linha do rodapé é *TOTAL:*. Altere o título do rodapé com a propriedade **Título do rodapé** na seção **Propriedade Lista de materiais** da **grade Propriedades**.

Para obter mais informações sobre uma linha de rodapé em uma BOM, consulte a seção Rodapé em Lista de Materiais.

Veja também: Adicionar um rodapé para a coluna.

21.4.10.1 Colunas agregadas
0

Nas tabelas BOM de top-level e bottom-level, quase qualquer coluna pode ser atribuída com uma função agregada. Agregar funções agrupa valores de várias linhas na BOM em um único valor, que diferem apenas naqueles valores. Por exemplo, funções agregadas podem ser usadas para calcular o comprimento total dos tubos em relação ao seu tamanho e espessura, ou para concatenar todos os tamanhos de parafusos e porcas usados na montagem em uma única célula.

Para obter mais detalhes, consulte a seção Colunas em Lista de Materiais.

Para definir uma função agregada, siga um destes procedimentos:

- Clique o botão-direito no título da coluna e selecione a função agregada exigida, a partir do submenu **Função agregada**.
- Selecione a coluna e altere a função agregada na propriedade coluna correspondente.

As colunas agregadas são marcadas com o símbolo Σ em seus títulos.

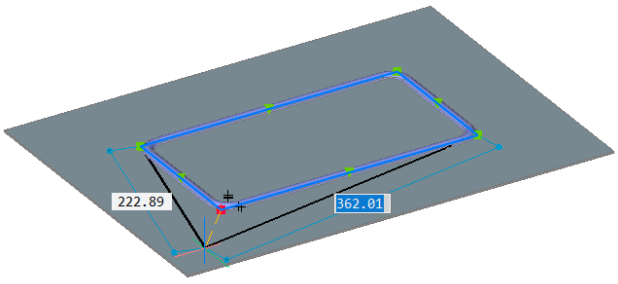


Figura 63: Exemplo de colunas agregadas

21.4.10.1 Templates

1

Você pode salvar configurações de BOM como templates. Use o gerenciador de BOM para criar uma nova tabela a partir de um template e para substituir a configuração de tabela atual por uma a partir de um template. Você pode salvar a configuração como um template a qualquer momento. Você pode alterar uma configuração de template carregada com as ferramentas regulares e salvá-la como outro template.

Para obter mais informações sobre templates BOM, consulte a seção Templates em Lista de Materiais.

21.4.10.1 Pesquisar

2



Figura 64: Pesquisar

O gerenciador de BOM tem um campo de pesquisa que permite a você pesquisar um valor específico na tabela. Você também pode usar curingas no campo de pesquisa. Por padrão, o BricsCAD pesquisa na tabela inteira. Para limitar a área de pesquisa, selecione as colunas necessárias. Se o valor for encontrado, o cursor se move para aquela célula. Se nenhuma coluna for selecionada, a linha é que vai ser selecionada. A **Grade** rola automaticamente, para tornar a célula visível. Para encontrar outro valor, use os botões ou . Você também pode usar **F3** ou **Ctrl+F3**. Para selecionar todos os resultados encontrados, use o botão .

Por padrão, o texto no campo de pesquisa é pesquisado. No entanto, você também pode usar **curingas**. Os seguintes símbolos especiais são suportados:

Tabela 19: Curingas

Nome	Caractere	Descrição
Aspas simples	`	Escapa o próximo caractere (toma-o literalmente).
Vírgula	,	Separa padrões.
Asterisco	*	Compara qualquer sequência de caracteres, incluindo uma sequência nula, em qualquer lugar.

Tabela 19: Curingas

Nome	Caractere	Descrição
Ponto de Interrogação	?	Compara qualquer caractere individual.
Ponto final	.	Corresponde a um único caractere não alfanumérico.
Sinal de arroba	@	Compara a um único caractere alfabético.
Sustenido	#	Compara a um único caractere numérico.
Til	~	Se o primeiro caractere de um padrão corresponder a algo que não seja o padrão.
Colchetes	[...]	Corresponde a um único caractere delimitado.
Til entre chaves	[~...]	Corresponde a um único caractere não-delimitado.
Hífen	-	Entre parênteses, corresponde a um único caractere no intervalo especificado.

Para abrir o campo de pesquisa, use a ferramenta **Pesquisar** na barra de ferramentas. Você também pode pressionar **Ctrl+F**.

21.4.10.1 Tabela associada 3

Cada tabela BOM no gerenciador pode ter uma tabela associada no documento, que é atualizada quando a configuração da BOM é alterada. Uma tabela BOM pode ter apenas uma tabela associada. No entanto, você pode criar tabelas de BOM que não tenham uma tabela associada no documento.

Para criar uma tabela associada, use **Colocar/Atualizar tabela**. Se a tabela BOM foi criada com o comando *BMBOM*, a tabela associada será criada automaticamente.

Você pode localizar a tabela associada no documento com **Tabela associada - Zoom para** ou **Tabela associada - Selecionar**. Se necessário, o layout ativo muda para o layout de tabela associada.

Você pode excluir a tabela associada com **Tabela associada - Excluir**. Como alternativa, você pode desanexar a tabela associada a partir da tabela BOM com **Tabela associada - Desanexar**. Após a desvinculação, a tabela associada perde seu link com a tabela BOM, e não será mais atualizada com a BOM.

21.4.10.1 Atualizar a BOM

4

Desde o BricsCAD V20.2, o gerenciador de BOM atualiza automaticamente o conteúdo da tabela no painel quando a montagem é modificada ou atualizada. Para grandes montagens, isso pode ser inconveniente. O gerenciador de BOM atualiza automaticamente a tabela quando estiver ativo.

Para evitar a atualização automática, feche o gerenciador de BOM para tornar o painel do Gerenciador de BOM inativo.

Use **Atualizar** para atualizar manualmente o conteúdo do gerenciador de BOM.

Nota: A tabela associada não é atualizada automaticamente quando a montagem é alterada ou modificada.

Use a opção **Local/Atualizar tabela** para atualizar a tabela associada.

21.4.10.1 Adicionar uma nova lista de materiais

5

1 Use o submenu **Nova Lista BOM** do **Menu principal**.

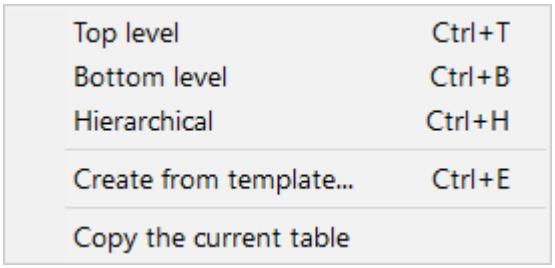


Figura 65: Novo submenu BOM

2 Selecione o tipo de BOM ou selecione **Do template** para escolher o arquivo de modelo para uma lista BOM.

21.4.10.1 Selecionando a BOM atual

6

Use o **Seletor de lista BOM** para selecionar uma lista de materiais já existente.

Nota: A tabela será atualizada automaticamente até o estado atual da montagem, bem como a tabela associada no desenho (se esta existir).

21.4.10.1 Alterar o tipo da lista BOM atual

7

Altere a propriedade **Tipo** na seção **Propriedades da Lista de materiais** na **grade das Propriedades**.

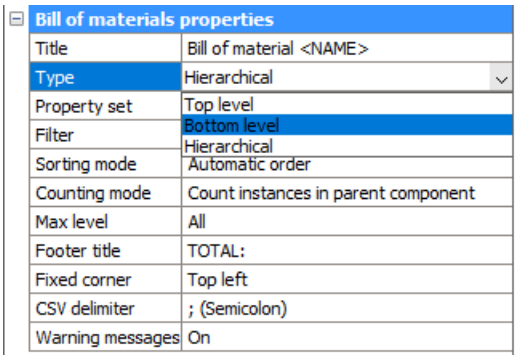


Figura 66: Propriedades de lista de materiais

21.4.10.1 Para alterar o conjunto de propriedades da atual lista BOM 8

Altere a propriedade **Conjunto de propriedades** na seção **Propriedades da Lista de Materiais** na **grade das Propriedades**.

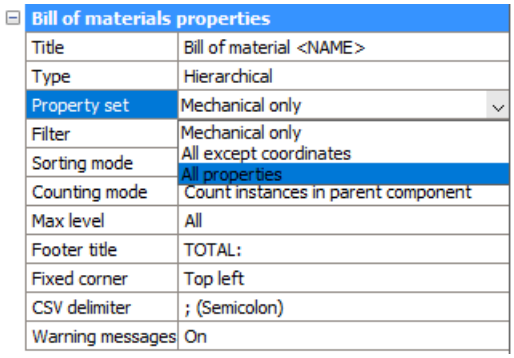


Figura 67: Todas as propriedades

21.4.10.1 Alterando o modo de contagem de uma BOM hierárquica 9

Altere a propriedade **Modo de contagem** na seção **Propriedades da Lista de materiais** na **grade das Propriedades**.

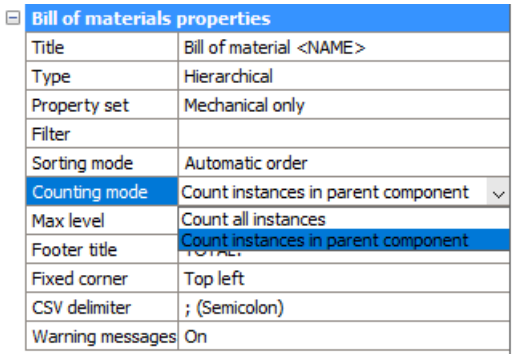


Figura 68: Propriedades de lista de materiais

21.4.10.2 Alterando o modo de agrupamento do BOM de nível superior ou inferior 0

Altere a propriedade **Modo de agrupamento** na seção **Propriedades da Lista de materiais** na **grade das Propriedades**.

Bill of materials properties	
Title	Bill of material <NAME>
Type	Top level
Property set	All except coordinates
Filter	
Sorting mode	Automatic order
Grouping mode	Auto
Footer title	Auto
Fixed corner	By components and columns
CSV delimiter	By columns only
Warning messages	; (Semicolon)
	On

Figura 69: Propriedades de lista de materiais

21.4.10.2 Definir um filtro personalizado para a BOM atual 1

1 Selecione a opção **Definir filtro** no **menu Principal**.

Uma caixa de diálogo *Configurador de Filtros* é exibida.

2 Configure o filtro usando a ferramente de diálogo.

Ver Filtrar na página 1615 para obter detalhes.

3 Clique em **OK** para aplicar o filtro.

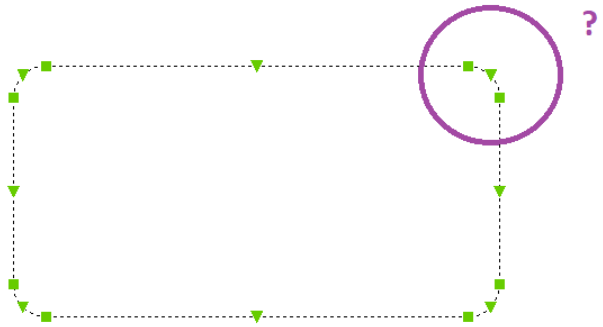


Figura 70: Configurador de Filtros

21.4.10.2 Para adicionar novas colunas à lista BOM atual 2

1 Se for aplicável, use o seletor **Categoria** para selecionar a categoria das propriedades que você deseja

adicionar.

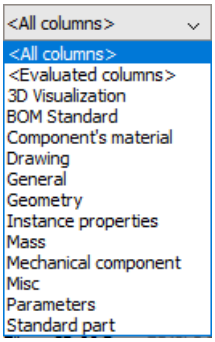


Figura 71: Todas as colunas

2 Selecione a propriedade a ser adicionada na lista das colunas disponíveis. Esta lista contém todas as colunas da categoria selecionada que ainda não foram adicionadas à BOM. Se você começar a digitar o nome do campo, o BricsCAD sugere automaticamente nomes para você. Clique para selecioná-lo na lista.

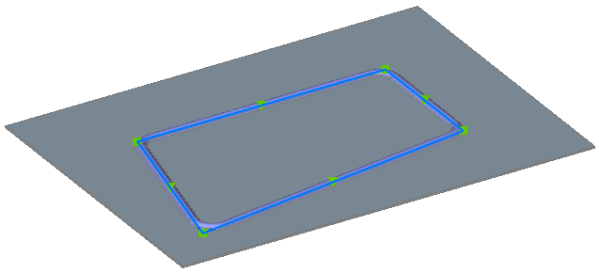


Figura 72: Descrição do componente

3 Se for aplicável, selecione a coluna na BOM em que você deseja colocar a nova coluna ao lado.

4 Clique em **Inserir a coluna depois** () para colocar a nova coluna após a que for selecionada, ou **Inserir a coluna antes** () para colocar a nova coluna antes da selecionada.

Se nenhuma coluna for selecionada, a nova coluna será colocada como a última ou a primeira posição, respectivamente.

21.4.10.2 Adicionar uma coluna avaliada

3

1 Selecione a categoria **Colunas avaliadas** usando o **seletor de Categorias**.

2 Selecione **Coluna de Fórmula** para criar uma coluna de fórmula ou selecione **Coluna de template** para criar uma coluna de template.

3 Clique em **Inserir a coluna depois** () para colocar a nova coluna após a que for selecionada, ou **Inserir a coluna antes** () para colocar a nova coluna antes da selecionada.

Se nenhuma coluna for selecionada, a nova coluna será colocada como a última ou a primeira posição, respectivamente.

Uma caixa de diálogo para configurar a coluna avaliada é exibida.

4 Altere o **Título** da coluna.

5 Se for aplicável, altere a **Unidade**.

6 Configure a expressão utilizando as ferramentas de diálogo.

7 Clique em **OK** para adicionar a coluna à tabela BOM.

21.4.10.2 Adicionar uma coluna template a partir de colunas selecionadas

4

1 Selecione as colunas a ser combinadas em uma coluna de template. Use a tecla Ctrl para selecionar múltiplas colunas, ou a tecla Shift para selecionar um intervalo de colunas.

2 Clique o botão-direito no título de uma das colunas selecionadas.

3 Selecione **Adicionar coluna de modelo na selecionada** no menu de contexto.

Uma caixa de diálogo para configurar a coluna avaliada, é exibida.

4 Altere o **Título** da coluna.

5 Se for aplicável, altere a **Unidade**.

6 Configure a expressão utilizando as ferramentas de diálogo.

7 Clique em **OK** para adicionar a coluna à tabela BOM.

8 Se for aplicável, remova ou oculte colunas que não são mais necessárias.

21.4.10.2 Alterar uma coluna avaliada

5

1 Clique o botão-direito no título da coluna na **grade BOM**.

2 Selecione **Editar expressão de coluna** no menu de contexto.

É exibida uma caixa de diálogo para configurar a coluna avaliada.

3 Se for aplicável, modifique o título da coluna, a unidade, e a expressão.

4 Clique em **OK** para aplicar as alterações.

21.4.10.2 Remover uma coluna da BOM atual

6

1 Selecione a coluna na grade.

2 Clique na ferramenta **Remover a coluna** (),

ou pressione **Excluir**.

21.4.10.2 Alterar o título da Lista BOM

7

Use o **Título da BOM** para editar o título

ou use a propriedade **Título** na seção **Propriedades da lista de materiais** da **grade Propriedades**.

Você pode usar o *<NOME>* espaço reservado para se referir ao nome da montagem.

21.4.10.2 Alterar o título da linha de rodapé 8

Use a propriedade **Titulo do rodapé** na seção **Propriedades da Lista de Materiais** na **grade Propriedades**.

21.4.10.2 Adicionar um prefixo ou sufixo aos números 9

1 Selecione a coluna **Número**.

2 Selecione a propriedade **Prefixo do número** na seção **Propriedades da coluna**.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

Figura 73: Propriedades de coluna

3 Digite a nova string de prefixo, se necessário.

4 Selecione a propriedade **Sufixo do número** na seção **Propriedades da coluna**.

5 Digite a nova string de sufixo, se necessário.

21.4.10.3 Alterar o delimitador entre números em Lista hierárquica 0

1 Selecione a coluna **Número**.

2 Selecione a propriedade **Delimitador** na seção **Propriedades da coluna**.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

Figura 74: Propriedades de coluna

3 Entre a nova string do delimitador.

21.4.10.3 Alterar o tipo de número na Lista hierárquica 1

1 Selecione a coluna **Número**.

2 Selecione a propriedade **Tipo de número** na seção **Propriedades da coluna**.

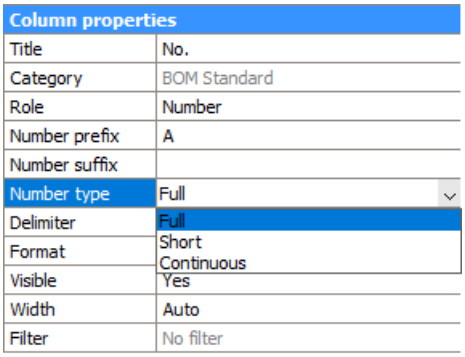


Figura 75: Propriedades de coluna

3 Selecione o tipo de número necessário.

21.4.10.3 Alterar o título da coluna

2

1 Selecione a coluna na **grade da BOM**.

2 Selecione a propriedade **Título** na seção **Propriedades da coluna**.

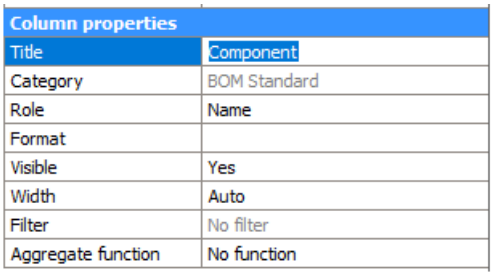


Figura 76: Propriedades de coluna

3 Altere o valor da propriedade **Título** para o novo título da coluna.

21.4.10.3 Alterar o formato da coluna

3

1 Selecione a(s) coluna.

2 Clique o botão-direito no título de uma coluna selecionada.

3 Selecione **Formatar** no menu de contexto.

A caixa de diálogo *Formatar* é exibida.

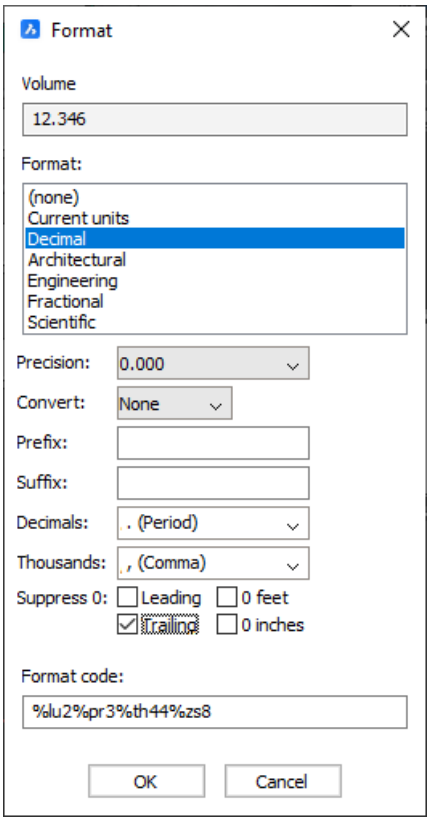


Figura 77: Formato

4 Defina o formato desejado.

5 Clique em **OK** para aplicar as alterações.

21.4.10.3 Alterar as unidades e os modos de posicionar esta numa coluna 4

1 Selecione a(s) coluna.

2 Se aplicável, altere a propriedade **Posicionamento da unidade** na seção **Propriedades da coluna**.

3 Se aplicável, altere a propriedade **Modo de unidades** na seção **Propriedades da coluna**.

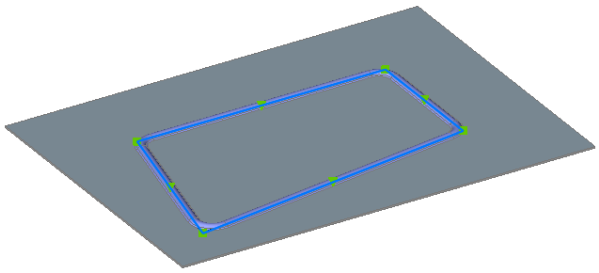


Figura 78: Propriedades de coluna

4 Se aplicável, altere o **Formato do título**.

- Usar *<Título>* para adicionar o título da coluna.

- Usar *<Unidade>* para adicionar a unidade de coluna.
- Usar *\n* para adicionar uma quebra de linha.

21.4.10.3 Adicionar um rodapé a uma coluna 5

1 Selecione a(s) coluna.

2 Use o submenu **Rodapé** do **menu Principal** para selecionar o tipo de valor do rodapé,

ou clique o botão-direito no rótulo da coluna e selecione o submenu **Rodapé**.

Os seguintes tipos estão disponíveis:

- **Nenhum rodapé** (a coluna será removida do rodapé. Se não houver coluna com outro tipo de rodapé, a linha do rodapé será removida completamente)
- **Soma**
- **Valor médio**
- **Valor mínimo**
- **Valor máximo**

Como alternativa:

1 Selecione a(s) coluna.

2 Na **grade de Propriedades**, selecione a propriedade **Tipo de rodapé** e escolha o tipo de rodapé desejado.

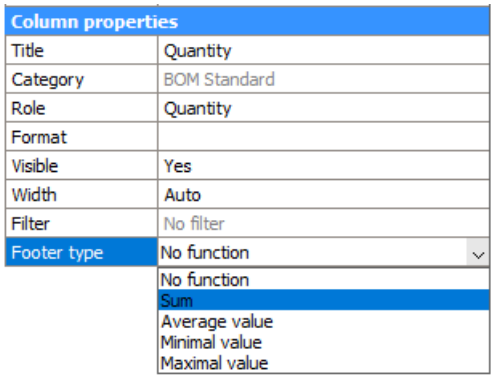


Figura 79: Propriedades de coluna

Nota: Para alternar entre **Soma** e **Sem rodapé**, clique no botão **Rodapé** ( ).

21.4.10.3 Salvar a configuração atual como um template 6

1 Escolha **Salvar como template** ( ) no menu principal ( ).

A caixa de diálogo para selecionar o nome e o local para um arquivo de template é exibida.



Figura 80: Selecione o arquivo para salvar o modelo de tabela de BOM

2 Especifique o nome e a localização do arquivo template.

3 Clique **Salvar**.

21.4.10.3 Exportar a tabela atual para CSV

7

1 Se aplicável, ajuste a propriedade **delimitador CSV** da tabela.

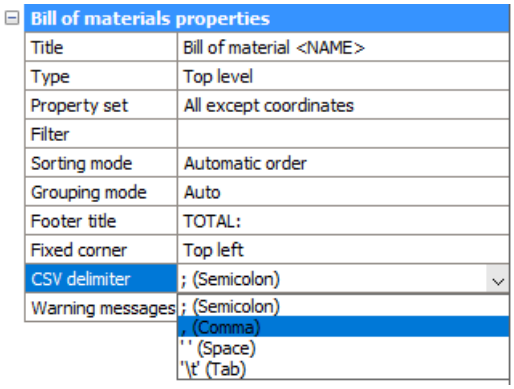


Figura 81: Delimitador CSV

2 Escolha **Exportar para CSV** no menu principal ( ).

A caixa de diálogo para selecionar o nome e o local de um arquivo CSV é exibida.

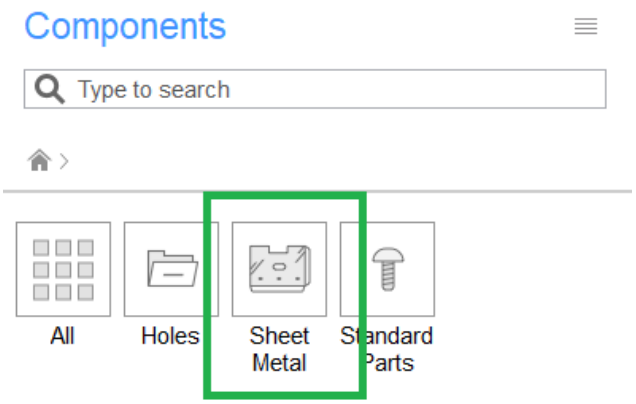


Figura 82: Entrar o nome do arquivo para exportar a tabela BOM

3 Especifique o nome e a localização do arquivo CSV.

4 Clique **Salvar**.

21.4.11 Lista de Materiais

21.4.11.1 Visão Geral

Você pode usar os comandos *BMBOM*, *BMBOMPANELOPEN* e *BMBOMPANELCLOSE*.

Uma Lista de Materiais (BOM) na engenharia é uma lista de componentes mecânicos exclusivos usados numa montagem. No BricsCAD, a lista é representada como uma entidade tabela (consulte Tabelas) e pode ser colocada em qualquer lugar do desenho. Cada entidade tem seu próprio número de série (começando com 1), nome e número de ocorrências.

Você pode exibir e configurar Listas de Materiais no Gerenciador de Listas de Materiais.

21.4.11.2 Tipos de Listas de Materiais

Existem 3 tipos de Lista BOM no BricsCAD:

- **Top level**

  Contém apenas componentes de nível superior. Os componentes de nível-superior são componentes inseridos diretamente na montagem principal. Este tipo de lista BOM traz apenas os componentes principais, sem detalhar os subcomponentes.
- **Bottom level**

  Contém apenas componentes de nível inferior (peças). Componentes de nível inferior são peças que não contêm outras peças ou sub-montagens. Este tipo de lista BOM traz todas as peças no conjunto principal, independentemente de saber a qual componente essas pertencem.
- **Hierárquica**

Contém todos os componentes com seus subcomponentes listados diretamente abaixo. Os números de série dos subcomponentes são uma sequência de números separados por pontos (por exemplo, "1.2.3"). Esse tipo de lista BOM mostra a estrutura hierárquica da montagem.

Nota: A propriedade **Status da BOM** controla quais peças estão incluídas em uma Lista de Materiais. O nível de um componente em uma tabela BOM é definido com relação ao **Status da BOM** para todos os componentes relacionados. Por exemplo, um componente com um status de BOM Terminal é sempre considerado como nível-inferior, mesmo que contenha outras peças ou subconjuntos.

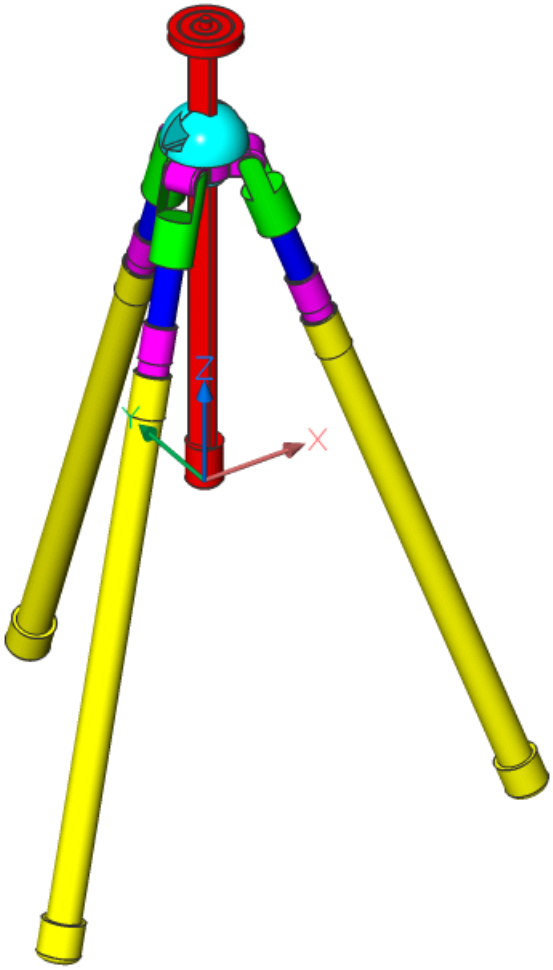


Figura 83: Exemplo: montagem de tripé

Tabela 20: Tipos de BOM

Tipo	Lista de Materiais
Top-level	Bill of Materials tripod_main_assy No. | Component | Quantity 1 | tripod_axis_assy | 1 2 | tripod_foot_assy | 3 **Figura 84:** Top-level
Bottom level	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 **Figura 85:** Bottom level
Hierárquica	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 **Figura 86:** Hierárquica

21.4.11.3 Modo de contagem e nível máximo

Para tabelas BOM hierárquicas, dois modos de contagem são suportados:

- **Contar todas as instâncias**

  O número total de sub-montagens e peças, na montagem inteira.

- **Contar instâncias no componente pai**

  O número total de sub-montagens e peças no componente pai. O componente pai é considerado como o componente raiz.

Por exemplo, a montagem do Tripé principal contém 3 montagens de pés do Tripé. Cada instância de uma montagem de pé do Tripé contém 1 montagem de Conexão e 1 montagens de Tubo.

Se o **modo de contagem** estiver definido como **Contar todas as instâncias**, o resultado será:

Tabela 21: Conjunto de tripé BOM (conte todas as instâncias)

#	Pai	Quantidade
1	Montagem do tripé	1
1.1	Montagem do pé do tripé	3
1.1.1	Montagem da Conexão	3
1.1.2	Montagem dos tubos	3

Se o **modo de contagem** for definido como **Contar Instâncias no componente pai**, o resultado será:

Tabela 22: Montagem do tripé BOM (contar instâncias por componente)

#	Pai	Quantidade
1	Montagem do tripé	1
1.1	Montagem do pé do tripé	3
1.1.1	Montagem da Conexão	1
1.1.2	Montagem dos tubos	1

O valor padrão para o **modo de contagem** é **Contar instâncias no componente pai.** Você pode alterar o modo de contagem no Gerenciador de BOM, ou usar o comando *BMBOM*.

Nota: Antes do BricsCAD V20.2, todas as tabelas de BOM hierárquicas usavam o modo **Contar todas as instâncias** por padrão. Quando essas tabelas são atualizadas no BricsCAD V20.2 ou posterior, o modo padrão é definido como **Contar instâncias no componente pai**. Isso pode resultar em quantidades diferentes para algumas peças. Se necessário, defina manualmente o modo de contagem no Gerenciador de BOM.

Por padrão, listas hierárquicas de materiais mostram todos os níveis no modelo. É possível limitar uma Lista BOM hierárquica a um determinado nível usando a propriedade de **Nível máximo**. Quando o nível máximo estiver definido, apenas peças até esse nível serão incluídas. Por exemplo, se **Nível máximo** =

2, somente componentes e subcomponentes em Top level (nível superior) diretamente incluídos nos componentes de nível superior serão listados.

21.4.11.4 Colunas da Lista BOM

Por padrão, a tabela BOM contém as colunas Número, Nome e Quantidade.

Ao usar a opção Configurar do comando *BMBOM* você pode adicionar mais colunas, como Descrição, Densidade, Volume, Massa, Material, Espessura (para peças de Sheet Metal), e Parâmetros (para componentes paramétricos).

Você pode adicionar qualquer propriedade interna de componentes, instâncias ou referências de bloco à tabela da BOM. Colunas avaliadas de forma personalizada também são suportadas, usando a tecnologia ExtracaoDados.

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

Figura 87: BOM com densidade, volume, e massa

Modo de agrupar

Tabelas BOM listam apenas componentes únicos; múltiplas inserções dos mesmos componentes são agrupadas, de acordo com o modo Agrupar.

A partir do BricsCAD V21, os seguintes modos de agrupamento são suportados:

- **Auto** (modo padrão)

  Múltiplas inserções dos mesmos componentes são agrupadas em relação aos seus parâmetros e propriedades adicionadas. Isso significa que várias inserções do mesmo componente paramétrico são sempre agrupadas em linhas BOM separadas em relação aos seus parâmetros, independentemente de esses parâmetros serem adicionados à BOM como colunas.

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

Figura 88: Estante BOM

- **Por componentes e colunas**

  Neste modo, múltiplas inserções dos mesmos componentes são agrupadas apenas em relação às propriedades adicionadas. Isso significa que várias inserções do mesmo componente paramétrico podem ser agrupadas na mesma linha, mesmo que seus parâmetros sejam diferentes, dependendo

de quais colunas estão adicionadas à BOM. No entanto, instâncias de componentes diferentes são sempre agrupadas em linhas diferentes. Este modo é útil para os casos em que alguns parâmetros correspondem a diferentes estados de componentes, mas não resultam em peças diferentes.

- **Somente por colunas**

  Neste modo, inserções de componentes são agrupadas em relação às suas propriedades adicionadas à BOM, independentemente de serem inserções do mesmo componente. Este modo é útil para obter tabelas de resumo, como lista de todos os materiais físicos ou lista de todos os tamanhos de tubos.

Colunas visíveis e invisíveis

Você pode tornar invisível qualquer coluna na tabela BOM. Colunas invisíveis não aparecerão na tabela no documento. No entanto, você ainda pode usar essas colunas nas fórmulas. Colunas invisíveis também são exibidas no Gerenciador da BOM.

Colunas agregadas

A partir do BricsCAD V21, é possível definir uma função agregada para quase todas as colunas em tabelas BOM de top level ou bottom level. Agregar funções agrupam valores de várias linhas na BOM em um único valor (que diferem apenas nesses valores). As seguintes funções de agregação são suportadas:

- **Soma**: Soma os valores agrupados.
- **Média**: Encontra o valor médio dos valores agrupados.
- **Mínimo**: Encontra o valor mínimo de valores agrupados.
- **Máximo**: Encontra o valor máximo de valores agrupados.
- **Concatenar**: Concatena todos os valores únicos no grupo, em ordem crescente.
- **Concatenar com contagens**: concatena todos os valores únicos com seu número de contagens no grupo.

Por exemplo, pegue a seguinte tabela de BOM:

Tabela 23: Tabelas BOM

Pai	Largura	Comprim	Altura	Quantidade
Viga	16	250	10	1
Viga	16	200	10	2
Viga	16	220	20	1
Viga	20	215	20	3
Viga	20	225	20	1

Aplicar uma função de agregar à coluna **Comprimento**. Os resultados são combinados na tabela a seguir:

Tabela 24: Tabelas BOM

Pai	Largura	Altura	Comprimento (soma)	Comprimento (média)	Comprimento (mínimo)	Comprimento (máximo)	Comprimento (concatenar)	Comprimento (concatenar com contagens)
Viga	16	10	650	216.6667	200	250	200; 250	2x200; 250
Viga	16	20	220	220	220	220	220	220
Viga	20	20	870	217.5	215	225	215; 225	3x215; 225

Para colunas **Concatenadas com contagem**, você pode:

- Ajustar a posição do número de contagens (na frente ou atrás do valor).
- Adicionar o delimitador entre valores.
- Adicionar o delimitador entre valores e seu número de contagens.
- Definir o sufixo e/ou prefixo para o número de contagens.

Se uma coluna agregada for usada em uma coluna avaliada, os valores finais são usados por padrão. No entanto, é possível aplicar uma função agregada a uma coluna avaliada. Nesse caso, os resultados dessa coluna avaliada serão primeiro calculados para todos os valores originais (antes da agregação) e, em seguida, os valores resultantes serão agregados.

Coluna de número e funções de coluna

Uma coluna numérica é uma coluna especial para enumerar linhas. Esta atribui automaticamente números para todas as linhas na tabela BOM, sem saltos e independentemente de outras tabelas. As linhas são numeradas a partir de 1.

As seguintes configurações podem ser alteradas para a coluna de número:

- **Prefixo**: Uma string que é adicionada na frente de qualquer número. Por exemplo, você quer que todos os números comecem com "A".
- **Sufixo**: Uma string que é adicionada após o número. Por exemplo, você deseja adicionar" -1" a todos os números.

Para uma BOM hierárquica, as seguintes configurações estão disponíveis:

- **Tipo de número**: Especifica o formato numérico:
  - **Completo**: Números são formatados como" 1.2.3".
  - **Curto**: Apenas números para o nível atual são mostrados. Este é usado em conjunto com a coluna Nível, que mostra o nível da peça atual na montagem.

  - **Contínuo**: Todos os componentes serão numerados continuamente a partir de 1, independentemente do seu nível.
- **Delimitador**: Especifica o símbolo para separar números para diferentes níveis quando o tipo de número é definido para *Completo*.

Em geral, a coluna numérica é usada como origem para anotar no desenho com o comando *BMBALLOON*. No entanto, qualquer coluna na tabela pode ser usada como origem para os balões de anotação, alterando sua função. As seguintes funções de coluna estão disponíveis:

- **Número**: A coluna contém números para anotações.
- **Nome**: A coluna contém nomes de peça para anotações.
- **Quantidade**: A coluna contém quantidades de peças para anotações.
- **Regular**: A coluna não tem função específica.

Apenas uma coluna pode ter uma função **Número**, **Nome** ou **Quantidade**. Se nenhuma coluna tiver uma dessas funções atribuída, as anotações automáticas não serão possíveis.

Largura da coluna



Figura 89: Propriedades de coluna

Por padrão, a largura de cada coluna é ajustada automaticamente, portanto, seu conteúdo se encaixa sem quebras de linha adicionais. No entanto, é possível definir uma largura fixa para a coluna.

Para definir a largura da coluna, execute um dos seguintes procedimentos:

- Selecione a coluna necessária no gerenciador de BOM, e digite um valor para a propriedade **Width** (Largura) em unidades do layout.
- Altere a largura da coluna necessária, na tabela associada. A partir do BricsCAD V20.2, as tabelas BOM vão automaticamente manter a largura das colunas que tenham sido alteradas manualmente.

Para restaurar o modo de largura automática, selecione a coluna no gerenciador de BOM e selecione **Auto** na lista suspensa para a propriedade **Width** (Largura).

Formato da coluna

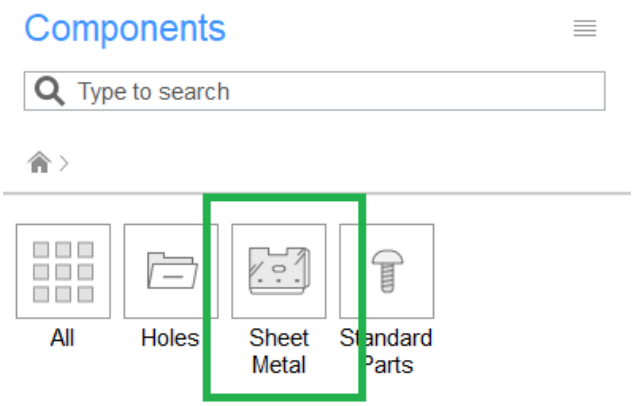


Figura 90: Formato

Você pode formatar valores de coluna:

1 Selecione a coluna necessária no gerenciador de BOM.

2 Escolha a opção **Formatar** no menu de contexto.

3 Defina o formato desejado na caixa de diálogo **Formatar**.

Nota: As configurações disponíveis dependem do tipo de dados da coluna selecionada. Você pode selecionar várias colunas do mesmo tipo para formatá-las igualmente.

Conjuntos de propriedades

Como as tabelas BOM usam a tecnologia de Extração de Dados (DataExtraction), você pode listar propriedades dos componentes mecânicos e propriedades de entidade, por exemplo, referências de bloco em tabelas de BOM.

Nota: Essas propriedades nem sempre são necessárias. Em grandes montagens, essas propriedades podem afetar negativamente a performance. Isso ocorre porque o BricsCAD armazena em cachê as propriedades durante a criação da tabela BOM.

Conjuntos de propriedades são conjuntos embutidos de propriedades. Você pode escolher o conjunto mais apropriado de propriedades para seu projeto. Os seguintes conjuntos de propriedades são suportados:

- **Somente mecânica** (conjunto de propriedades padrão): Somente as propriedades dos componentes mecânicos são listadas.
- **Tudo exceto coordenadas**: Lista todas as propriedades dos componentes, exceto suas coordenadas. Propriedades das coordenadas são normalmente usadas apenas para instâncias de componentes.
- **Todas as propriedades**: Lista todas as propriedades possíveis de componentes e instâncias de componentes.

Você pode alterar o conjunto de propriedades para uma tabela BOM no Gerenciador de BOM ou usar o comando *BMBOM* quando adicionar uma nova tabela.

21.4.11.5 Filtrar uma tabela BOM

Você pode filtrar peças que formarão a Lista de Materiais. Começando com o BricsCAD V21, existem 2 tipos de filtro: **principal** e **coluna**.

Filtro principal

Um filtro principal funciona da mesma forma que o comando *EXTRACAODADOS*. Primeiro, todas as peças da montagem, no nível requerido (superior, inferior ou hierárquico), são comparadas com a expressão do filtro. Em seguida, apenas as peças que passaram pelo filtro são incluídas na BOM.

A expressão de filtragem pode não se referir às colunas BOM porque a filtragem é feita antes que a BOM seja gerada. Por exemplo, é possível criar um filtro para todas as peças padrão porque **É padrão** é uma propriedade normal do componente. No entanto, não é possível definir o filtro principal para filtrar todas as peças com uma quantidade superior a 1. Você deve usar filtros de Coluna para isso.

Configurador de filtros

A maneira mais fácil de configurar o filtro principal é usar o **Configurator de Filtros** no gerenciador de BOM.

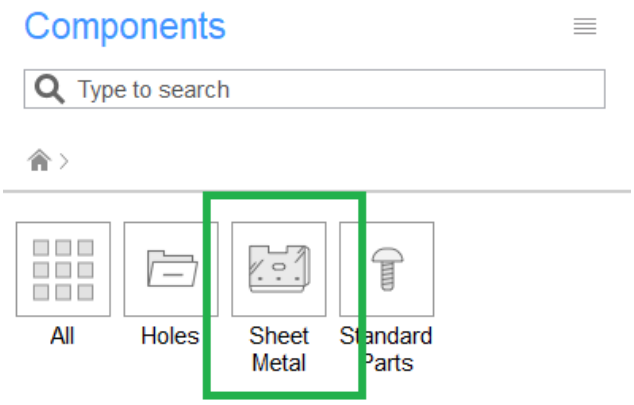


Figura 91: Configurador de Filtros

O Configurador de Filtros permite definir um conjunto de condições. Cada condição define uma relação entre uma propriedade selecionada e um valor de destino.

As relações possíveis são:

- Igual a
- Não igual a
- Menor que
- Maior que
- menor que ou igual a
- maior que ou igual a

As condições podem ser adicionadas a um grupo, que define como essas condições são tratadas. Existem 2 tipos de grupos:

- Grupo **Todo de**: é "verdadeiro" se todas as condições desse grupo forem "verdadeiras". Isso é o mesmo que o operador "and" (e).
- Grupo **Qualquer de**: é "verdadeiro" se pelo menos uma condição nesse grupo resultar em "verdadeira". Isso é o mesmo que o operador "or" (ou).

Um grupo pode conter um número arbitrário de condições. Este também pode conter grupos aninhados, tornando assim possíveis filtros complexos.

Para adicionar uma condição:

- Selecione uma propriedade na lista de propriedades.
- Selecione um operador de relação necessário e especifique o valor de destino.

  Para valores de texto, um valor de destino também pode ser selecionado na lista de todos os valores possíveis. Para valores numéricos, sua unidade de destino também pode ser especificada para converter automaticamente o valor em relação à unidade padrão do documento.

Para valores de texto, relações de "Igual a" e "Não igual a" suportam caracteres curinga.

O Configurador de filtros converte automaticamente o conjunto de condições em uma expressão de filtro, que pode ser visível clicando em :

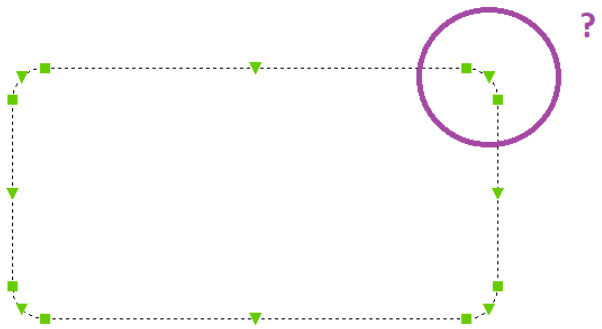


Figura 92: Expressão

Esta expressão é totalmente editável e bidirecional associativa com o conjunto de condições.

Descrição da sintaxe:

Tabela 25: Operadores relacionais:

Operador	Significado
==	igual a
!=	não igual a
<	menor que

Tabela 25: Operadores relacionais:

Operador	Significado
>	maior que
<=	menor que ou igual a
>=	maior que ou igual a

- Nomes das propriedades devem estar entre aspas.
- Operadores relacionais podem ser combinados usando operadores "and" e "or". Operador "and" (e) resultará em "true" (verdadeiro) se ambos os lados do operador forem verdadeiros. Operador "or" (ou) resultará em "falso" se um dos lados do operador for verdadeiro.
- Operadores lógicos "and" (e) e "or" (ou) diferenciam maiúsculas de minúsculas.
- Use "("and")" para combinar dois operadores em um.
- Valores de strings (sequências) são colocados entre aspas duplas ""
- Operadores Relacionais para strings diferenciam maiúsculas de minúsculas.
- Operadores "==" e "!=" suportam curingas.

Por exemplo, para obter uma Lista BOM de todas as peças padrão, use a seguinte expressão:

"É padrão"=="Sim"

Para obter uma lista BOM de todas as peças padrão ISO, use a seguinte expressão:

"É padrão"=="Sim" and "Padrão da peça"=="ISO"

E para obter uma Lista BOM de todas as peças padrão ISO ou DIN, use a seguinte expressão:

"É padrão"=="Sim" and ("Padrão da peça"=="ISO" or "Padrão da peça"=="DIN")

Curingas

Os curingas permitem que você combine valores de propriedade com um template específico. Por exemplo, se você quiser encontrar todos os componentes que contêm "nozzle" em seus nomes, uma expressão a seguir pode ser usada: "Nome do componente"=="*nozzle*".

Curingas também podem ser usados como valores-alvo no Configurador de Filtros.

Somente operadores relacionais "Igual a" e "Não igual a" suportam caracteres curinga.

Nota: Curingas no filtro principal diferenciam maiúsculas de minúsculas. Para Filtros de coluna, você pode especificar se precisa ou não diferenciar entre maiúsculas e minúsculas.

Os seguintes símbolos especiais são suportados:

Tabela 26: Símbolos especiais

Nome	Caractere	Descrição
Aspas simples	`	Escape o próximo caractere (toma-o literalmente).
Vírgula	,	Separa padrões.
Asterisco	*	Compara qualquer sequência de caracteres, incluindo uma sequência nula, em qualquer lugar.
Ponto de Interrogação	?	Compara qualquer caractere individual.
Ponto final	.	Compara a um único caractere não alfanumérico.
Sinal de arroba	@	Compara a um único caractere alfabético.
Hash tag	#	Compara a um único caractere numérico.
Til	~	Se o primeiro caractere de um padrão corresponder a algo que não seja o padrão.
Colchetes	[...]	Compara a um único caractere delimitado.
Til entre chaves	[~...]	Compara a um único caractere não-delimitado.
Hífen	-	Entre parênteses, compara a um único caractere no intervalo especificado.

Filtro de coluna

Filtros de coluna foram introduzidos no BricsCAD V21. Esses permitem que você defina rapidamente as condições que as linhas devem exibir, dependendo do conteúdo da coluna (um pouco semelhante aos filtros em alguns processadores de planilha). Ao contrário do filtro principal, esses são aplicados depois que a tabela BOM foi gerada. Portanto, é possível usar filtros de coluna para colunas com conteúdo agregado, como quantidades.

Cada coluna pode ter apenas um filtro. No entanto, você pode adicionar filtros de coluna a quantas colunas precisar. Somente as linhas que passam por todos os filtros de coluna são exibidas na BOM final.

Você pode configurar filtros de coluna usando a caixa de diálogo **Filtro de Coluna** no Gerenciador de BOM:

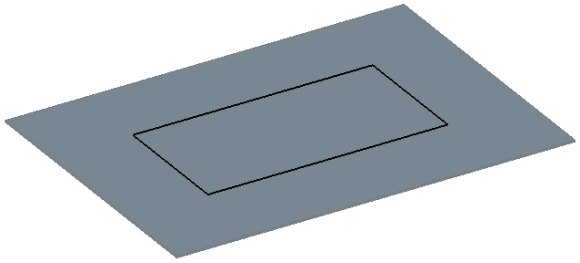


Figura 93: Filtro Rápido

Nesta caixa de diálogo, você pode selecionar o tipo de filtro na lista e configurá-lo de acordo. A lista de possíveis filtros depende do tipo de conteúdo da coluna, como é mostrado na tabela abaixo:

Tabela 27: Filtros

Tipo de filtro	Tipo de coluna	Descrição
sempre aceito	qualquer	todos os valores de coluna são aceitos
igual a	numérico	valor deve ser igual ao destino
não igual a	numérico	valor não deve ser igual ao destino
menor que	numérico	o valor deve ser menor que o destino
maior que	numérico	o valor deve ser maior que o destino
menor que ou igual a	numérico	valor deve ser menor ou igual ao destino
maior que ou igual a	numérico	valor deve ser maior ou igual ao destino
entre	numérico	valor deve estar no intervalo dado, ambas as extremidades estão incluídas
não no intervalo	numérico	valor não deve estar no intervalo dado, ambas as extremidades excluídas
corresponde exatamente	texto	valor deve corresponder exatamente ao destino
não corresponde	texto	valor deve ser diferente do destino
começa com	texto	valor deve começar com o texto dado
não começa com	texto	valor não deve começar com o texto dado

Tabela 27: Filtros

Tipo de filtro	Tipo de coluna	Descrição
termina com	texto	valor deve terminar com o texto fornecido
não termina com	texto	valor não deve terminar com o texto fornecido
contém	texto	valor deve conter o texto fornecido
não contém	texto	valor não deve conter o texto fornecido
corresponde ao padrão	texto	o valor deve corresponder ao modelo de curinga fornecido
não corresponde ao padrão	texto	o valor não deve corresponder ao template curinga fornecido
em	texto e numérico	valor deve estar incluído no conjunto dado de valores
não em	texto e numérico	valor deve estar incluído no conjunto dado de valores

Para todos os filtros de texto, você também pode especificar se eles devem diferenciar maiúsculas de minúsculas ou minúsculas.

21.4.11.6 Ordenar uma tabela BOM

Existem 3 modos de ordenamento para tabelas de BOM:

- **Ordem automática**:

  O conteúdo da tabela será ordenado como segue: nome do componente, nome do material físico, espessura (se aplicável), parâmetros (em ordem alfabética), e quantidade.
- **Ordem natural**:

  O conteúdo da tabela não é classificado. Em vez disso, todos os componentes estão na ordem de seu aparecimento no documento. Na maioria dos casos, as peças adicionadas anteriormente vão preceder as peças adicionadas depois.
- **Ordem personalizada**:

  O conteúdo da tabela é classificado de acordo com as colunas que você selecionou. Se nenhuma coluna for especificada, a ordem natural será usada.

A partir do BricsCAD V21, as tabelas BOM usam a chamada "classificação natural" do texto. Isso significa que, se o texto contiver uma mistura de letras e números, este será classificado em relação a esses números. Por exemplo, strings "N1", "M2", "N11", "N2", "AA1" será classificado como "M2", "N1", "N2",

"N11", "AA1." Portanto, é possível atribuir números personalizados com vários prefixos para componentes, usando parâmetros personalizados, e a seguir classificar as tabelas BOM por esses números na ordem correta.

21.4.11.7 Colunas avaliadas

As colunas avaliadas são colunas especiais, das quais os resultados dependem de outras colunas usando expressões definidas pelo usuário. A partir do BricsCAD V21, são suportados 2 tipos de colunas avaliadas: colunas de **fórmula** e colunas de **template**.

Colunas de fórmula

Uma Lista de Materiais suporta campos de fórmula. A mesma sintaxe é usada no comando *EXTRACAODADOS*. **FormulaExpression** é a expressão no formato de campos de fórmula **\AcExpr**. Esta pode conter constantes como PI ou funções como SIN (seno) (consulte Usar Expressões).). Um operando em uma expressão pode ser um **TituloColuna** colocado entre colchetes de seta (<>).

Por exemplo, fórmula *=<Mass>/<Volume>* calcula a densidade média do componente.

Se uma linha não contiver um valor, um valor padrão será usado. É possível especificar o valor padrão para um operando usando a seguinte sintaxe: *<"TituloColuna",ValorPadrao>* onde **TituloColuna** é o título da coluna exigida, e **ValorPadrao** é o valor padrão. **TituloColuna** deve estar entre aspas duplas ("). Se nenhum valor padrão estiver especificado, um valor vazio será usado, o que na maioria dos casos resulta em uma saída vazia.

Uma coluna de fórmula pode conter múltiplas expressões. Cada expressão deve ser incluída por colchetes de soma ({}). Essas expressões são avaliadas da primeira para a última, e a primeira que produz uma saída não-vazia é usada.

A partir do BricsCAD V21, uma fórmula pode se referir a outras colunas de fórmula, desde que elas não formem um loop (referência circular). Em uma BOM hierárquica, uma fórmula também pode se referir a valores da linha-pai, adicionando *Pai:* antes do nome da coluna. Esta também pode se referir a propriedades que não estão explicitamente adicionadas à tabela, mas todas as outras colunas de fórmula referidas na fórmula devem existir na tabela. Você ainda pode usar essas colunas exibidas como invisíveis na tabela BOM em expressões de fórmula.

Para propriedades de distância, área, volume e massa, os valores serão convertidos para a unidade **definida-pelo-usuário**, antes de usá-los na expressão da fórmula. Se a unidade **definida-pelo-usuário** não estiver definida, será usada a unidade do documento. Para a propriedade de massa, quilograma será usado em documentos métricos e libras (pound) em documentos imperiais.

Colunas de template

As colunas de template permitem combinar várias propriedades, bem como um texto personalizado, em uma única coluna. Uma sintaxe especial, baseada em expressões Diesel, é usada para definir colunas de template. Expressões para colunas de template devem começar com o sinal **$**. Um operando em uma expressão pode ser um **TituloColuna** entre colchetes de seta (<>). Também é possível usar funções especiais:

Tabela 28: Funções especiais

Função	Descrição
$(+, arg1, arg2[, ... , arg9])	Adiciona os argumentos fornecidos ao primeiro.
$(-, arg1, arg2[, ... , arg9])	Subtrai do primeiro os argumentos fornecidos.
$(*, arg1 , arg2[, ... , arg9])	Multiplica o primeiro argumento pelos argumentos fornecidos.
$(/, arg1, arg2[, ..., arg9])	Divide o primeiro argumento pelos argumentos fornecidos.
$(=, arg1, arg2)	Retorna 1 se arg1 for igual a arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser números.
$(<, arg1, arg2)	Retorna 1 se arg1 for menor que arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser números.
$(>, arg1, arg2)	Retorna 1 se arg1 for maior que arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser números.
$(!=, arg1, arg2)	Retorna 1 se o arg1 não for igual ao arg2; caso contrário, retorna 0. Tanto o arg1 quanto o arg2 devem ser números.
$(<=, arg1, arg2)	Retorna 1 se arg1 for menor ou igual a arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser números.
$(>=, arg1, arg2)	Retorna 1 se arg1 for maior ou igual a arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser números.
$(and, arg1, arg2[, ... , arg9])	Aplica a operação AND bit-a-bit aos argumentos
$(angtos, angle[, mode, precision])	Converte o ângulo para string em relação ao modo dado (0 - graus, 1 - graus/min/seg, 2 - grados, 3 - radianos) e precisão.
$(delim, delimiter_string)	Insere um delimitador inteligente O delimitador de string será adicionada à saída somente se a saída antes e depois deste delimitador não estiver vazia.
$(edtime, time, format)	Formata a data de acordo com a string de formato dada
$(eq, arg1, arg2)	Retorna 1 se arg1 for igual a arg2; caso contrário, retorna 0. Ambos arg1 e arg2 devem ser texto.

Tabela 28: Funções especiais

Função	Descrição
$(eval, expression)	Avalia a expressão dada como uma expressão Diesel.
$(fix, value)	Trunca o valor real para o inteiro.
$(getvar, variable)	Retorna o valor da variável de sistema dada.
$(if, arg, then[, else])	Retorna 'then' se 'arg' não for 0; caso contrário, retorna 'else' ou string vazia se 'else' não for fornecido.
$(index, position, comma_separated_string)	Retorna a substring na posição dada na string separada-por-vírgulas.
$(nth, position, arg1[, arg2, ... , arg9])	Retorna o argumento na posição dada.
$(or, arg1, arg2[, ..., arg9])	Aplica operação OR bit-a-bit aos argumentos.
$(rtos, value[, mode, precision])	Formata o valor dado em relação ao modo e precisão necessários.
$(strlen, string)	Retorna o comprimento da string dada.
$(substr, string, start[, length])	Retorna a substring da 'string' de 'iniciar' do 'comprimento' dado. A numeração começa a partir de 1.
$(upper, string)	Converte 'string' em maiúsculas (caixa alta).
$(xor, arg1, arg2[, ... , arg9])	Aplica XOR bit-a-bit a todos os argumentos.

Uma coluna de template pode conter múltiplas expressões. Cada expressão deve ser incluída por colchetes de soma ({}). Essas expressões são avaliadas da primeira para a última, e a primeira que produz uma saída não vazia é usada.

A partir da V21, uma coluna de template pode se referir a outras colunas avaliadas, desde que essas não formem um loop. Em uma BOM hierárquica, uma coluna de modelo também pode se referir a valores na linha-pai adicionando *Pai:* antes do nome da coluna. Isto também pode se referir a propriedades que não são explicitamente adicionadas à tabela, mas todas as outras colunas avaliadas referidas em qualquer uma das expressões devem estar presentes na tabela. Você ainda pode usar colunas exibidas como invisíveis na tabela BOM em expressões de modelo.

Você pode usar a string "\n" para adicionar uma quebra de linha.

Por exemplo, a expressão a seguir combina as propriedades PartNumber, Width, Height e Length:

${<PartNumber>$(delim," X ")<Width>$(delim," X ")<Height>$(delim," X ")<Lenght>}

Configurador de expressões

Você pode utilizar a caixa de diálogo *Configurador de Expressão* para configurar uma coluna avaliada. Esta caixa de diálogo está disponível no Gerenciador de Lista de Materiais.



Figura 94: Editor de modelos

Esta caixa de diálogo permite que você defina o título e a unidade da coluna, edite a expressão usando um editor com destaque de sintaxe e preenchimento automático, e veja uma tabela BOM com a coluna avaliada. Para uma coluna de template, a seção **Template** permite facilmente inserir, mover e excluir propriedades, assim como delimitadores e textos personalizados.

21.4.11.8 Unidades da coluna

Você pode ajustar o modo de **posicionar unidades** para colunas que têm unidades associadas. As seguintes opções estão disponíveis:

- **Não mostrar unidades**

  A tabela BOM não exibe unidades para esta coluna.
- **No título da coluna**,

  se todos os valores na tabela usarem a mesma unidade ou puderem ser convertidos em uma unidade comum, todos os valores serão convertidos para essa unidade comum. O nome da unidade é adicionado ao título da coluna. Você pode configurar a posição do título e da unidade usando a propriedade **Formato de título**. Usar *<Título>* para denotar o título, e *<Unidade>* para denotar a unidade. Por exemplo, o formato de título padrão é "*<Título>*, **<unit>**." Você pode usar "\n" para adicionar uma quebra de linha.

  A unidade comum é escolhida na seguinte ordem:

  a Uma unidade definida por você.

  b Se a propriedade for distância, área, volume ou massa, e o modo de unidade for **Melhor para todos os valores**, a melhor representação para a maioria dos valores será usada.

  c Se todos os valores tiverem a mesma unidade, essa será usada.
- **Na célula**

  O nome da unidade é adicionado ao valor. Valor e unidade são exibidos na mesma célula.

- **Em coluna adicional**

  O nome da unidade é colocado em uma coluna separada.

Você pode definir o **Modo de unidade** para valores de distância, área, volume e massa, para definir as unidades usadas para exibir um valor específico. Por exemplo, se houver massas indo de 1 grama até várias toneladas, escolha **Melhor para cada valor** para exibir cada massa convertida para a unidade de massa mais adequada. Para ver todas as massas em quilogramas, defina quilogramas como a unidade **definida pelo usuário** para a coluna. **Melhor para todos os valores** é o padrão.

Opções do **Modo de unidades**:

- **Não mostrar unidades**

  As unidades não são exibidas.
- **Melhor para todos os valores**

  A unidade com a melhor representação para a maioria dos valores é exibida. Somente as propriedades de dimensões, área, volume e massa são suportadas.
- **Definida-pelo-usuário**

  Você escolhe a unidade a ser exibida. Além das propriedades de dimensões, área, volume e massa, também é possível selecionar uma unidade necessária para propriedades do material, como densidade, condutividade térmica e calor específico.
- **Melhor para cada valor**

  A unidade mais adequada para cada valor é usada. O modo **No título da coluna** do **Posicionamento da unidade** muda automaticamente para o modo **Em coluna adicional**. Somente dimensões, área, volume e propriedades de massa são suportadas.

Nota: Se uma unidade definida-pelo-usuário for definida para uma coluna e essa coluna for usada em um campo de fórmula, os valores convertidos para essa unidade serão substituídos na fórmula. Caso contrário, a unidade do documento será usada.

21.4.11.9 Rodapé

A tabela de Lista de Materiais suporta um rodapé. Para definir quais colunas vão formar o rodapé, defina o **Tipo de rodapé** para as colunas necessárias:

- **Sem rodapé**

  O ID da coluna não será incluído na linha do rodapé.
- **Soma**

  A soma de todos os valores para cada instância de componente individual é exibida no rodapé.

  Nota: Este não é o equivalente à soma de todos os valores exibidos na tabela, pois o número de cada componente é usado. Para campos de fórmula, a soma dos valores para Quantidade=1 é usada.
- **Média**

  O valor médio, de todos os valores, para cada componente individual, é exibido no rodapé. Para campos de fórmula, a soma dos valores para Quantidade=1 será usada.
- **Mínimo**

O menor valor para cada componente individual é exibido no rodapé. Para campos de fórmula, a soma dos valores para Quantidade=1 será usada.

- **Máximo**

  O maior valor para cada componente individual é exibido no rodapé. Para campos de fórmula, a soma dos valores para Quantidade=1 será usada.

Se todas as colunas receberem o tipo de rodapé **Sem rodapé**, a linha de rodapé não será exibida.

21.4.11.1 Templates

0

Você pode salvar uma configuração de BOM como um template, e usar esta para criar novas tabelas de BOM. O arquivo template contém todos os dados necessários. Este inclui tipo de tabela, título, filtro e conjunto de colunas com todas as propriedades associadas. Você pode usar <NOME> no título como um espaço reservado para o nome real da montagem. Por exemplo, se o nome da sua montagem for 'Minha Montagem' e o título da BOM for especificado como Lista de Materiais para <NOME>, o título da BOM será 'Lista de Materiais para Minha Montagem', porque 'Minha Montagem' vai substituir <NOME>.

Por padrão, os arquivos template da BOM recebem a extensão .BOM.

Um arquivo template de BOM é uma extensão do arquivo Data Extraction Definition (.dxd). No entanto, você não pode usar um arquivo *.DXD como um template de BOM, porque alguns dados específicos do BOM estarão ausentes. Você não pode usar um arquivo *.BOM para definir um modelo de ExtracaoDados.

Você pode editar arquivos de template em qualquer editor de texto. No entanto, a Bricsys recomenda que use o comando *BMBOM* ou o gerenciador de BOM para criar e editar arquivos de template.

21.4.11.1 Status da BOM

1

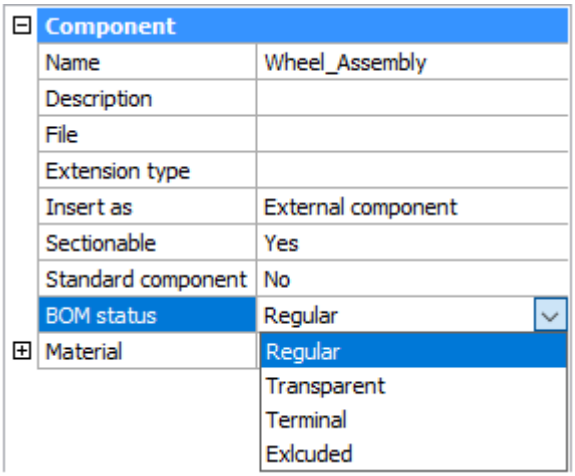


Figura 95: Pai

Cada componente tem uma propriedade de **Status BOM**. Você pode usar isso para controlar o comportamento do componente nas tabelas **Lista de Materiais**. Esta propriedade pode ter um dos seguintes valores:

- **Regular**

  O componente vai aparecer numa Lista de Materiais na mesma posição como está na estrutura da montagem e com o valor padrão do modo selecionado para a Lista de Materiais.
- **Transparente**

  O componente atua como um container transparente para seus subcomponentes. O componente não aparece na Lista de Materiais, mas subcomponentes podem aparecer na BOM, dependendo do status da BOM, da posição na estrutura de montagem, e do modo da tabela BOM.
- **Terminal**

  O componente é tratado como um componente de folha (não tem subcomponentes). Isso é útil para peças complexas ou componentes padrão.
- **Excluído**

  O componente e seus subcomponentes não vão aparecer na Lista de Materiais.

Para definir o Status da Lista BOM de um componente externo ou de raiz

1 Abra o arquivo que contém o componente.

2 Selecione o componente-raiz no Navegador de Mecânica.

3 Defina a propriedade **Status da BOM**.

Para definir o Status da lista BOM de um componente local

1 Selecione o componente no Navegador de Mecânica.

2 Clique o botão direito no nó selecionado para abrir o menu de contexto.

3 Selecione um status da lista BOM a partir do submenu de **Status da lista BOM**.



Figura 96: Submenu de status da BOM

Nota: Não é possível definir o status da BOM através do menu de contexto para componentes locais que são inseridos em um componente externo. Para fazer isso, você deve abrir o componente que contém o componente local.

21.4.11.1 Cantos fixos

2

A partir do BricsCAD V21, você pode definir qualquer canto da tabela como fixo. O canto fixo sempre manterá sua posição inicial durante a atualização do BOM. Quando a tabela tiver sido movida manualmente, a nova posição deste canto será mantida. Cantos fixos permitem alinhar tabelas BOM com outros objetos.

Você pode escolher qual canto fixar durante o posicionamento da tabela pelo comando *BMBOM* ou pelo gerenciador de BOM; use a tecla Tab para circular entre os cantos. Você pode alterar o canto fixo de uma tabela BOM no Gerenciador BOM.

21.4.12 Animações 3D Multi-Paramétricas

21.4.12.1 Visão Geral

Usar comandos: *ANIMEDITORABRIR* e *ANIMEDITORFECHAR*.

Você pode animar diversos parâmetros e programar movimentos da câmera com o painel Editor de Animação. Quando a animação estiver programada, você poderá reproduzi-la no BricsCAD ou renderizá-la como um vídeo 2D.

Cada animação de parâmetro tem sua própria "linha de tempo". Você pode adicionar quantas sequências de animação quiser a um parâmetro, desde que essas sequências não se sobreponham.

Há também uma linha do tempo especial para movimentos da câmera. Você pode adicionar quantas sequências de câmera quiser. Você pode definir o ponto de foco e a posição da câmera, tanto com uma polilinha como um ponto. Isso é semelhante ao comando *ANIMCAMINHO*.

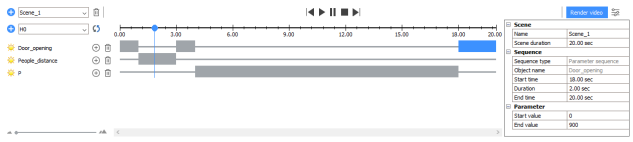


Figura 97: Linha de tempo

21.4.12.2 Cenas, Linhas de tempo e Sequências de Animação

Cenas

Uma cena consiste em uma ou mais **linhas de tempo** e cada uma dessas linhas de tempo tem uma ou mais **sequências de animação**. Quando o editor é iniciado pela primeira vez, este cria automaticamente uma cena. Uma cena define a duração da animação. Juntos, esses valores definem a **programação de animação**, usado para reprodução e renderização. Atualmente, você só pode renderizar a cena atual com o Editor de Animação. No entanto, você pode ter muitas cenas no mesmo documento e alternar entre elas.

Sequências de animação

Uma sequência de animação é a base de uma animação. Representa a animação única e contínua de um objeto. Uma sequência de animação tem associadas uma 'hora de início' e 'duração'. Este também tem

valores específicos para o objeto animado, como valores inicial e final para parâmetros, ou caminhos de câmera e ponto de vista, para sequências de câmera:

- **Parâmetro**

  Uma sequência de animação representa a mudança contínua de um parâmetro de um valor inicial para um valor final. Por exemplo, numa restrição de distância entre duas faces: a animação moverá uma face para um valor de distância máximo e mínimo.
- **Câmera**

  Uma sequência de animação corresponde a um movimento contínuo da câmera, em um caminho. Você pode ajustar vários movimentos da câmera com a 'linha de tempo' da câmera.

Linhas de tempo

Uma linha de tempo é uma representação visual de uma sequência de animação para um objeto. A linha de tempo exibe como e quando os parâmetros serão animados. Cada parâmetro tem sua própria linha de tempo em uma cena. Sequências de animação não podem se sobrepor. Uma linha de tempo pode ser temporariamente desativada para ser excluída de uma animação. Todas as linhas de tempo são independentes. Isso significa que você pode animar muitos parâmetros ao mesmo tempo, e ver seus efeitos combinados.

21.4.12.3 Para abrir o Editor de Animação

Execute um dos seguintes procedimentos:

- Coloque o cursor sobre uma barra, clique o botão-direito e escolha **Editor de Animação** no menu de contexto.
- Clique na ferramenta **Editor de Animação** ( ) na seção **Vista** da Faixa de opções.
- Escolha **Editor de Animação** no menu **Vista**.
- Entre *AnimEditorAbrir* na linha de Comando.

21.4.12.4 A Interface do Editor de Animação

O Editor de Animação é um painel. Possui 3 áreas principais:

- **Barra de ferramentas**: Contém ferramentas para selecionar uma cena, ajustar uma animação, e renderizar um vídeo.
- **Área de Linhas de tempo**: adiciona e edita linhas de tempo e seqüências de animação.
- **Configurador**: Configure os parâmetros da cena atual e a sequência de animação selecionada.

BarraFerr

Figura 98: BarraFerr

A barra de ferramentas contém ferramentas para:

- Adicionar e remover cenas.
- Controlar a reprodução da animação.

- Definir a duração da cena atual.
- Adicionar uma linha do tempo da câmera.
- Iniciar uma renderização.

Tabela 29: BarraFerr

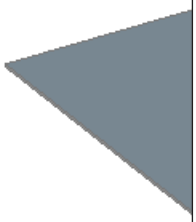
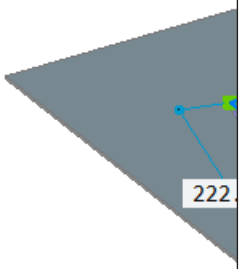
Ferramenta	Nome	Descrição
	Adic. Cena	Adiciona uma cena vazia com uma duração padrão
	**Seletor de Cena**	Seleciona a cena atual.
	Remover Cena	Remove a cena atual do documento.
	Para sequência anterior	Salta para o início da sequência de animação anterior
	Play	Inicia a animação
	Pausa	Pausa a animação
	Parar	Interrompe a animação e retorna o cursor para a posição inicial
	Para próxima sequência	Salta para o início da próxima sequência de animação

Tabela 29: BarraFerr

Ferramenta	Nome	Descrição
	**Render vídeo**	Abra uma caixa de diálogo para iniciar a renderização de um vídeo 2D.
	Abrir/fechar o configurador	Abre/fecha o painel de configuração

Area das Linhas de tempo

Use a **Area das Linhas de tempo** para adicionar e remover programas, para adicionar e remover sequências de animação, e para exibir a programação da animação. Clique em uma sequência de animação para selecioná-la, ou arraste o início ou o fim da barra para ajustar a duração. A sequência de animação selecionada é exibida em azul, outras sequências são exibidas em cinza. Cada linha do tempo na área de linhas de tempo pode ser temporariamente desativada para excluí-la da animação.

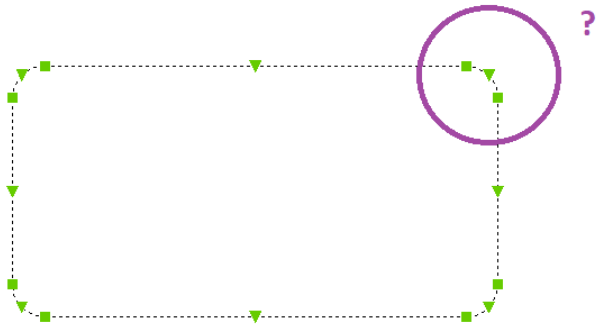


Figura 99: Area das Linhas de tempo

A **Area das Linhas de tempo** tem um cursor que representa a posição atual na animação. Quando a animação é reproduzida, o cursor representa a posição de reprodução atual. Você também pode mover manualmente o cursor para uma nova posição, a qualquer momento. Quando o cursor é movido, todos os parâmetros se alteram para corresponder ao quadro atual. A posição da câmera também será atualizada.

Para garantir que o estado do modelo corresponda à posição atual da animação, clique .

Nota: Você pode ajustar isso manualmente a qualquer momento, mas isso não vai afetar a animação.

Escala de Tempo

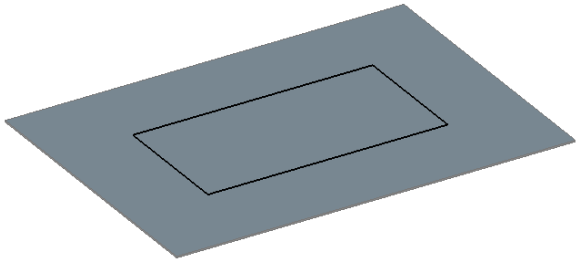


Figura 100: Escala de Tempo

Tabela 30: Escala de Tempo

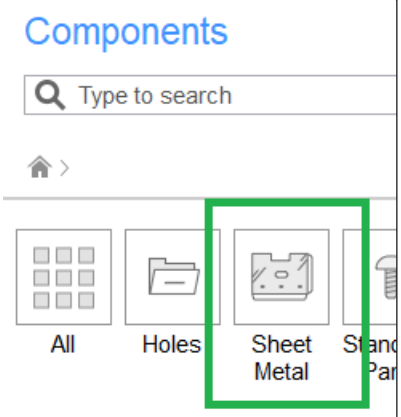
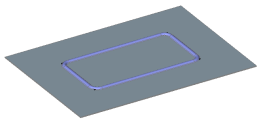
Ferramenta	Nome	Descrição
	Adic. Linha de tempo	Adiciona o parâmetro selecionado à linha de tempo
	**Seletor de Objeto**	Seleciona o objeto a ser adicionado à linha do tempo
	Atualizar modelo	Atualiza o estado do modelo com respeito à posição atual do cursor
	**Escala de Tempo**	Exibe as marcas de tempo ao longo da linha de tempo

Tabela 30: Escala de Tempo

Ferramenta	Nome	Descrição
	**Cursor**	Exibe a posição atual de reprodução da animação

Linha de tempo

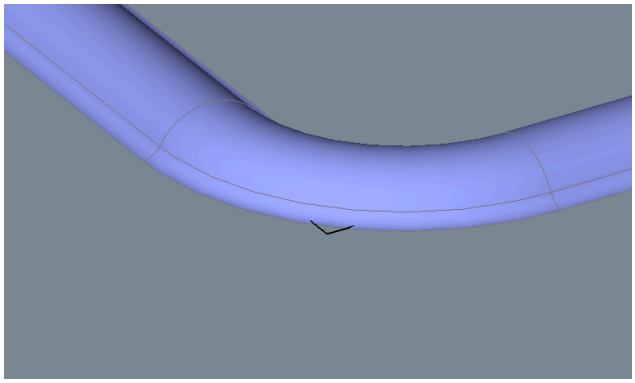


Figura 101: Linha de tempo

Tabela 31: Linha de tempo

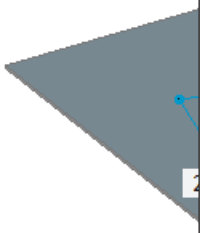
Ferramenta	Nome	Descrição
	**Linha de tempo Ativada**	Indica que a linha de tempo está ativada. Clique para desabilitar a linha de tempo.
	**Linha de tempo Desativada**	Indica que a linha de tempo está desativada. Clique para habilitar a linha de tempo.
	**Objeto da Linha de tempo**	Exibe o nome do objeto associado.

Tabela 31: Linha de tempo

Ferramenta	Nome	Descrição
⊕	**Adic. Sequência**	Adiciona uma nova sequência de animação, na posição do cursor
🗑	**Remover Linha de tempo**	Remove uma linha de tempo da animação.
▭	**Sequência de Animação**	Representa a duração e a posição de uma sequência de animação. Clique o botão esquerdo para selecionar a sequência.
▬	**Sequência de animação selecionada**	Representa a duração e a posição da sequência de animação selecionada. Você pode ver e alterar as propriedades no **Configurador**. Mova o mouse para o final da sequência, até que o cursor mude para uma seta de dupla face. Então clique o botão esquerdo e arraste para alterar a hora de início, hora de término e a duração. Clique o botão esquerdo do mouse no meio da sequência e arraste para uma nova posição na linha de tempo, sem alterar a duração.

Configurador

Acesse todas as propriedades da cena de animação e a sequência selecionada através do **Configurador**. Todas as propriedades estão agrupadas por objeto.

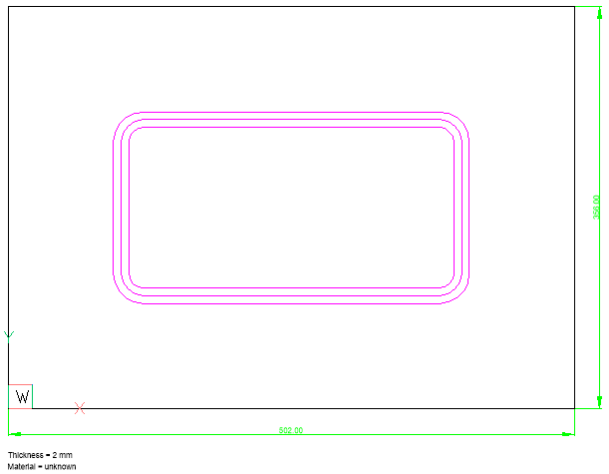
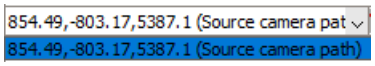
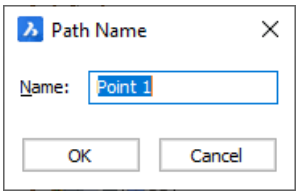


Figura 102: Configurador

- **Cena**: Propriedades da cena atual.
  - **Nome**: Nome da cena.
  - **Duração**: Duração da cena, em segundos.

- **Sequência**: propriedades da sequência selecionada.
  - **Tipo de sequência** (somente-leitura): este é o tipo da sequência.
  - **Nome do objeto** (somente-leitura): o nome do objeto controlado por aquela sequência.
  - **Hora de início**: Hora do início da sequência, em segundos. A duração será ajustada, o tempo de término permanecerá inalterado.
  - **Duração**: Duração da cena, em segundos. Quando alterado, a hora de término será ajustada em relação à hora de início e duração.
  - **Tempo final**: Hora de término da sequência, em segundos. Quando alterado, a duração será ajustada, a hora de início permanecerá inalterada
- **Parâmetro** (para sequências paramétricas)
  - **Valor inicial**: Valor inicial do parâmetro.
  - **Valor final**: Valor final do parâmetro.
- **Câmera** (para sequência da câmera)
  - **Tipo de caminho da câmera**: Tipo de entidade de caminho da câmera. Pode ter um dos seguintes valores:
    - **Ponto**: A câmera é fixada no ponto dado.
    - **Caminho**: A câmera se move ao longo do caminho dado.
  - **Ponto da câmera** (quando o **Tipo de caminho da câmera** está definido como **Ponto**). Os seguintes controles são exibidos:

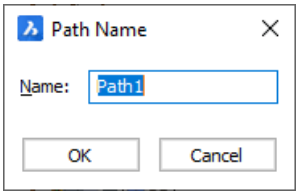
Tabela 32: Ponto da câmera

	Lista de pontos usados anteriormente. Selecione o ponto na lista suspensa, para usar um deles.
	Clique para selecionar o ponto no modelo. Depois que o ponto for selecionado, uma caixa de diálogo será exibida:  **Figura 103:** Nome do Caminho • **Nome**: Insira o nome do ponto. • **Ok**: Clique para aceitar a seleção do ponto • **Cancelar**: Clique para cancelar a seleção de pontos.
	Clique para remover o ponto da lista.

  - **Caminho da câmera** (quando o **Tipo de caminho da câmera** é definido como **Caminho**). Os

seguintes controles são exibidos:

Tabela 33: Caminho da câmera

	Lista os caminhos usados anteriormente. Selecione o ponto na lista suspensa, para usar um deles.
	Clique para selecionar o caminho no modelo. Depois que o caminho é selecionado, uma caixa de diálogo é exibida:  **Figura 104:** Nome do Caminho • **Nome**: Entre o nome do caminho • **Ok**: Clique para aceitar a seleção do ponto • **Cancelar**: Clique para cancelar a seleção de pontos.
	Clique para remover o caminho da lista.

- **Tipo de caminho de destino**: Tipo do caminho de destino. O alvo (destino) define o ponto em que a câmera vai olhar naquele determinado momento. Pode ter um dos seguintes valores:
  - **Ponto**: O alvo é fixado em um determinado ponto.
  - **Caminho**: O alvo se move ao longo de um determinado caminho.
- **Ponto de destino** (quando o **Tipo de caminho de destino** está definido como **Ponto**). Os seguintes controles são exibidos:

Tabela 34: Ponto destino

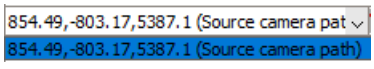
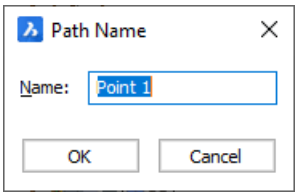
	Lista de pontos usados anteriormente. Selecione o ponto na lista suspensa, para usar um deles.

Tabela 34: Ponto destino

	Clique para selecionar o ponto no modelo. Depois que o ponto for selecionado, uma caixa de diálogo será exibida:  **Figura 105:** Nome do Caminho • **Nome**: Insira o nome do ponto. • **Ok**: Clique para aceitar a seleção do ponto • **Cancelar**: Clique para cancelar a seleção de pontos.
	Clique para remover o ponto da lista.

- **Caminho de destino** (quando o **Tipo de caminho de destino** está definido como **Caminho**). Os seguintes controles são exibidos:

Tabela 35: Caminho de destino

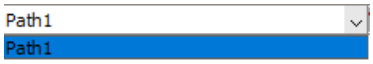
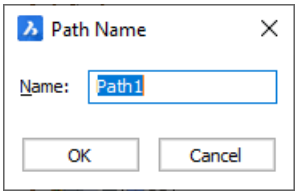
 	Lista os caminhos usados anteriormente. Selecione o ponto na lista suspensa, para usar um deles.
	Clique para selecionar o caminho no modelo. Depois que o caminho for selecionado, uma caixa de diálogo é exibida:  **Figura 106:** Nome do Caminho • **Nome**: Entre o nome do caminho • **Ok**: Clique para aceitar a seleção do ponto • **Cancelar**: Clique para cancelar a seleção de pontos.

Tabela 35: Caminho de destino

	Clique para remover o caminho da lista.

Nota: Semelhante a *ANIMCAMINHO*, todos os pontos e caminhos selecionados estão armazenados no desenho.

21.4.12.5 Reprodução da Animação (Play)

Use os controles de reprodução de animação para visualizar uma animação.

- Para **iniciar** uma animação, clique o botão **Play** (Reproduzir).

  A reprodução começa a partir da posição do cursor.
- Para **pausar** a animação, clique no botão **Pausa**.
- Para **interromper** a animação, e mover o cursor para o início da animação, clique no botão **Parar**.

Você pode saltar para a próxima ou para a anterior sequência de animação (em ordem cronológica) com as ferramentas **Ir para a Próxima Sequência** e **Ir para Sequência Anterior**, respectivamente.

Nota:

- Durante a reprodução, os parâmetros envolvidos na cena e a posição da câmera são alterados diretamente no desenho. Para modelos complexos, o recálculo do modelo pode levar uma quantidade significativa de tempo. Você pode usar o cursor para verificar os momentos-chave da animação em vez disso.
- Durante a reprodução, a câmera se moverá com todas as sequências de câmera definidas. Se não houver sequências de câmera, a vista atual será usada. O vídeo de saída é o mesmo que a viewport atual.
- Após a conclusão da reprodução, o modelo reverte automaticamente para o estado antes da animação ser iniciada. Em alguns casos, pode fazer com que o modelo esteja em um estado diferente do que a posição do cursor iria ditar. Para atualizar o modelo em relação à posição atual do cursor, clique  .

21.4.12.6 Render

1 Clique no botão **Renderizar vídeo** para renderizar a animação como um vídeo 2D.

  A caixa de diálogo *Renderizar vídeo* é exibida:

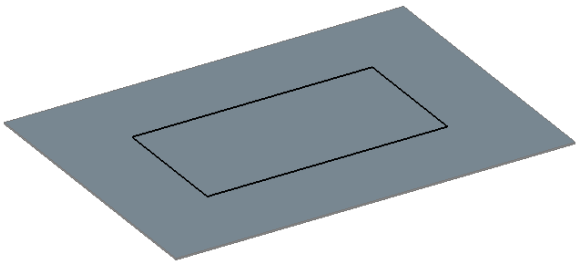


Figura 107: Render vídeo

2 Ajuste os parâmetros de renderização:
   - **Quadros/frames por segundo (fps)**: A taxa de quadros por segundo.
   - **Resolução**: A resolução do vídeo.
   - **Formato do arquivo**: O formato do vídeo (os formatos AVI, MPG e WMV são suportados).
   - **Tempo inicial**: A posição de tempo em que a renderização vai iniciar.
   - **Tempo final**: A posição de tempo em que a renderização vai terminar.

3 Clique no botão **Renderizar** para iniciar a renderização.

   Uma caixa de diálogo de arquivo é exibida:

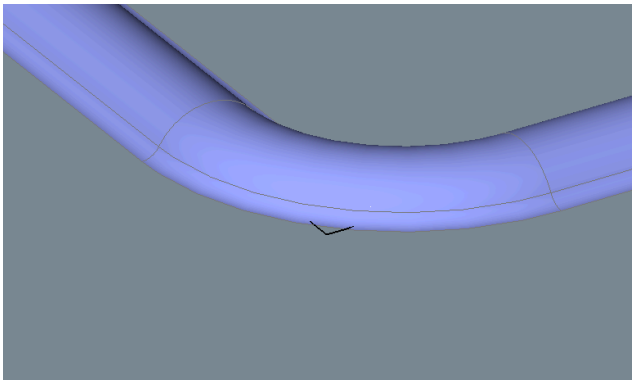


Figura 108: Salvar Vídeo Como

4 Especifique o local e o nome do arquivo de saída.

5 Clique em **Salvar** para iniciar a renderização.

   Uma vista prévia mostra o progresso.

21.4.12.7 Para adicionar uma nova cena

1 Clique no botão **Adicionar Cena**.

   A caixa de diálogo Adicionar cena exibe:

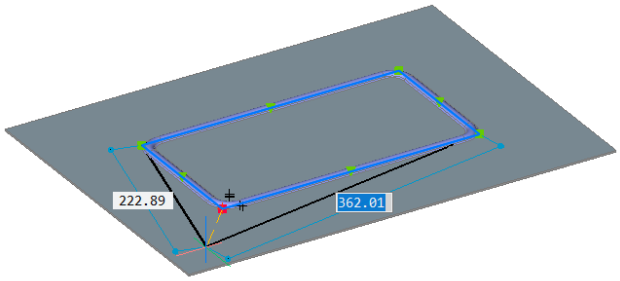


Figura 109: Adicionar cena

2 Entre o nome da cena.

3 Entre a duração da cena, em segundos.

4 Pressione **Adic.** para adicionar a cena.

21.4.12.8 Para remover uma cena

1 Selecione a cena com o **Seletor de Cena**.

2 Use a ferramenta **Remover Cena**.

21.4.12.9 Para adicionar uma animação de um parâmetro

1 Adicione o parâmetro desejado à linha de tempo:
   - Selecione o parâmetro com o **Seletor de Parâmetros**.
   - Use a ferramenta **Adicionar Linha de tempo** para adicionar uma nova linha do tempo.

2 Coloque o cursor na posição inicial desejada na **Escala de Tempo**.

3 Use a ferramenta **Adic. Sequência** para adicionar uma nova sequência de animação,
ou clique duas vezes na linha do tempo em que você deseja adicionar uma nova sequência.

4 Arraste para ajustar a duração desejada da sequência,
ou use o **Configurador** para definir a hora de início desejada, hora de término ou duração, em segundos.

5 Use o **Configurador** para definir os valores inicial e final desejados para o parâmetro.

21.4.12.1 Para adicionar um movimento da câmera 0

1 Se for aplicável, adicione uma linha do tempo da câmera selecionando **Câmera** no **Seletor de objetos**.

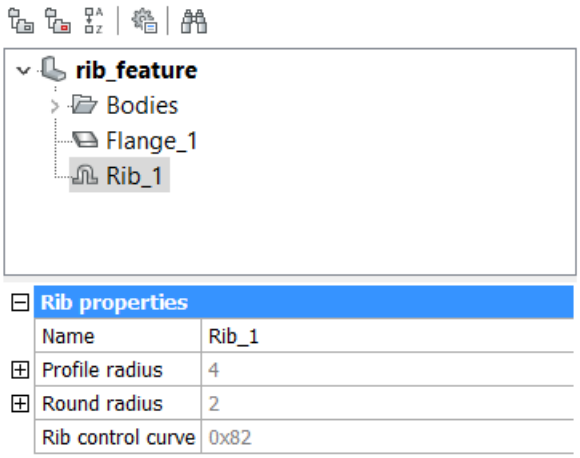
2 Coloque o cursor na posição inicial desejada na **Escala de Tempo**.

3 Use a ferramenta **Adicionar sequência** na linha do tempo da câmera para adicionar uma nova sequencia de câmera
ou clique duas vezes na linha do tempo em que você deseja adicionar uma nova sequência de câmera.

4 Clique e arraste para ajustar a duração desejada da sequência,
ou use o **Configurador** para definir a hora de início desejada, hora de término ou duração, em segundos.

5 Use o **Configurador** para definir o caminho da câmera:

- Escolha o **Tipo de caminho da câmera** dentre os seguintes:
  - Tara especificar que a posição da câmera deveria ser fixa, ou
  - **Caminho** para especificar que a câmera deverá se mover ao longo do caminho especificado.
- Use a propriedade Ponto da câmera para selecionar o ponto ou o **Caminho da câmera** para selecionar o caminho.
- Selecione a propriedade para edição, então clique



para selecionar o caminho ou o ponto no modelo

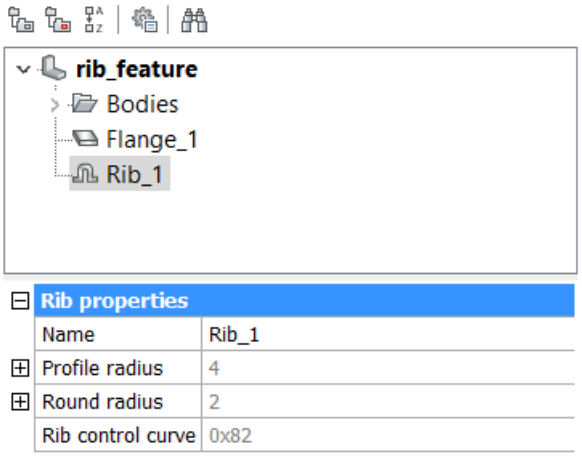
ou selecione um dos pontos ou caminhos anteriores na lista suspensa.

6 Use o **Configurador** para definir o caminho da câmera. O **alvo** é o ponto em que a câmera se concentrará.

Para escolher o **tipo de caminho de destino**, faça o seguinte:

- Especifique que o alvo deve ser fixado.
- **Caminho** para especificar que o destino deveria se mover ao longo do caminho fornecido.

Use a propriedade **Ponto de destino** para selecionar o ponto ou o **Caminho de destino** para selecionar o caminho:



Selecione a propriedade para edição e clique em
para selecionar o caminho ou o ponto no modelo
ou selecione um dos pontos ou caminhos anteriores, na lista suspensa.

21.4.12.1 Para remover uma sequência de animação 1

1 Clique o botão-direito com o cursor sobre a sequência.

2 No menu de contexto selecione **Excluir**.

Como alternativa, você pode:

1 Clique o botão esquerdo na sequência.

2 Clique a tecla **Delete**.

21.4.12.1 Para desativar ou ativar uma linha de tempo 2

1 Verifique o ícone de status da linha do tempo. Se o ícone estiver , a linha do tempo está desativada.
Se o ícone estiver , a linha do tempo está ativada.

2 Pressione o ícone de status da linha de tempo para alternar o status.

Sequências de animação em linhas de tempo desabilitadas não serão incluídas na animação.

21.4.12.1 Para animar uma vista explodida 3

Como o editor de animação não é suportado no Editor de Blocos ou modos de Editar referência, apenas vistas explodidas inseridas no Model Space podem ser animadas.

1 Insira a vista explodida no Model Space.

2 Se aplicável, suprima a vista explodida e o modelo, se você quiser ocultá-los ou mostrá-los durante a animação.

3 Selecione o objeto de exibição explodida no **Seletor de Objeto**.

4 Clique para adicionar uma linha do tempo.

5 Use a ferramenta **Adicionar sequência** na linha do tempo da vista explodida para adicionar uma nova sequência de vista explodida,

ou clique duas vezes na linha do tempo em que você deseja adicionar uma nova sequência.

6 Clique e arraste para ajustar a duração desejada da sequência,

ou use o **Configurador** para definir a hora de início desejada, hora de término ou duração, em segundos.

7 Se aplicável, use o **Configurador** para definir o passo inicial e o passo final da vista explodida, ou para habilitar a opção de auto-ocultar.

21.4.12.1 Para renderizar um vídeo

4

Veja a seção Renderizar.

21.5 Sheet Metal

21.5.1 Metodos de Projeto em Sheet Metal

Projeto de chapa metálica permite modelar as peças de chapa metálica, e gerar suas representações desdobradas com informações para fabricação.

Você pode criar peças complexas de chapa metálica com BricsCAD de forma fácil e rápida porque o processo de projeto é diferente de um processo de fabricação. Não pense em termos de uma folha planar que deve ser cortada e dobrada, mas modele sua parte diretamente à medida que você cria corpos sólidos com ferramentas de Modelagem Direta.

Tutorial: Apresentando o BricsCAD Sheet Metal

Tutorial: Projetar a partir do zero no BricsCAD Sheet Metal

Você pode criar peças de chapa metálica no BricsCAD em quatro diferentes maneiras (ou em qualquer combinação entre elas):

1 Criar uma flange base de um perfil planar fechado, e então adicionar flanges:
   - Puxe as bordas para criar flanges de borda adicionais com dobras.
   - Crie flanges parciais com criação automática dos alívios de dobras correspondentes.
   - Puxe várias arestas adjacentes para criar várias flanges ao mesmo tempo com criação automática de alívios de canto e junções entre as flanges.
   - Dobre uma flange para criar uma nova flange com uma curva a partir de sua porção.

2 Criar uma combinação de flanges e dobras em loft, conectando dois perfis 3D.

   Use a ferramenta Loft em Sheet Metal.

3 Criar uma combinação de flanges e dobras a partir de um perfil planar.

   Use a ferramenta Extrusão de Sheet Metal.

4 Criar a casca de um corpo, a partir de um sólido 3D.

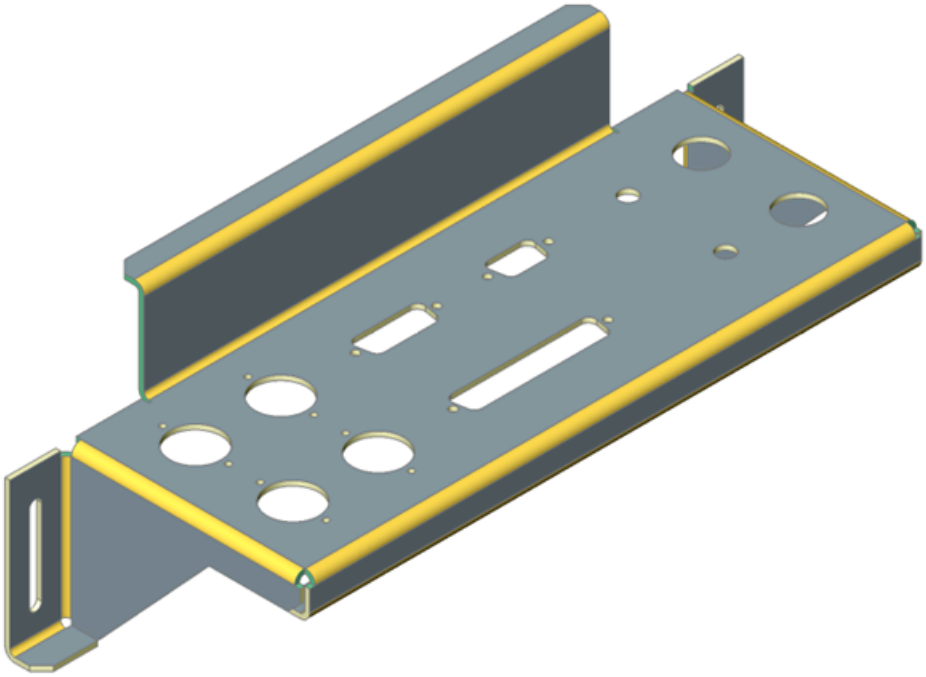
Use *Casca* a opção do comando *EDITARSOLIDO* com a espessura desejada, então converta este em uma peça de Sheet Metal com o comando *SMCONVERT*, e crie dobras e junções a partir de suas bordas rígidas.

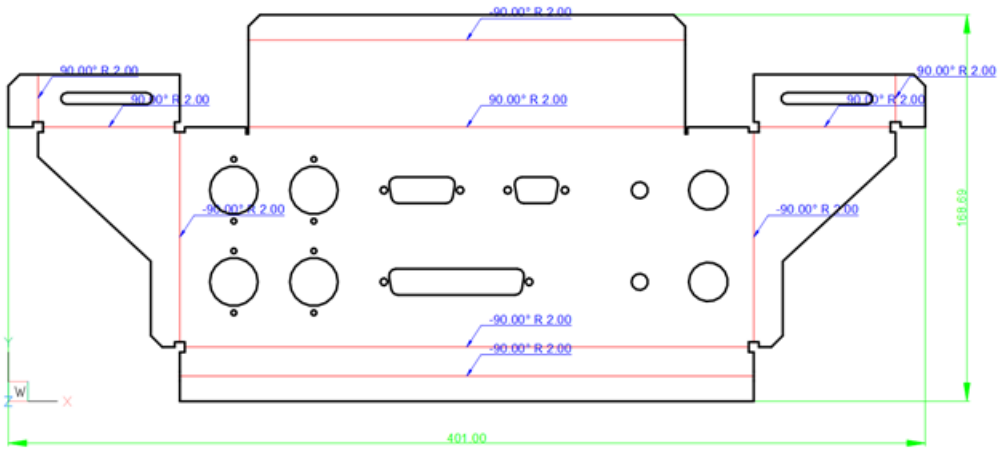
- Faça furos por desenhos com perfis fechados, nas faces das flanges, e empurre os desenhos através das flanges.
- Ajustar a espessura do material e os raios de dobra, usando parâmetros pré- definidos.
- Aplicar operações de Modelagem Direta e restrições 3D para ajustar ainda mais seu projeto, mantendo sempre a intenção de projeto nos recursos da chapa metálica.
- Gere automaticamente a representação desdobrada de sua peça e envie esta para um sistema de CAM, exportando o desenho 2D correspondente, com anotações das dobras, em um arquivo .dwg ou .dxf .

Você pode detalhar mais sua peça de chapa metálica, usando as seguintes operações:

- Faça furos por desenhos com perfis fechados, nas faces das flanges, e empurre os desenhos através das flanges.
- Inserir Recursos de Forma de uma biblioteca. Uma biblioteca de recursos de forma paramétricos é instalada com o BricsCAD Sheet Metal, e você pode criar seus próprios recursos de forma.
- Criar recursos de forma em nervura, a partir de uma curva 2D depositada sobre a flange, com uso do comando *SMRIBCREATE*.
- Ajustar a espessura do material e os raios de dobra, usando parâmetros pré- definidos.
- Aplicar operações de Modelagem Direta e restrições 3D para ajustar ainda mais seu projeto, mantendo sempre a intenção de projeto nos recursos da chapa metálica.

Quando a peça de chapa metálica estiver pronta, você pode desdobrá-la automaticamente e enviar o resultado para um engenheiro de produção, exportando o desenho 2D correspondente.





Tutorial: Processamento em Lote no BricsCAD Sheet Metal

21.5.2 Comandos e Barras de Sheet Metal

Tabela 36: Comandos e Barras de Sheet Metal

Ícone da barra Sheet Metal	Ícone da Barra desdobrável	Comando	Descrição
		*SMFLANGEBASE*	Cria a flange base (inicial) de uma peça de chapa metálica, a partir de um perfil planar fechado.
		*SMEXTRUDE*	Cria uma peça de chapa metálica extrudando uma polilinha na direção da normal ao plano da curva. A espessura é atribuída em relação às dimensões externas da polilinha.
		*SMLOFT*	Cria a peça de chapa metálica com dobras em loft e flanges, a partir de duas curvas não-coplanares.
		*SMCONVERT*	Reconhece automaticamente flanges, dobras e dobras em loft, em um sólido 3D.
		*SMFLANGEEDGE*	Cria uma ou mais flanges em uma peça de chapa metálica, puxando uma ou mais arestas de uma flange existente.
		*SMFLANGECONTOUR*	Cria uma nova flange a partir de um contorno fechado, e a anexa a uma peça de chapa metálica existente.
		*SMFLIP*	Modifica o lado de referência para flanges selecionadas, com um deslocamento opcional da geometria da flange pela sua espessura.
		*SMFLANGEROTATE*	Rotaciona uma flange.
		*SMHEMCREATE*	Cria uma ou mais bainhas Fechadas em uma peça de sheet metal, puxando uma ou mais bordas de uma flange existente.
		*SMHEMCREATE*	Cria uma ou mais bainhas Abertas em uma peça de sheet metal, puxando uma ou mais bordas de uma flange existente.
		*SMHEMCREATE*	Cria uma ou mais bainhas em formato de Lágrima em uma peça de sheet metal, puxando uma ou mais bordas de uma flange existente.

Tabela 36: Comandos e Barras de Sheet Metal

Ícone da barra Sheet Metal	Ícone da Barra desdobrável	Comando	Descrição
		*SMHEMCREATE*	Cria uma ou mais bainhas em formato Redondo em uma peça de sheet metal, puxando uma ou mais bordas de uma flange existente.
		*BMINSERT*	Insere recursos do forma.
		*SMFORM*	Cria um recurso de forma a partir de um determinado conjunto de faces.
		SMFORM	Explode o recurso de forma em flanges e curvas, se possível.
		*SMRIBCREATE*	Cria um recurso de forma associativo de Nervura em uma face de flange, a partir de uma curva planar 2D.
		*SMREPLACE*	Substitui recursos de forma (incluindo aqueles reconhecidos) em peças de Sheet Metal por recursos de forma de uma biblioteca interna, ou do usuário.
		*SMBENDCREATE*	Converte bordas rígidas (arestas alinhadas entre faces de flange) e junções em dobras.
		*SMBENDSWITCH*	Converte dobras para dobras em loft.
		*SMFLANGEBEND*	Dobra uma flange existente ao longo de uma linha, obedecendo o Fator-K para um dado raio de dobra.
		*SMJUNCTIONCREATE*	Converte arestas rígidas em junções.
		*SMJUNCTIONSWITCH*	Altera um recurso de junção simétrica para um com faces sobrepostas.
		*SMFLANGECONNECT*	Fecha lacunas entre duas flanges arbitrariamente orientadas.

Tabela 36: Comandos e Barras de Sheet Metal

Ícone da barra Sheet Metal	Ícone da Barra desdobrável	Comando	Descrição
		*SMTABCREATE*	Cria o recurso de aba única para uma peça de chapa metálica, trocando uma junção existente pela aba.
		*SMTABCREATE*	Cria o recurso de matriz de abas para uma peça de chapa metálica, trocando a junção existente por uma aba e ranhura.
		*SMRELIEFCREATE*	Faz alívios adequados para cantos e dobras. Os alívios de canto são construídos em cantos que têm três ou mais flanges adjacentes. Os alívios de dobra são criados no início e no fim de uma borda de flange.
		*SMRELIEFSWITCH*	Converte um alívio de canto em um alívio Circular.
		*SMRELIEFSWITCH*	Converte um alívio de canto em alívio Retangular.
		*SMRELIEFSWITCH*	Converte um alívio de canto em alívio de Tipo-V.
		*SMRELIEFSWITCH*	Converte um alívio de canto em alívio Suave.
		*SMRELIEFSWITCH*	Converte um alívio de canto em alívio Redondo.
		*SMRELIEFSWITCH*	Converte um alívio de canto em alívio de Rasgo.
		*SMSPLIT*	Divide uma flange ao longo de uma linha traçada na sua face.
		*SMSPLIT*	Divide uma flange por um vértice isolado no canto.
		*SMSPLIT*	Divide uma flange por um único vértice no canto e sugere automaticamente várias divisões de um tipo semelhante.

Tabela 36: Comandos e Barras de Sheet Metal

Ícone da barra Sheet Metal	Ícone da Barra desdobrável	Comando	Descrição
		*SMIMPRINT*	Divide faces de espessura de uma peça de chapa metálica ao estampar bordas, de acordo com flanges e dobras adjacentes.
		*SMDISSOLVE*	Remove os dados de Sheet Metal das faces selecionadas ou entidades.
		*SMDELETE*	Remove uma dobra ou uma junção ao restaurar uma borda rígida entre duas flanges; remove uma flange com todas as dobras adjacentes a esta. As flanges adjacentes são estendidas até uma configuração de junção, com a flange sendo excluída. Remove a mitra restaurando a geometria que foi cortada pelo recurso.
		*SMREPAIR*	Corrige problemas específicos de peças de Sheet Metal, tanto fluxos de trabalho globais quanto locais são suportados. Opcionalmente corrige recursos de dobra em loft: mescla dobras adjacentes e fornece conexão tangencial com flanges.
		*SMREPAIR*	Corrige problemas específicos das peças de Sheet Metal. Apenas a flange selecionada ou a dobra errada são reparadas.
		*SMSELECT* Mesmos recursos de forma	Seleciona todos os recursos de forma idênticos no modelo de Sheet Metal
		*SMSELECT* Arestas rígidas	Seleciona todas as bordas rígidas em peças de chapa metálica.
		*SMSELECT* Bordas da espessura não-ortogonal	Seleciona todas as bordas de espessura não-ortogonal na peça de chapa metálica.
		*SMSELECT* Bordas planas	Seleciona bordas planas na peça de Sheet Metal.

Tabela 36: Comandos e Barras de Sheet Metal

Ícone da barra Sheet Metal	Ícone da Barra desdobrável	Comando	Descrição
		*SMSELECT* lado da peça de chapa metálica	Selecione o lado da peça de chapa metálica.
		*SMPARAMETRIZE*	Cria um conjunto consistente de restrições 3D para a peça selecionada de Sheet Metal.
		*SMUNFOLD*	Desdobra o corpo da peça em Sheet Metal.
		*SMUNFOLD* Associativamente	Abre uma janela adicional com a representação planificada do modelo.
		*SMEXPORTOSM*	Exporta um sólido de Sheet Metal para o formato *.osm (Open Sheet Metal) formato de arquivo.
		*SMEXPORT2D*	Exporta a representação desdobrada de um corpo de Sheet Metal como um perfil 2D em formato *.dxf / *.dwg .
		*SMASSEMBLYEXPORT*	Executa uma exportação em lote de componentes da montagem, para desdobramento, criando um arquivo *.dxf para cada sólido 3D reconhecido como peça de Sheet Metal.
		*FEATURECOLORS*	Alternar a *FEATURECOLORS* variável de sistema.

As ferramentas de Sheet Metal estão disponíveis:

- No menu **Quad** (dependendo do recurso sob o cursor):

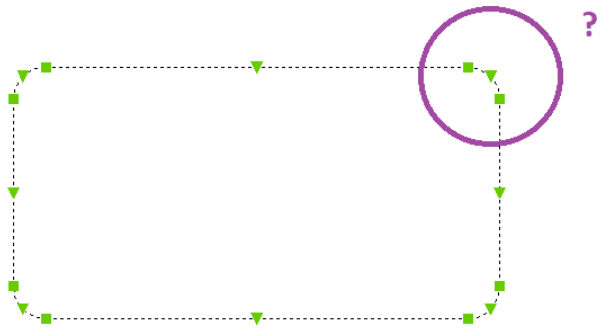
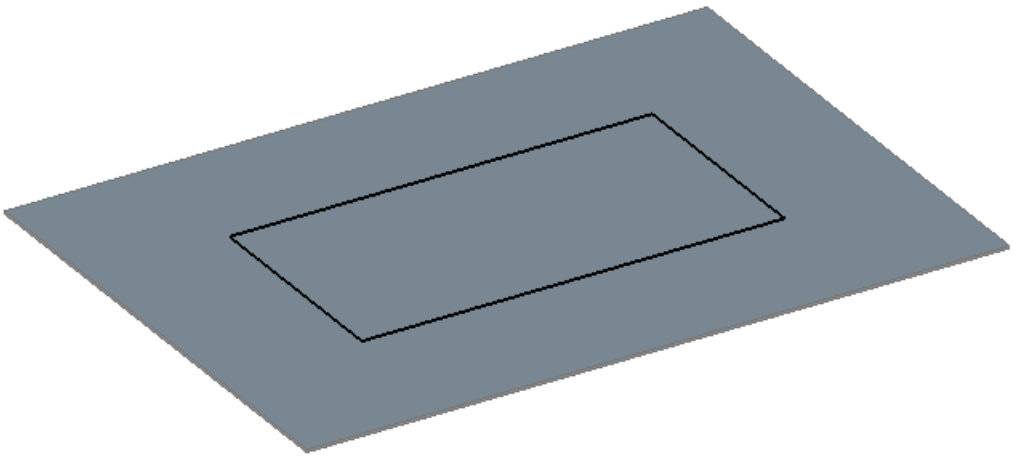


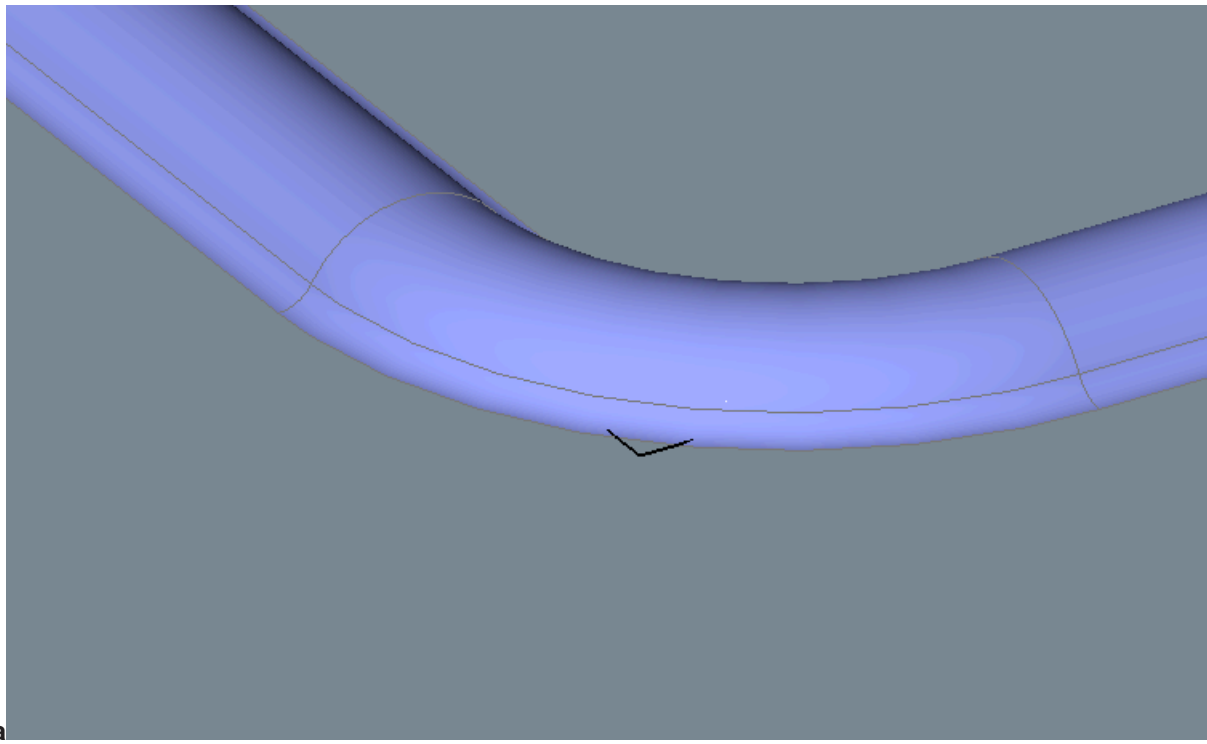
Figura 110: Menu de cursor Quad

- Nas barras de ferramentas **Sheet Metal**:
  - **Sheet**



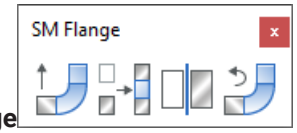
Metal

  - **SM**

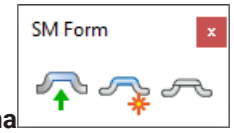


Dobra

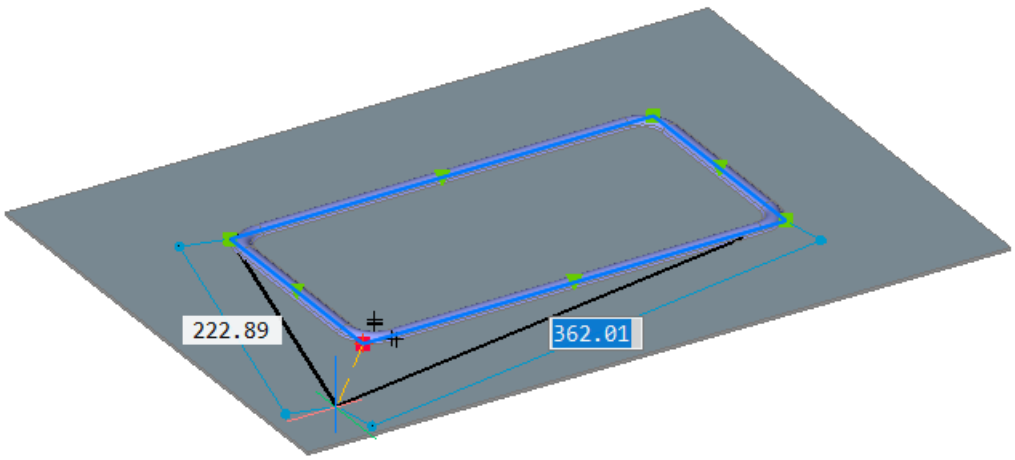
- **SM Flange**



- **SM Forma**

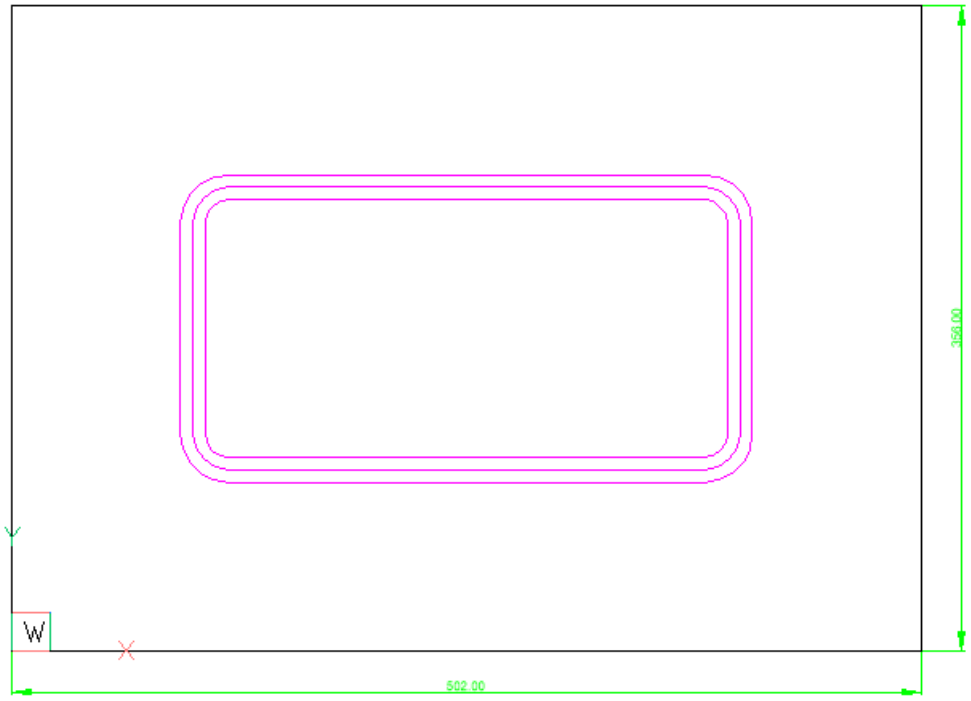


- **SM**



Bainha

- **SM**

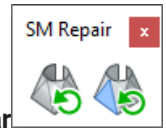


Junção

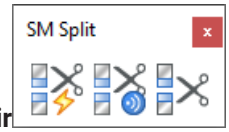
- **SM Alívio**



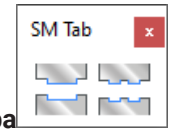
- **SM Reparar**

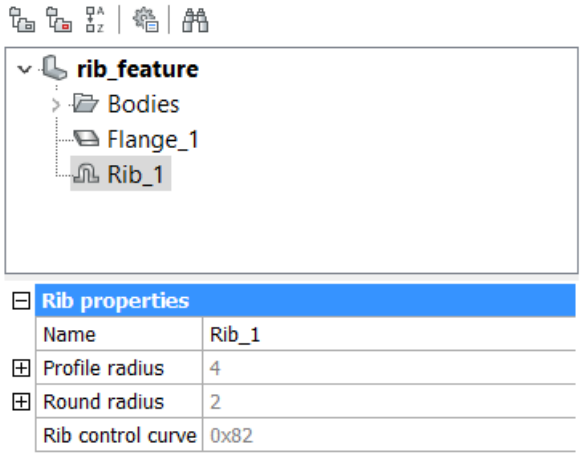


- **SM Dividir**



- **SM Aba**





- **SM Desdobrar**

- No menu **Sheet Metal** (apenas na área de trabalho **Mecânica**):

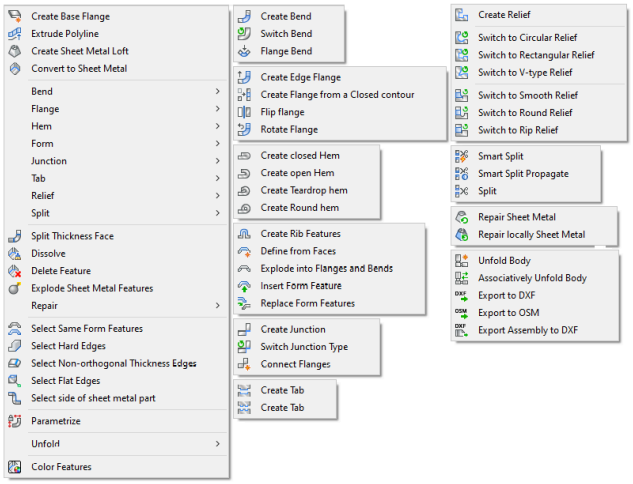
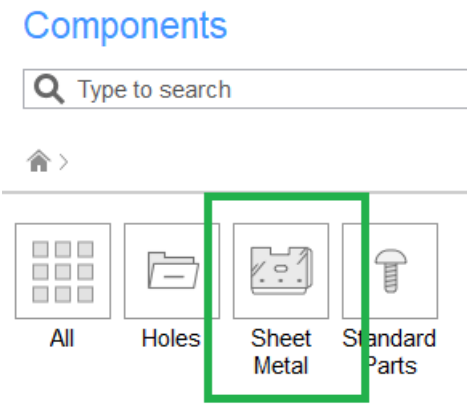


Figura 111: Menu de Sheet Metal

- Na aba **Sheet Metal**, na área de trabalho **Mecânica** da faixa de





opções.

Consulte também Modelagem direta ferram.

21.5.3 Recursos de Sheet Metal

21.5.3.1 Visão Geral

Os recursos são grupos inteligentes de faces em uma peça 3D. Cada recurso mantém uma relação espacial e paramétrica específica entre suas faces e algumas faces adjacentes. Os recursos permitem que você incorpore a intenção de projeto no seu modelo. Os recursos são criados automaticamente por uma operação geométrica específica que você aplica. As imagens abaixo mostram os recursos de Sheet Metal no BricsCAD:

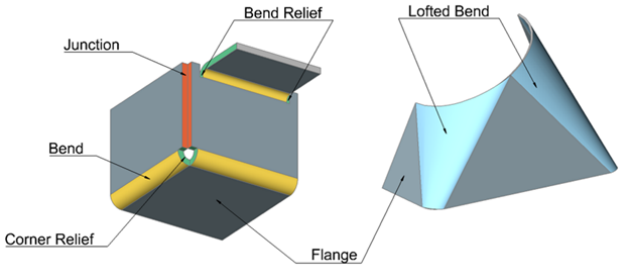


Figura 112: Exemplo 1 de recursos de chapa metálica



Figura 113: Exemplo 2 de recursos de chapa metálica

21.5.3.2 Flange

O principal recurso de todas as peças de chapa metálica é uma flange, que consiste em duas faces planares paralelas localizadas uma na frente da outra, com algum volume sólido entre essas, numa distância que é significativamente menor do que o tamanho linear das faces. Esta distância é igual à espessura do material. Quando você modifica o modelo, essa relação de distância é sempre mantida automaticamente. As demais faces, que são adjacentes às faces da flange e não pertencem a dobras, são chamadas de faces de espessura. Essas são sempre perpendiculares às faces da flange.

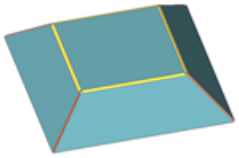
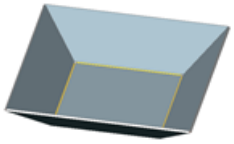
Os recursos de flange são criados pelos seguintes comandos: *SMFLANGEBASE*, *SMEXTRUDE*, *SMLOFT*, *SMFLANGEEDGE*, *SMFLANGEEND*, e *SMCONVERT*.

Se a *FEATURECOLORS* variável de sistema está *Ligada*, a cor da tela distingue entre a face de referência de uma flange e seu lado oposto. As modificações paramétricas de uma peça de chapa metálica, tal como

a alteração de espessura, não movem as faces de referência, se possível. O comando *SMFLIP* troca os lados de uma flange selecionada, de modo que faces de referência ficam então no outro lado geométrico da flange; opcionalmente, o comando desloca a flange sobre a espessura da peça de sheet metal.

As imagens abaixo ilustram a legenda de cores para as faces da flange da mesma peça

Tabela 37: Flange

	
Faces de Referência	Outras Faces

21.5.3.3 Dobra

Duas flanges são conectadas por uma dobra. Uma dobra consiste em duas faces cilíndricas coaxiais, que estão sempre tangentes às faces planas adjacentes das flanges.

Os recursos de dobra são criados pelos comandos *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND*, e *SMCONVERT*.

21.5.3.4 Dobra Zero

Este é um conceito de projeto de dobra, mas a geometria tem uma face cilíndrica para o lado externo da dobra com um raio igual à espessura. Em vez de uma face interna, há uma borda. Assim, efetivamente a geometria representa uma dobra de raio zero, que é usada para fins de projeto. A operação *SMUNFOLD* suporta dobras deste tipo e gera dimensões como se essas dobras fossem dobras regulares com um raio igual à espessura.

21.5.3.5 Dobra Errada

Os recursos de dobra incorretos são reconhecidos pelo comando *SMCONVERT* e refletem suposições sobre a intenção de design do usuário. Tais como dobras regulares, as dobras incorretas conectam flanges, mas suas faces podem ser não-coaxiais, ou podem ter conexões não-tangentes com as faces adjacentes de flange, o que impede usar a geometria como um recurso de dobra regular. Antes de desdobrar a peça, todas as dobras erradas devem ser reparadas pelo comando *SMREPAIR* ou removidas pelo *SMDELETE* e recriadas pelo *SMBENDCREATE*.

21.5.3.6 Dobras em Loft

Um recurso de dobra em loft consiste em duas superfícies regradas, localizadas uma na frente da outra (com deslocamento igual à espessura do material) com algum volume sólido entre elas. Uma superfície regrada pode ser descrita como o conjunto de pontos varridos por uma linha reta em movimento. Exemplos típicos são cilindros, cones, helicoides. Dobras em loft podem ser criadas pelo espessamento

de superfícies regradas. Uma dobra em loft pode conectar uma ou duas flanges, ou ser independente, sem flanges. Algumas combinações de adjacência entre dobras regulares e dobras em loft são também permitidas.

Diferenças entre dobras e dobras em loft:

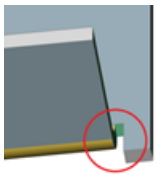
Tabela 38: Diferenças

	Dobras	Dobras em Loft
Superfície	Cilíndrica	Cilíndrica, cônica ou spline
Desdobrar	Exatamente	O desdobramento aproximado de dobras em loft é controlado por um número ajustável de operações de dobra, que são chamadas de Número de Subdivisões. -Quanto mais operações forem realizadas, mais suave será a peça real produzida e melhor aproximação será do design inicial. As dobras em loft cilíndricas e cônicas têm uma opção para desdobramento analítico exato.

Os recursos de dobra em loft são criadas pelos comandos *SMLOFT* e *SMCONVERT*.

21.5.3.7 Alívio de Dobra

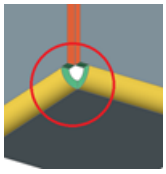
Um grupo de faces representando um corte tecnológico entre duas flanges de diferentes larguras, conectadas com uma dobra. O recurso de alívio de dobra mantém a distância entre duas faces opostas do corte.



Os recursos de alívio de dobra são criados pelos comandos *SMFLANGEEDGE*, *SMBENDCREATE* e *SMRELIEFCREATE*.

21.5.3.8 Alívio de Canto

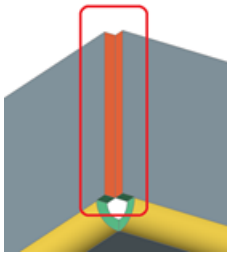
Um grupo de faces representando um corte no canto onde três ou mais flanges se encontram. Este corte é necessário para evitar a deformação indesejada do material durante a dobradura da peça de chapa metálica. Um recurso de alívio de canto mantém a forma e o tamanho desse corte.



Os recursos de alívio de canto são criados pelos comandos *SMFLANGEEDGE*, *SMBENDCREATE* e *SMRELIEFCREATE*.

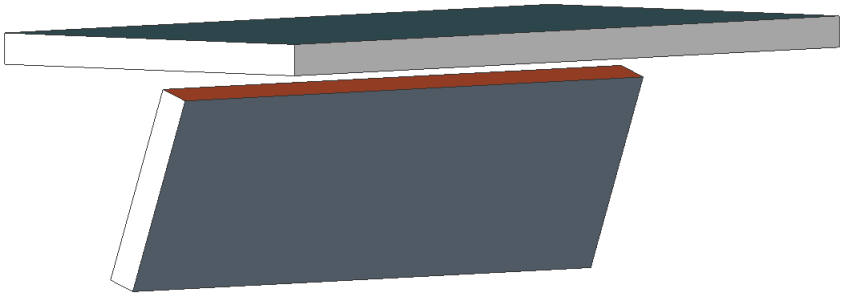
21.5.3.9 Junção e junção-T

Um recurso de junção consiste de duas faces de espessura, de flanges adjacentes, que não estão conectadas por uma dobra. Uma junção pode ser simétrica ou sobreposta. Os recursos de junção mantêm a distância mínima entre as flanges: a lacuna de junção.



Um recurso de Junção-T consiste em uma face de espessura de uma única flange, que está localizada perto de uma face de outra flange. Normalmente, uma Junção-T precede a criação de uma Aba-T.

Para esclarecer, a lacuna de junção na imagem abaixo é intencionalmente muito grande.

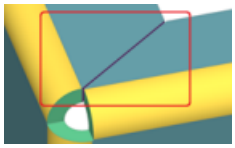


Os recursos de junção são criados pelos comandos *SMFLANGEEDGE*, *SMJUNCTIONCREATE*, e *SMFLANGECONNECT*.

Os recursos de Junção-T são criados pelos comandos *SMJUNCTIONCREATE* e *SMFLANGECONNECT*.

21.5.3.10 Recurso de Mitra (meia-esquadria)

Um recurso de miter representa uma divisão entre flanges coplanares criados pelos comandos *SMFLANGEEDGE* e *SMSPLIT*. Um recurso de miter (meia-esquadria) mantém a lacuna da meia-esquadria.



21.5.3.11 Recursos de Forma e de Nervura

Recursos de forma e nervuras representam uma rica variedade de formas que são produzidas por punção ou perfuração.

Duas representações dos recursos de forma são importantes: uma em 3D e um símbolo 2D na geometria desdobrada. No caso de geometria importada, o símbolo é obtido pela projeção das faces 3D do recurso de forma. Caso o recurso de forma tenha sido inserido/substituído na Biblioteca de Recursos de Forma, o símbolo de desdobramento é retirado do arquivo da biblioteca.

Nervuras representam uma estamparia na flange, feita por uma roda seguindo uma trajetória arbitrária.

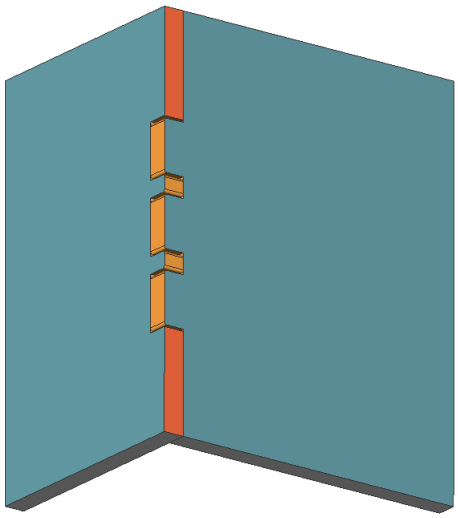
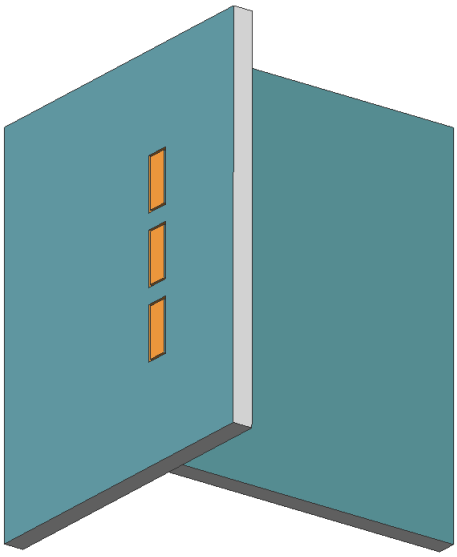
Recursos de forma e nervura são criados pelos comandos *BMINSERT*, *SMRIBCREATE* e *SMCONVERT*.

21.5.3.12 Aba e aba-T

Ambos recursos são usados para conectar 2 flanges, restringindo seu deslocamento relativo.

Os recursos Aba e Aba-T são criados pelo comando *SMTABCREATE*.

Tabela 39: Aba e aba-T

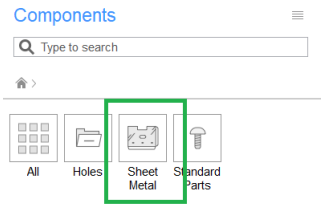
 **Figura 114:** Exemplo de aba	 **Figura 115:** Exemplo de Aba-T
Recurso de Aba	Recurso aba-T



21.5.3.13 Bisel

Recursos de Bisel representam chanfros nas faces de espessura das peças de Sheet Metal. Ambas as faces, inclinada e ortogonal definem um recurso de chanfro. Os recursos de Bisel devem ser compartilhadas estritamente por uma flange única, e não podem ser adjacentes aos recursos de dobra.

Os recursos de Bisel podem ser reconhecidos ou apagados pelo comando *SMCONVERT*, e são suportados pelo comando *SMUNFOLD*.



21.5.4 Colorir os Recursos

Para uma melhor experiência e fácil distinção entre os recursos, diferentes cores são aplicadas aos vários tipos de recursos na geometria. Essas cores são apenas para visualização e nunca serão mostradas em documentos gerados, vistas no paper space, inserções de bloco, etc.

The display of feature colors is controlled through the *FEATURECOLORS* 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 box.

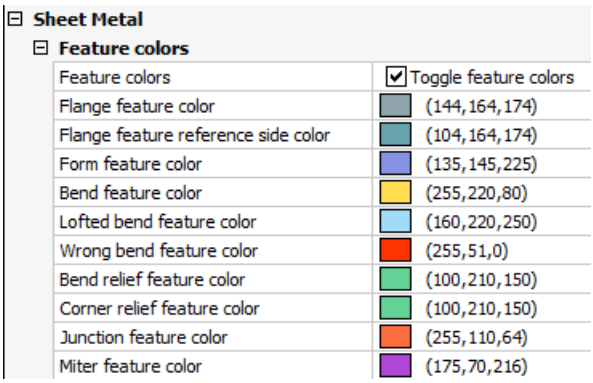


Figura 116: Sheet Metal

Para alternar a exibição de cores de recurso:

- Clique no botão **Recursos de Cor** ( ) na barra **Sheet Metal** .
- Clique no botão **Recursos de Cor** ( ) na seção **Sheet Metal** da Faixa de opções.
- Escolha **Recursos de Cor** no menu **Sheet Metal**.

Tutorial: Colorir os recursos em BricsCAD Sheet Metal

21.5.5 Navegador de Mecânica para Sheet Metal

21.5.5.1 Visão Geral

O Navegador de Mecânica abre automaticamente ao alternar para a área de trabalho **Mecânica**.

Para abrir manualmente o Navegador de Mecânica faça o seguinte:

1 Mova o cursor sobre uma barra, e clique o botão-direito.

Um menu de contexto é exibido

:

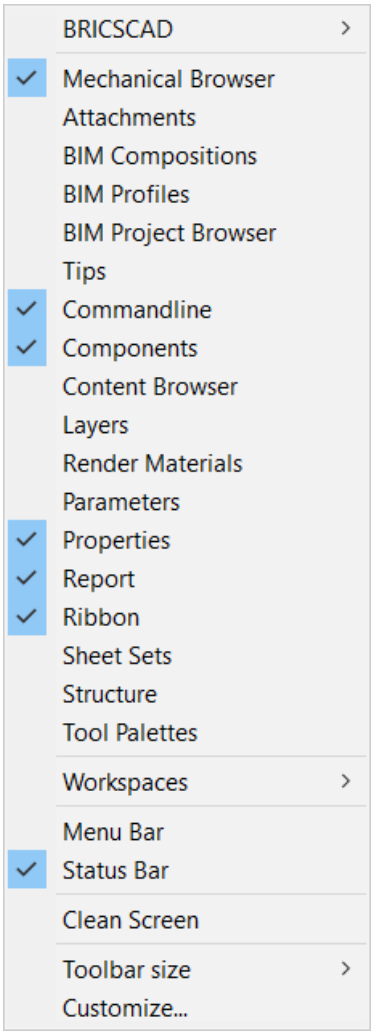


Figura 117: Menu de contexto

2 Escolha **Navegador de Mecânica**.

21.5.5.2 Contexto de Sheet Metal

Se você trabalha com uma peça de chapa metálica (se tiver criado algum recurso de sheet metal), as propriedades do nó raiz no **Navegador de Mecânica** contêm a seção "Sheet Metal" (também chamada Contexto de Sheet Metal associado ao documento).

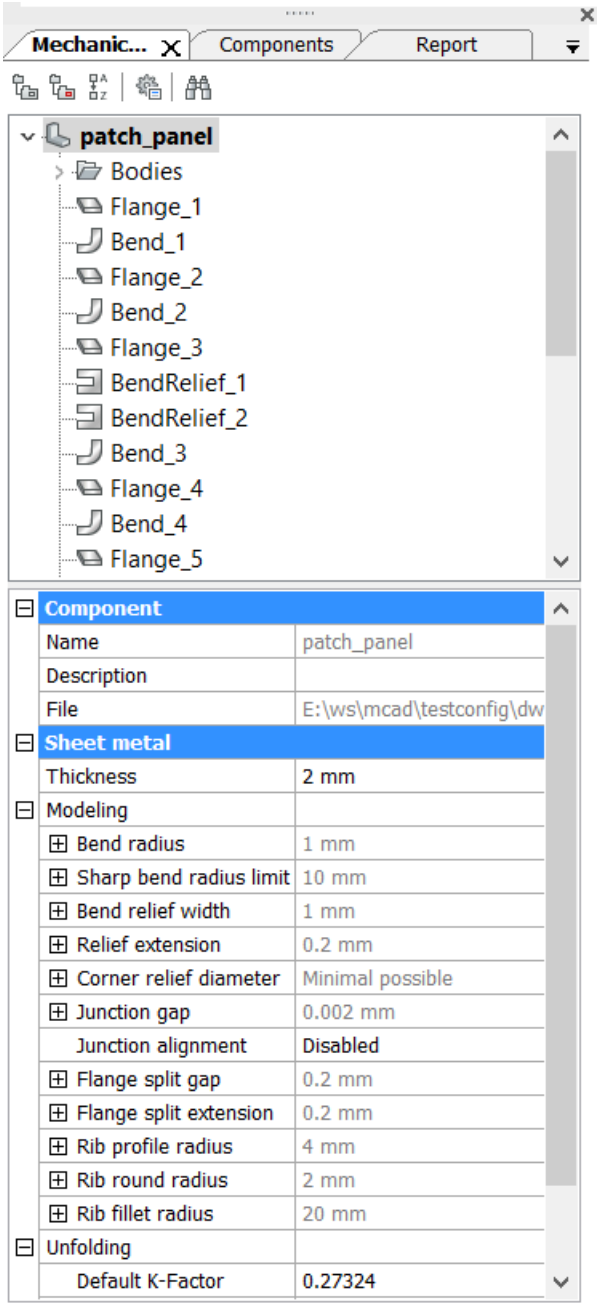


Figura 118: Navegador de Mecânica

As propriedades de Sheet Metal são representadas como campos agrupados.

O campo **Espessura** controla a espessura do material de sólidos em Sheet Metal que deve ser igual para todos os sólidos do documento.

O grupo Modelagem desempenha duas funções:

- Este fornece configurações padrão para Comandos de Sheet Metal, como *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND*, *SMLOFT*, e *SMCONVERT* em um modelo recém-criado. Os parâmetros iniciais são salvos no desenho, o que permite criar um conjunto de desenhos de template .dwt para cada tipo de modelo de Sheet Metal criados em uma empresa.
- Novos recursos criados pelos comandos geralmente derivam suas propriedades do Contexto de Sheet Metal. Por exemplo, o valor padrão da propriedade de **Tipo Raio** para o recurso de dobra é "Usar valor global". Assim, quando você altera o **Raio da dobra** no Contexto de Sheet Metal, todos os recursos de dobra com a propriedade derivada vão alterar seu raio. Assim, a segunda aplicação das propriedades do Contexto de Sheet Metal é que elas permitem alterar globalmente as propriedades da peça de chapa metálica.

O grupo de propriedades de **Desdobramento** no Contexto de Sheet Metal lista algumas propriedades dos comandos *SMUNFOLD*, *SMEXPORT2D*, e *SMASSEMBLYEXPORT*.

O Contexto de Sheet Metal é criado quando a primeira peça de chapa metálica aparece no documento, e é inicializado a partir das configurações listadas nos Parâmetros Iniciais. Fluxos de trabalho típicos para a criação de peças de chapa metálica estão listados nos Métodos de Projeto em Sheet Metal. Contexto de Sheet Metal é removido quando todos os recursos do documento são dissolvidos pelo comando *SMDISSOLVE*.

Nota: Observe que, se você selecionar todas as peças de chapa metálica no documento e Apagar essas, o Contexto de Sheet Metal é mantido, apesar de não mais haver sólidos de chapa metálica nem recursos de chapa metálica.

21.5.5.3 Trabalhar com Recursos

Todos os recursos de uma peça de chapa metálica estão listadas no **Navegador de Mecânica**. Quando você seleciona uma característica no **Navegador de Mecânica**, as faces desta são destacadas no modelo:

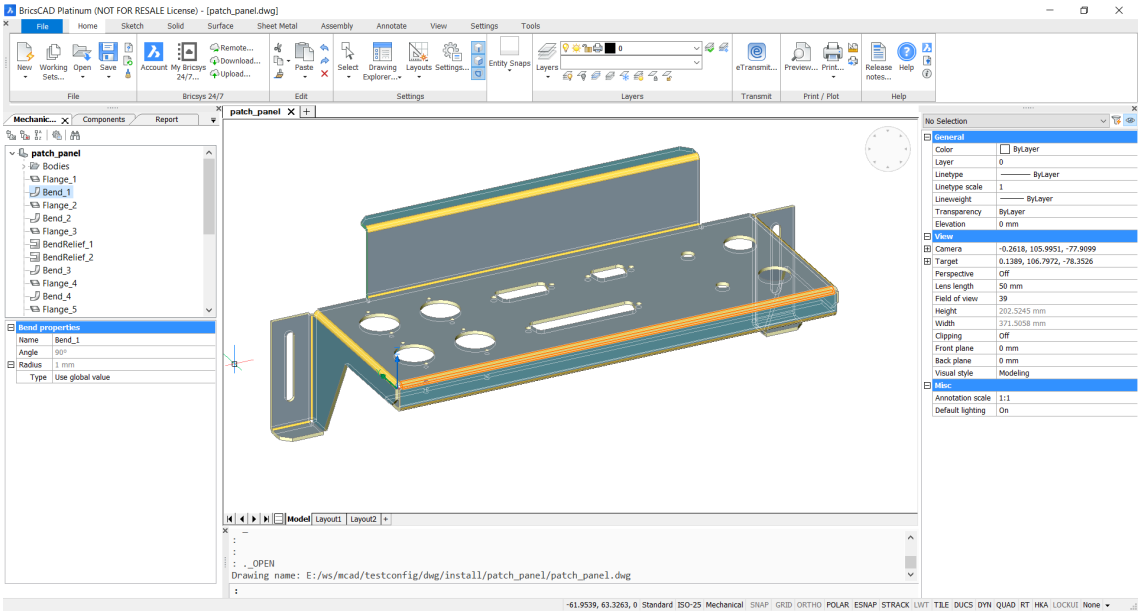


Figura 119: Navegador de Mecânica

Nota: O grupo de propriedades **Sheet Metal** mudou para propriedades de **Dobra**. Clicar no recurso no Navegador Mecânico permite:

- Localizar recurso na peça, ao destacar sua geometria.
- Ser propriedades de determinado recurso.

21.5.5.4 Alterar a espessura

Para alterar a espessura de uma peça em chapa metálica:

1 Selecionar o nó raiz no **Navegador de Mecânica**.

2 Digite um valor no campo **Espessura**.

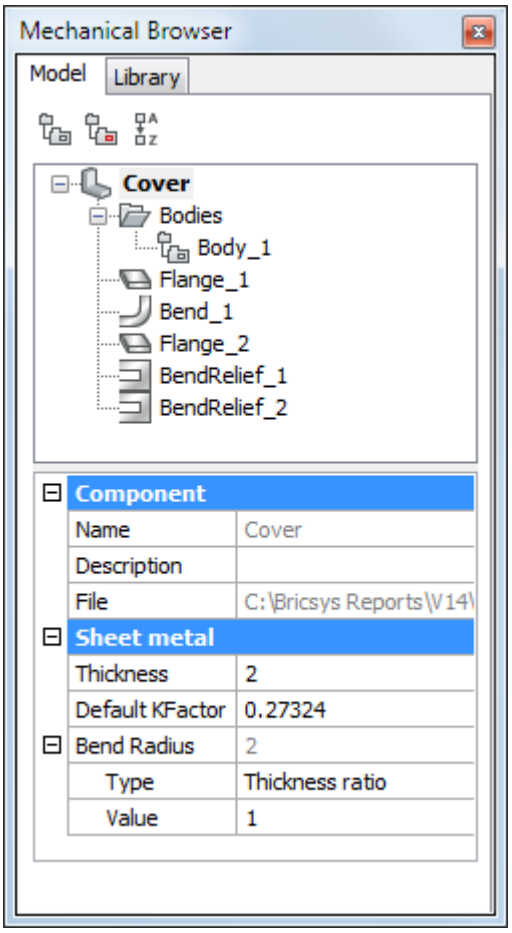
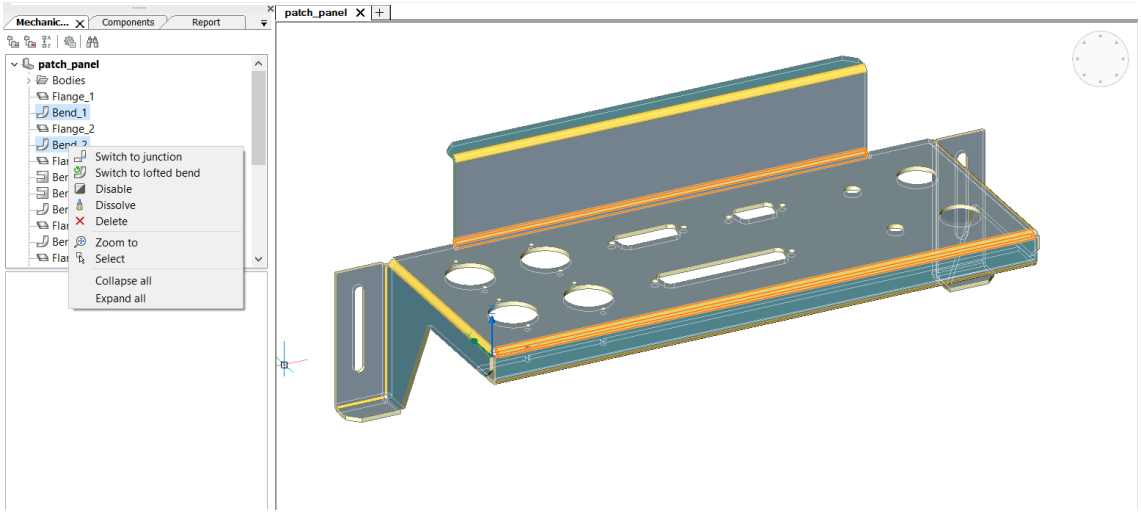


Figura 120: esPEssura

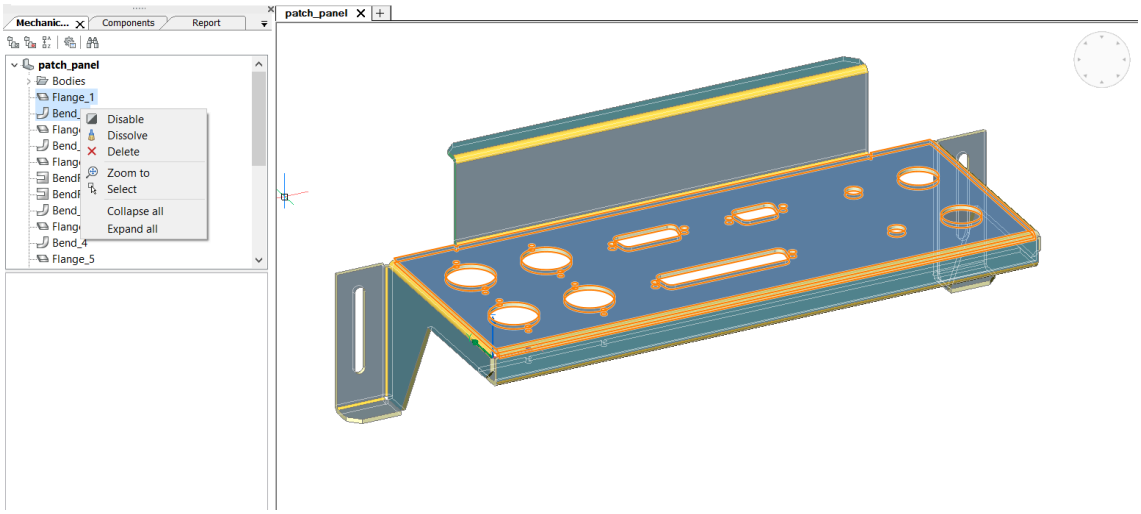
21.5.5.5 Operações de recursos

Navegador de Mecânica fornece acesso rápido às operações sobre determinado recurso de chapa metálica, a lista depende do tipo de recurso. A lista de operações depende dos recursos selecionados e contém operações disponíveis para a seleção.

Selecionando duas dobras resulta em operações para o recurso de dobra:



Selecionando flange e dobra limita para operações disponíveis para ambos recursos:



21.5.5.6 Entender a diferença entre Desabilitar, Dissolver e Excluir

Quando você clicar o botão-direito em um recurso no **Navegador de Mecânica**, e então selecionar **Dissolver** no menu de contexto, o recurso selecionado será removido da peça, mas esta manterá a geometria. No entanto, é removida a intenção de projeto (relações espaciais e paramétricas entre as faces dos recursos) associadas à geometria de um recurso dissolvido.

Se sua intenção é deixar de lado temporariamente um determinado recurso, você poderá escolher a opção **Desabilitar**. O recurso vai permanecer no **Navegador de Mecânica**, mas a peça vai se comportar como se o recurso estivesse dissolvido. É benéfico em contraste com a dissolução, que esse recurso ainda atualiza as mudanças de geometria e você pode ativá-lo quando quiser, evitando recriá-lo ou reconhecê-lo pelo comando *SMCONVERT*.

Alguns dos recursos de Sheet Metal suportam a operação **Excluir** no **Navegador de Mecânica**, que é o análogo do comando *SMDELETE*. Nesse caso, o recurso é removido do navegador e sua geometria é alterada, dependendo do tipo do recurso.

21.5.6 Propriedades Globais e dos Recursos de Sheet Metal

21.5.6.1 Visão Geral

Recursos de chapa metálica e menus de contexto estão localizados no **Navegador de Mecânica**. Ver o Navegador de Mecânica para Sheet Metal para entender os princípios gerais. Este artigo descreve as propriedades específicas e menus de contexto para cada tipo de nó no **Navegador de Mecânica**.

Ao trabalhar com peças de chapa metálica, clicar no nó raiz no **Navegador de Mecânica**, exibe as propriedades do desenho inteiro. Às vezes, essas propriedades são chamadas de Contexto Global de Sheet Metal, associado ao desenho: cada peça de Sheet Metal compartilha esse contexto por padrão.

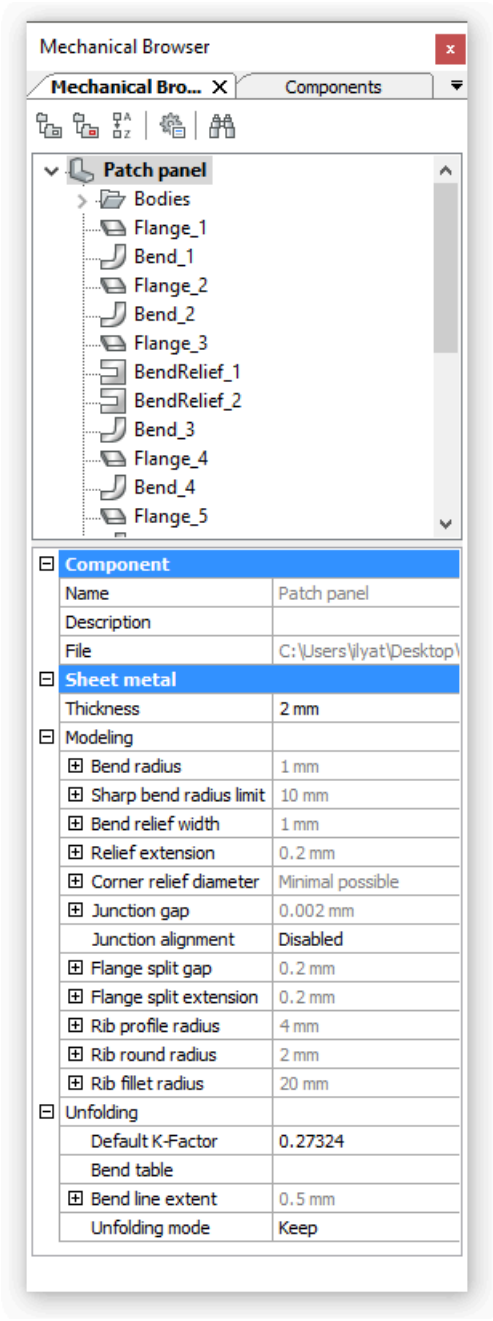


Figura 121: Navegador de Mecânica

A partir do BricsCAD V21, você pode atribuir um contexto local de Sheet Metal para determinado sólido, o que permite definir peças de Sheet Metal com diferentes parâmetros de projeto, e além disso, alterar a espessura da peça. Antes do BricsCAD V21 todas as peças no Model Space deviam ter a mesma espessura, o que forçava o usuário a espalhar as peças com espessura diferente para diferentes blocos ou componentes. Cenários com múltiplos corpos importados envolvendo *SMCONVERT* e *SMASSEMBLYEXPORT* também se beneficiam do novo recurso. Outra vantagem é que o contexto local de Sheet Metal pode definir sua própria Tabela de Dobras.

21.5.6.2 Para mudar o contexto de um sólido, de Global para Local

Faça o seguinte:

1 Selecione o corpo no **Navegador de Mecânica**

2 Na seção Sheet Metal, alterne o **Tipo de Contexto** de **Contexto global** para **Contexto para sólido**.

Nota: O comando *SMCONVERT*, no caso de várias entradas com diferentes espessuras, alterna automaticamente alguns contextos para 'local'.

Propriedades globais de chapa metálica são colocadas na parte inferior do **Navegador de Mecânica**.

Tabela 40: Propriedades Globais

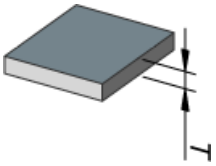
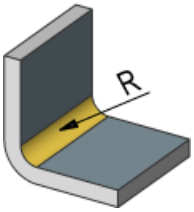
Propriedades Globais	Descrição
esPEssura	Espessura global do material em chapa metálica. 
Raio da dobra	Raio padrão de recursos de dobra; este também pode ser alterado para cada dobra. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Limite do raio de dobra afiada	Raio de dobra máximo, para ser considerado uma curva precisa (regular). Usado nos comandos *SMCONVERT* e *SMRELIEFCREATE* para distinguir entre dobras regulares e dobras em loft. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor.

Tabela 40: Propriedades Globais

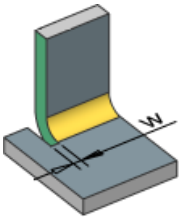
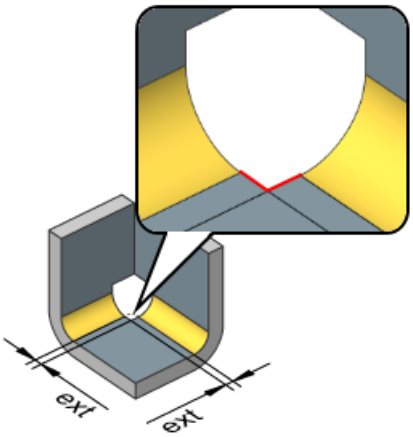
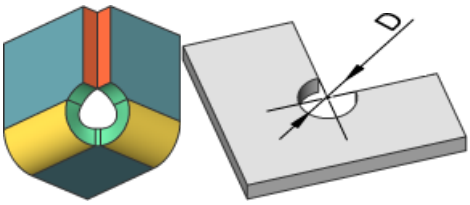
Propriedades Globais	Descrição
Largura alívio da dobra	Largura do corte, próximo de uma dobra única. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Extensão do alívio	Tamanho da extensão de alívio de dobra ou de canto. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Diâmetro de alívio de canto	Diâmetro padrão de um alívio de canto circular. **Tipo**: mínimo possível, ou valor absoluto. 

Tabela 40: Propriedades Globais

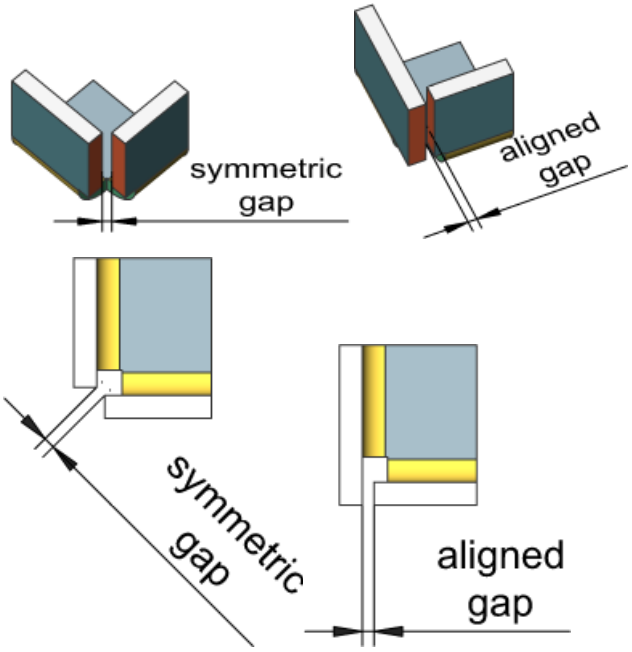
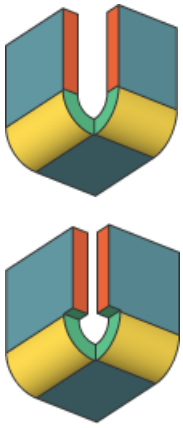
Propriedades Globais	Descrição
Lacuna da junção	Lacuna entre duas flanges adjacentes. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Alinhamento da junção	Controla o alinhamento das faces de junção com as faces do alívio de canto correspondente. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. • **Alinhamento de Junção**: habilitado ou desabilitado  **Figura 123:** Desabilitado **Figura 122:** Ativado

Tabela 40: Propriedades Globais

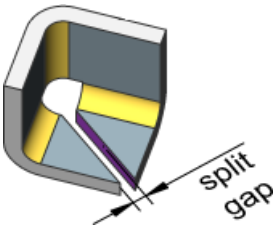
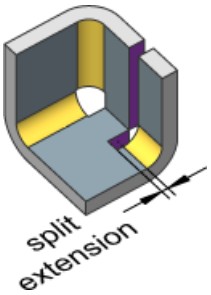
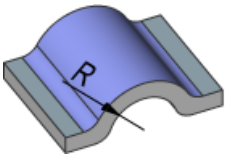
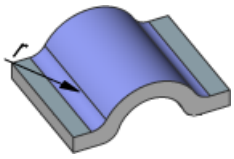
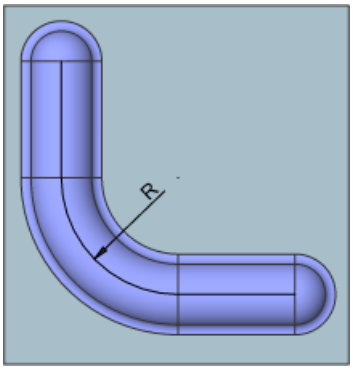
Propriedades Globais	Descrição
Lacuna ao dividir a flange	Espaço entre as faces de espessura de uma flange ou a dobra adjacente ao dividir, consulte *SMSPLIT*. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Extensão da divisão de flange	Controla o corte de profundidade de uma flange adjacente, no caso de dividir uma flange. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Raio do perfil da nervura	Raio de uma ferramenta, a qual cria uma nervura. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 
Raio redondo da nervura	Raio de concordância entre a flange e uma nervura. • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor. 

Tabela 40: Propriedades Globais

Propriedades Globais	Descrição
Raio de concordância da nervura	Raio do concordância da dobra de controle da nervura. Consulte Como Trabalhar com Recursos de Forma e de Nervura. • **Tipo**: tanto a razão do raio do perfil quanto um valor absoluto. • **Valor**: valor numérico. 
Desdobrar	Parâmetros para a representação desdobrada.
Fator-K padrão	Fator-K para o raio de dobra igual à espessura. Consulte o Fator-K.
Tabela de dobra	Exibe a caixa de diálogo Abrir para tabelas de dobra, no formato de arquivo CSV.
Extensão da linha de dobra	Controla até onde uma linha de dobra se estende além do limite de desdobramento (são permitidos valores de zero e negativo). • **Tipo**: razão da espessura, ou valor absoluto. • **Valor**: tanto razão como valor.
Modo de desdobramento	Controla a aparência dos recursos de forma na representação desdobrada. • **Manter**: mantém a representação 3D do recurso de forma. • **Remover**: remove completamente o recurso de forma da representação desdobrada. • **Projetar**: projeta todas as bordas do recurso de forma como entidades 2D. • **Contorno**: projetar apenas o contorno dos recursos de forma como entidades 2D. • **Símbolo**: insere símbolos 2D nos pontos de inserção, em representação desdobrada.

As propriedades do recurso são exibidas automaticamente ao selecionar um recurso no **Navegador de Mecânica**.

Clique o botão-direito em um recurso para exibir o menu de contexto desse recurso.

Tabela 41: Menus de Contexto dos Recursos

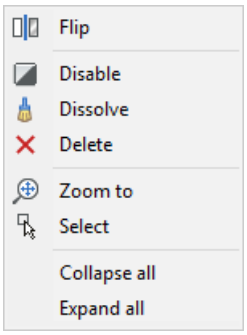
Menus de Contexto dos Recursos	Descrição
	Ferramentas comuns: • **Desabilitar**: desativa temporariamente um recurso, e ignora sua restrição. • **Dissolver**: remove um recurso, mas mantém sua geometria. A intenção do projeto associada com a geometria de um recurso dissolvido será removida. • **Excluir**: Consulte o comando *SMDELETE*. Remove recursos do Navegador de Mecânica e exclui a geometria correspondente no modelo. • **Zoom in**: faz um zoom no recurso selecionado. • **Recolher tudo**: recolhe o componente principal e todos os componentes e subcomponentes. • **Expandir tudo**: expande o componente principal e todos os componentes e subcomponentes.
Recurso de flange	Clique o botão-direito em um recurso de flange:   **Inverter**: troca os lados de uma flange selecionada, para que a face de referência fique no lado geométrico oposto da flange. Esta operação desloca a flange pela distância de sua espessura, a fim de preservar a posição da face de referência.

Tabela 41: Menus de Contexto dos Recursos

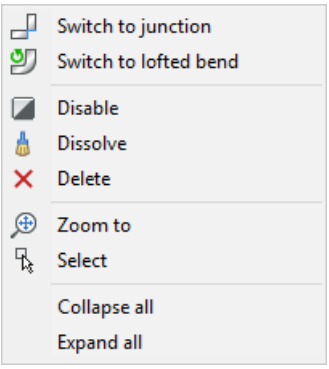
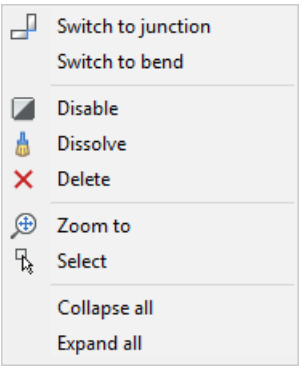
Menus de Contexto dos Recursos	Descrição
Recurso de dobra	Clique o botão-direito em um recurso de dobra:  • **Trocar para junção**: troca para um tipo de recurso de junção, com uma folga entre flanges vizinhas. • **Trocar para dobra em loft**: troca para um recurso de dobra em loft.
Recurso de Dobra em Loft	Clique o botão-direito em um recurso de dobra em loft:  • **Trocar para junção**: troca para um tipo de recurso de junção, com uma folga entre flanges vizinhas. • **Trocar para dobra**: troca para um recurso de dobra.

Tabela 41: Menus de Contexto dos Recursos

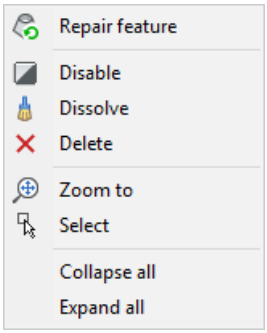
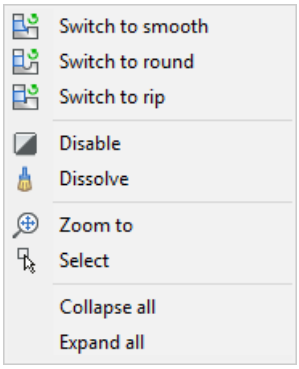
Menus de Contexto dos Recursos	Descrição
Recurso de Dobra Errada	Clique o botão-direito em um recurso de dobra errado:   **Reparar recurso**: repara um recurso de dobra incorreta, criando o recurso de dobra com a geometria apropriada, se isso for possível.
Recurso do Alívio de Dobra	Clique o botão-direito no recurso de alívio de dobra:   **Trocar para suave/redondo/rasgo**: troca um alívio de dobra para o tipo de recurso selecionado.

Tabela 41: Menus de Contexto dos Recursos

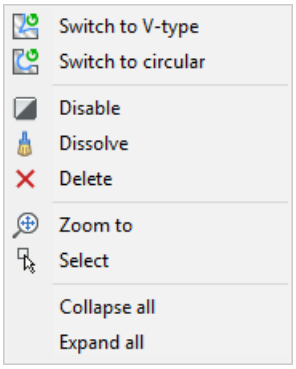
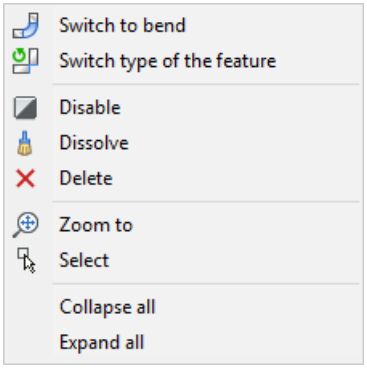
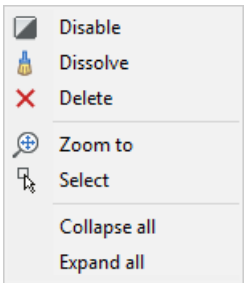
Menus de Contexto dos Recursos	Descrição
Recurso de alívio de canto	Clique o botão-direito no recurso de alívio de canto:   **Trocar para Tipo-V/circular**: troca um recurso de alívio de canto para o tipo de recurso selecionado.
Recurso de junção	Clique botão-direito no recurso de junção:   • **Trocar para dobra**: troca para um recurso de dobra. • **Trocar o tipo do recurso**: troca o modo de alinhamento entre simétrico e alinhado com uma das flanges.
Recurso de miter	Clique o botão-direito no recurso de miter:  

Tabela 41: Menus de Contexto dos Recursos

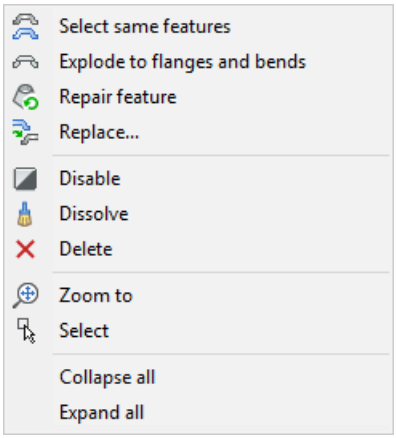
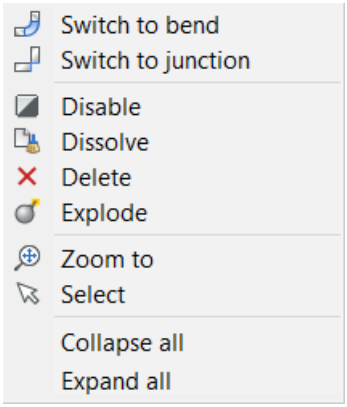
Menus de Contexto dos Recursos	Descrição
Recurso de forma	Clique o botão-direito no recurso de forma:  • **Selecionar o mesmo recurso**: seleciona todos os recursos de forma, do mesmo tipo que tem no modelo. • **Explodir para flanges e dobras**: explode um recurso de forma em um conjunto de flanges e dobras, se for possível. • **Reparar recurso**: recalcula o ponto de inserção do recurso de forma. • **Substituir...**: exibe a caixa de diálogo Abrir para que você escolha um recurso de forma para a substituição.
Recurso de Aba	Clique o botão-direito no recurso de aba:  • **Trocar para dobra**: converte um recurso de aba em um recurso de dobra • **Trocar para junção**: converte um recurso de aba para um recurso de junção

Tabela 41: Menus de Contexto dos Recursos

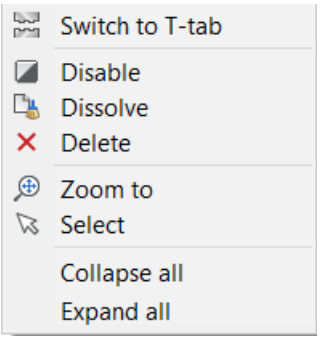
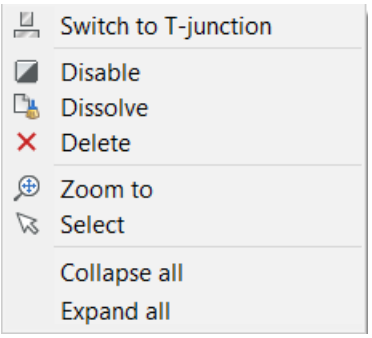
Menus de Contexto dos Recursos	Descrição
Recurso de Junção-T	Clique o botão-direito em um recurso de Junção-T:  **Trocar para Aba-T**: converte um recurso de Junção-T em um recurso Aba-T.
Recurso aba-T	Clique o botão-direito no recurso Aba-T:  **Trocar para Junção-T**: converte um recurso de Aba-T para um recurso de Junção-T

Tabela 42: Propriedades do Recurso

Propriedades do Recurso	Descrição
	Propriedade comum: **Nome**: o nome do recurso selecionado, de Sheet Metal. Opcionalmente, digite um nome para o recurso. O nome padrão de um recurso é o tipo de recurso, mais um incremento, por exemplo, Flange_1, Flange_2,...

Tabela 42: Propriedades do Recurso

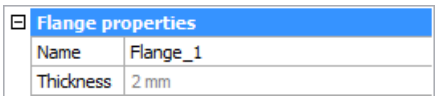
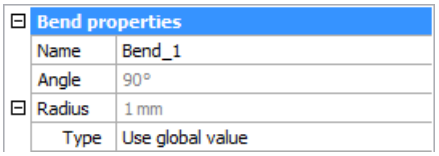
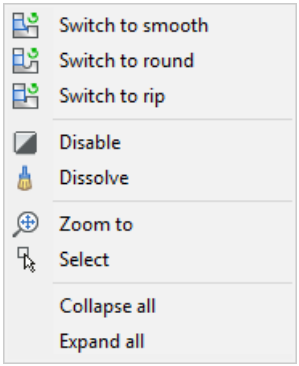
Propriedades do Recurso	Descrição
Recurso de flange	Clique em um recurso de flange:  **Espessura**: espessura atual. O valor é somente-leitura; é igual à propriedade de espessura do componente de Sheet Metal.
Recurso de dobra	Clique em um recurso de dobra:  • **Angulo**: ângulo atual do recurso de dobra; pode ser alterado pelo comando *SMFLANGEROTATE*. • **Raio**: raio atual de dobra. • **Tipo de raio**: - Razão da espessura: razão da espessura multiplicada por um coeficiente - Valor absoluto: o valor global (propriedades do componente principal de chapa metálica).
Alívio de dobra	Clique num recurso de alívio de dobra:  • **Tipo de alívio**: tipo de alívio atual; seja retangular, suave, redondo, ou em rasgo • **Largura**: largura do corte tecnológico próximo da dobra; pode ser global, razão da espessura ou valor absoluto.

Tabela 42: Propriedades do Recurso

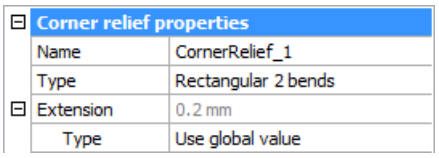
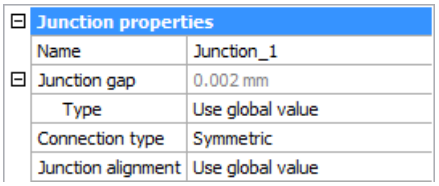
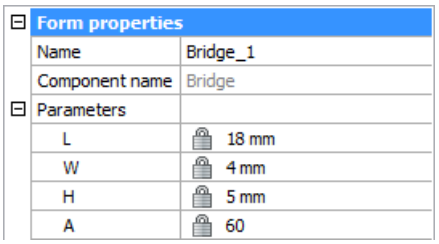
Propriedades do Recurso	Descrição
Alívio de canto	Clique em um recurso de alívio de canto:  **Corner relief properties** Name: CornerRelief_1 Type: Rectangular 2 bends Extension: 0.2 mm Type: Use global value • **Tipo de alívio**: tipo atual de alívio; pode ser retangular com 2 dobras, retangular, tipo-v, ou circular • **Largura**: largura do corte próximo do canto; pode ser global, razão da espessura ou valor absoluto.
Recurso de junção	Clique num recurso de junção:  **Junction properties** Name: Junction_1 Junction gap: 0.002 mm Type: Use global value Connection type: Symmetric Junction alignment: Use global value • **Lacuna da Junção**: folga entre duas flanges vizinhas. • **Tipo de lacuna da Junção**: pode ser global, razão da espessura, ou valor absoluto. • **Tipo de conexão**: controla a posição relativa das flanges adjacentes; tanto as simétricas como adjacentes a uma das flanges. • **Alinhamento da junção**: controla a posição das faces de um recurso de junção; - global - habilitado (alinhado com as faces do recurso de alívio de canto correspondente) - desabilitado.
Recurso de miter	Clique em um recurso de miter:  **Miter properties** Name: Miter_1 Miter gap: 0.2 mm Type: Use global value • **Lacuna da mitra**: distância entre as faces laterais da mitra. • **Tipo**: razão da espessura ou valor absoluto.

Tabela 42: Propriedades do Recurso

Propriedades do Recurso	Descrição
Recurso de forma	Clique em um recurso de forma:  • **Nome do componente**: nome da definição do componente do recurso de forma. • **Parâmetros**: contém os parâmetros de um recurso de forma paramétrico de chapa metálica.
Recurso de bisel	Clique em um recurso de bisel:  • Tipo de bisel: define um tipo de bisel, dependendo da geometria. Valores possíveis: um-lado, pouso, bisel-X e bisel-K. Sobre um tipo de bisel, os parâmetros podem alterar. • Bisel lateral: parâmetros de chanfro em um lado deste bisel. • Bisel ortogonal: parâmetros do lado ortogonal.

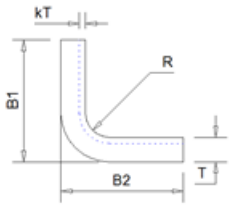
21.5.7 Fator-K

21.5.7.1 Visão Geral

Quando uma chapa de metal é dobrada com uma prensa dobradeira, seu material é deformado, o que é o motivo da diferença no comprimento total de uma folha entre os estados: dobrado e desdobrado. O

BricsCAD considera automaticamente essa deformação quando você desdobra uma peça de Sheet Metal. Duas abordagens são usadas para calcular isto: uma simplificada (Fator-K) e uma mais avançada.

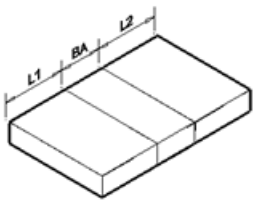
As propriedades de deformação do material ao ser dobrado são baseadas na suposição de que há uma superfície interna para a peça de Sheet Metal que não é deformada quando a folha é dobrada. A partir da variedade de superfícies internas, é selecionada uma superfície que é equidistante a partir de um lado da dobra. A superfície é local para cada dobra e, para casos simples pode ser propagada na peça inteira, como esta na imagem. Assumimos que essa superfície não é esticada durante a dobra.



T: (thickness) espessura da peça de chapa metálica

R: raio interno da dobra

O Fator-K (k) é a relação entre a localização da superfície neutra e a espessura do material. Como resultado, a superfície neutra encontra-se a uma distância k*T da superfície interna da dobra. O raio de dobra da superfície neutra é igual a R + kT. O Fator-K é um cálculo geométrico simples de localização da superfície neutra. Tensões de formatação e outros fatores desconhecidos (erro) não são considerados. O Fator-K depende de muitos fatores, como o tipo de material, tipo de dobra e ferramentas. O Fator-K normalmente fica entre 0.3 e 0.5. O Fator-K padrão para um raio de dobra igual à espessura do material (T) é 0.27324.



No estado desdobrado de uma peça de sheet metal, as flanges (L1 e L2) não são esticadas. A parte da curva é indicada como BA: (bend allowance) permissão de dobra. O BA é igual ao comprimento da superfície neutra desdobrada:

BA = BendAngle [AnguloDeDobra] * (R + kT)

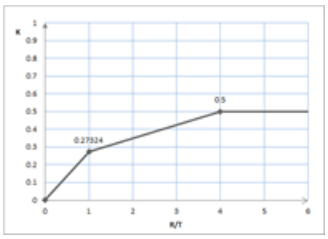
Na verdade, é difícil medir o Fator-K ou o subsídio de dobra. A seguinte fórmula permite calcular a Dedução de Dedução de dobra (BD) (bend deduction):

BD = B1 + B2 - Comprimento Desdobrado = B1 + B2 - (L1 + L2 + BA)

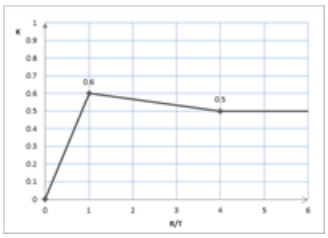
Para casos simples, o valor do Fator-K pode ser substituído no navegador. Para uma precisão máxima, uma tabela de dobras contendo deduções tangenciais de dobra deve ser fornecida. Para cada peça de chapa metálica, você pode especificar um Fator-K escolhido, ou usar o valor padrão.

21.5.7.2 Para alterar o fator-K para sua peça de Sheet Metal:

Para alterar o Fator-K de uma peça de chapa metálica, selecione o nó raiz no Navegador de Mecânica e digite um valor no campo **Fator-K**. O valor deve estar no intervalo [0, 1], uma vez que a superfície neutra está localizada dentro da peça de chapa metálica. No BricsCAD, o Fator-K é definido para raio de dobra igual à espessura do material (R/T = 1). Para calcular o valor do **Fator-K** para um raio de curvatura arbitrário, o BricsCAD usa uma técnica de interpolação especial que provou ser válida para aplicações industriais de chapas metálicas. A imagem abaixo mostra a estratégia de interpolação: para R/T < 1 a interpolação linear é feita entre 0 e 0.27324, para 1 < R/T < 4 o Fator-K é interpolado linearmente entre 0.27324 e 0.5 e se R/T for maior que 4, o Fator-K é constante e igual a 0.5. Fisicamente significa que se o raio de dobra for muito maior do que a espessura (pelo menos 4 vezes), o estiramento do material pode ser ignorado.



O único parâmetro de Fator-K que se pode controlar no navegador é para R/T. Isso significa que se você definir o valor como 0.6, a lei de interpolação leva ao seguinte resultado:

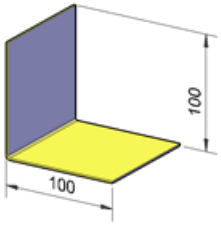


Pic: o fator-K variável no navegador leva à seguinte curva efetiva.

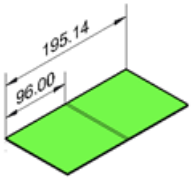
O recurso distintivo da dobra é que variando R na faixa de (0, T], L1 + L2 + BA dará o mesmo valor. Isto é explicado pelo fato de que para raios menores do que a espessura que eles se relacionam são para conveniência de projeto em vez de alguma modelagem do processo físico, uma vez que durante a dobra o raio da ferramenta é geralmente maior ou igual à espessura.

Vamos rever o exemplo com variação do Fator-K pela configuração da curva de controle (o raio da dobra é mantido constante).

T = 2, R = 2, B1 = 100, B2 = 100



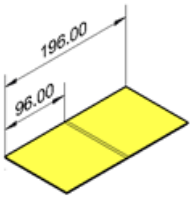
Configurando K=0 dá L1=L2=96, Comprimento Desdobrado = 195.14, assim BA é 3,14.



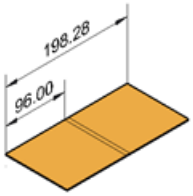
Este valor se encaixa exatamente na fórmula: um ângulo reto em radianos aproximadamente é igual a 1,57 (PI/2). O fator-K padrão 0.27324 dá um Comprimento Desdobrado = 196.

Ou: L1=L2=96 (as flanges não são deformadas), o BA = 4 como podemos calcular a partir das dimensões (196 - 2*96) ou da fórmula de Permissão de Dobra:

BA = BendAngle * (R + kT) = 1,57 * (2 + 2*0,27324) = 4



Finalmente, maximizar o fator-K resulta em um Comprimento Desdobrado = 198,28. Este é o valor máximo que se pode alcançar neste exemplo, uma vez que a superfície neutra é tomada do lado externo da curva.



A Dedução da Dobra/Bend Deduction BD para k = 0.27324:

BD = B1 + B2 - Comprimento Desdobrado = 100 + 100 - 196 = 4

21.5.8 Tabela de Dobras

21.5.8.1 Visão Geral

Tabelas de dobras são uma maneira mais confiável de expressar propriedades de deformação do material do que o Fator-K. Ao dobrar fisicamente uma folha plana de um determinado material, você pode medir

seu comprimento antes e depois desse processo. Você pode repetir esse procedimento para diferentes ângulos dobrados, raios de dobra e espessura da folha, e salvar as medições em uma tabela de dobras. Em seguida, essas medições podem ser consideradas para calcular o comprimento desdobrado correto para todas as peças feitas do mesmo material.

Em uma tabela de dobras, você pode adicionar as medidas para quantos ângulos diferentes de dobradura quiser. No entanto, na maioria dos casos, basta medir o comprimento para um ângulo de 90 graus. O BricsCAD vai calcular automaticamente o comprimento para outros ângulos de curvatura usando uma confiável técnica de interpolação.

Tabelas de dobra são armazenadas em um arquivo em formato *.csv (valores separados por vírgula). Você pode criar uma tabela de dobra em um editor de texto ASCII, como o Bloco de Notas. No entanto, uma maneira mais confiável e fácil é criar uma tabela em uma planilha, como Microsoft Excel ou LibreOffice, e exportá-la para o formato de arquivo *.csv.

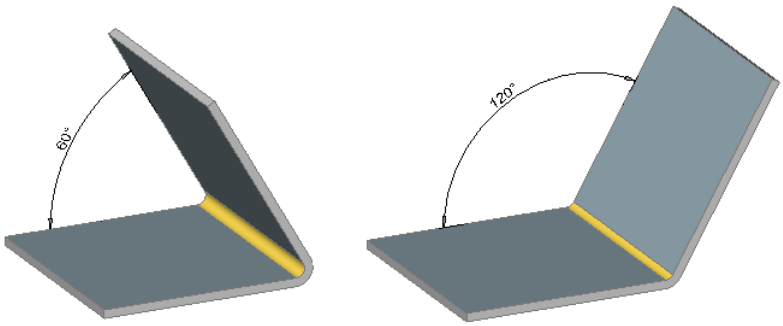
Tabelas de dobras no BricsCAD tem a seguinte estrutura:

	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

Figura 124: Exemplo de uma tabela de dobra

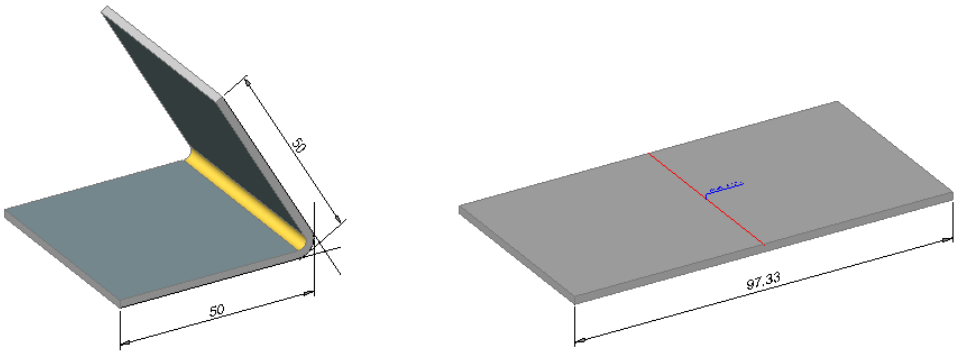
As seguintes regras se aplicam:

- O conteúdo da primeira célula lê "BricsCAD".
- **Versão**: a versão atualmente suportada é 1.
- **AngleType**: ctualmente, somente os ângulos de dobra internos são suportados:

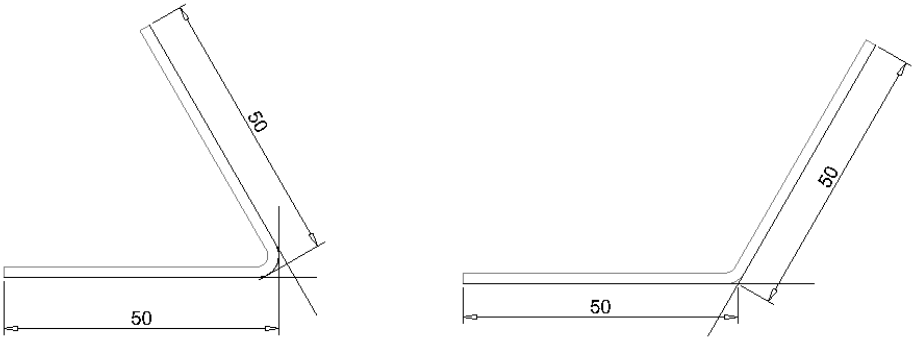


- **LengthType**: a semântica do valor nas células da tabela de dobras correspondente a um ângulo de dobra específico e um raio de dobra. Agora, apenas a dedução de dobra medida a partir do ponto tangente é suportada (“BendDeductionTangent”). Dedução de dobra - bend

deduction - (BD) é a diferença entre a soma dos comprimentos de duas flanges medidas até o ponto tangente em um modelo 3D, e o comprimento do mesmo fragmento no estado desdobrado:



Em profundidade: métodos para calcular a dedução tangencial da curva para ângulos agudos e obtusos.



- Dentro desse cabeçalho definido, diversas tabelas podem existir. Para cada tabela, a primeira chave é a **Thickness** (Espessura) da peça de chapa metálica. A segunda chave, **DieWidth** (Matriz de dobra), é opcional. Os pares **Thickness**/**DieWidth** devem ser únicos. Se **DieWidth** não estiver definido, apenas uma chave única de valor de **Thickness** é permitida, mas múltiplos valores de espessura podem ser adicionados. Quando uma tabela de dobras é usada para controlar o processo de desdobramento, a espessura **Thickness** e a matriz **DieWidth**, definidas na peça de Sheet Metal, devem existir na tabela de dobras. Se a tabela não for encontrada, a política padrão do Fator-K será usada.
- Uma vez que a tabela de dobra é encontrada no *.csv, ela é usada para desdobrar a peça de chapa metálica. Usando a tabela de dobra de amostra acima, vamos verificar alguns casos:

  T = Thickness (Espessura), R = Raio Interno de Dobra, A = Angulo de Dobra
  - T = 1, R = 2, A = 45. Então, após o desdobramento, obterá o BD = 0,466, porque R e A são encontrados na tabela de dobras para a espessura dada.
  - T = 1, R = 2.5, A = 45. A = 45 é encontrado, mas R = 2,5 está faltando: a interpolação da BD entre R=2 e R=3 é usada.
  - T = 1, R = 2,5, A = 75. Nem R nem A são encontrados na tabela: as seguintes células são usadas para calcular o valor de BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) and (R=3, A=90).

  O algoritmo é usado para calcular a interpolação. Por exemplo, interpolação linear em valores de BD adjacentes apresenta resultados não-naturais, que não corresponderão aos resultados reais de

dobradura. Tenha também em mente que para um determinado T, R e A nem todos os valores de BD fazem sentido: um caso eventual pode colocar a superfície neutra fora da peça de chapa metálica, ou expressa em Fator-K (que pode ser deduzido do BD), o Fator-K pode ficar fora do intervalo [0, 1]. Nesse caso, a BD (dedução de dobra) para o Fator-K padrão será usada.

- Se você produzir suas peças de chapa metálica com ferramentas diferentes, pode descrevê-las em uma tabela de dobragem usando o parâmetro **DieWidth**. Uma tabela de dobra de amostra é fornecida para sua conveniência na **[Pasta de instalação do BricsCAD]\ Samples\ Mechanical\ bend_tables**.

  Nota: Os direitos de gravação para essa pasta podem ser limitados. Nesse caso, você precisa copiar as tabelas de dobra para uma pasta diferente na qual tenha direitos de acesso total.

21.5.8.2 Para definir a tabela de dobras para sua peça de chapa metálica

1 Selecionar o nó raiz no Navegador de Mecânica.

2 Selecione o campo **Tabela de dobras**. 

3 Clique no botão **Procurar** ().

4 Selecione o arquivo *.csv.

5 Clique duas vezes ou pressione o botão **Abrir**.

21.5.8.3 Para atualizar tabela de dobras definida anteriormente

1 Selecionar o nó raiz no Navegador de Mecânica.

2 Selecione o campo **Tabela de dobras**.

3 Clique no botão **Atualizar** ().

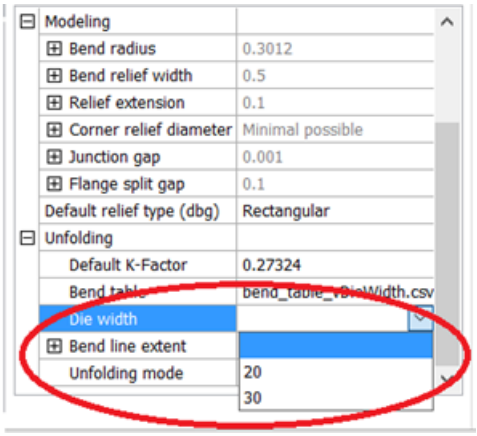
Se a tabela de dobra não está atribuída ao documento, ou não tiver uma chave secundária **DieWidth**, o parâmetro **DieWidth** não pode ser atribuído a uma dobra única ou a todas as dobras. Para fazer a atribuição, siga as seguintes etapas:

	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	

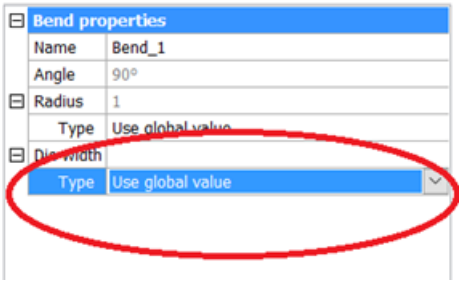
1 Selecione o arquivo *.csv com as chaves secundárias **DieWidth** e a espessura que corresponde à peça atual. No exemplo dado supomos que a peça tenha a espessura de 1 mm, e o raio de dobra também é igual a 1. Vamos variar a dedução de dobra (BD) para diferentes ferramentas: a primeira sub-tabela é quando a ferramenta não está especificada, a segunda sub-tabela é para o caso quando a ferramenta **DieWidth** é igual a 20, e a última é para uma ferramenta com **DieWidth** igual a 30.

2 Atribua a tabela de dobra à peça de sheet metal.

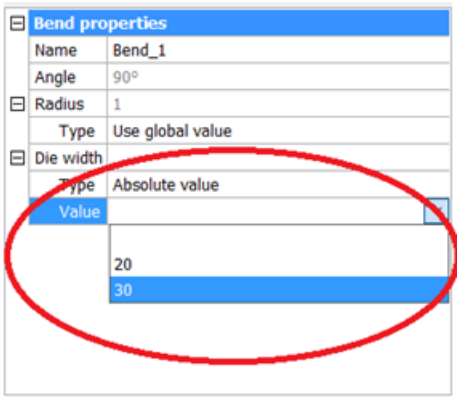
3 Verifique se nos novos campos do Navegador de Mecânica aparecem:



Nesta configuração global, podemos selecionar qualquer valor **DieWidth** padrão adequado para a espessura da peça. O valor é atribuído por padrão às dobras da peça, que podem ser verificadas pela propriedade da dobra (clique na dobra no Navegador de Mecânica):



4 No entanto, podemos selecionar uma ferramenta específica que será usada para a dobra citada, expandindo a lista drop-down:



5 Com a seleção dada, um desdobramento da Bend_1 (Dobra_1) vai usar uma terceira sub-tabela da nossa tabela de dobras para dedução de dobra, e usa o valor 1.8. Para o restante das dobras, a menos que **DieWidth** seja definido para elas, o valor padrão no-die-width (sem-largura-de-matriz) será usado, o qual vai acionar primeiro a tabela de dobras, e usa o valor 2.1 para 90 graus com um raio de dobra igual a 1.

Nota: Depois de atribuir o valor de **DieWidth** às dobras e salvar o documento, o valor é salvo para as dobras. No entanto, da próxima vez que o desenho for aberto, para atribuir o valor de **DieWidth** a outra dobra, você terá que recarregar a tabela de dobras primeiro (pressione o botão de seta verde), consulte Para atualizar a tabela de dobras definida anteriormente. Após a recarga, os valores da tabela de dobras estarão disponíveis.

Nota: Desde o BricsCAD V21, você pode alterar tabelas de dobra para diferentes sólidos no Model Space, definindo um Contexto local de Sheet Metal para cada sólido.

21.5.9 Configurações de Sheet Metal

21.5.9.1 Visão Geral

A seção Sheet Metal na caixa de diálogo *Configurações* controla as diferentes propriedades do Sheet Metal no BricsCAD. As propriedades são divididas em quatro grupos: **Comandos**, **Cores de recurso**, **Parâmetros iniciais**, e **Modelagem**.



Figura 125: Configurações: Sheet Metal

Nota: A maioria das configurações de chapa metálica estão armazenadas no desenho. Para melhores resultados sempre comece com um dos arquivos template de Sheet Metal (*.dwt), ou crie o seu próprio.

21.5.9.2 Comandos

A seção **Comandos** contém configurações que controlam o comportamento dos vários comandos de Sheet Metal.

Nota: Antes do BricsCAD V19, esses controles estavam disponíveis como opções dentro dos comandos individuais. Esses foram movidos para a caixa de diálogo de configurações para simplificar a interface de comando.

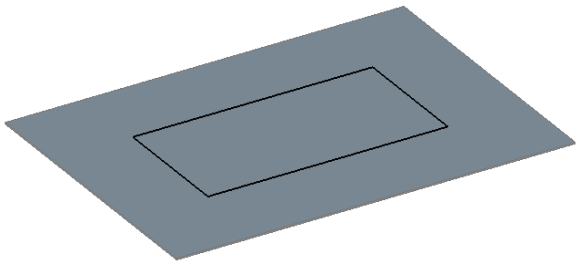


Figura 126: Configurações: Comandos

As Camadas usadas para o Desdobramento de peças em Sheet Metal determinam a aparência das peças desdobradas.

21.5.9.3 Cores dos recursos

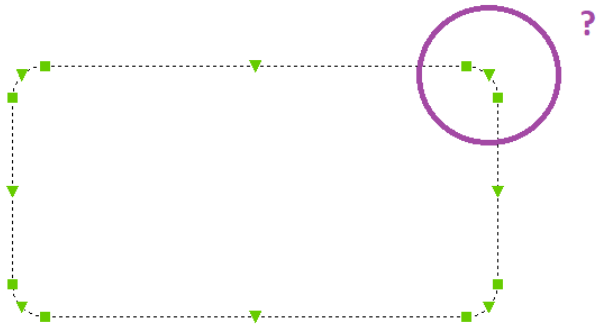


Figura 127: Configurações: Cores dos recursos

As configurações de **Cores dos recursos** distinguem visualmente diferentes tipos de recursos de chapa metálica, por cor. Você pode ajustar as cores para cada tipo de recurso.

21.5.9.4 Parâmetros iniciais

Esses parâmetros são responsáveis pela inicialização do **Contexto de Sheet Metal** associado a um desenho.

Modelagem

As configurações de modelagem controlam o comportamento dos recursos de chapa metálica. As configurações são agrupadas por tipo de recurso.

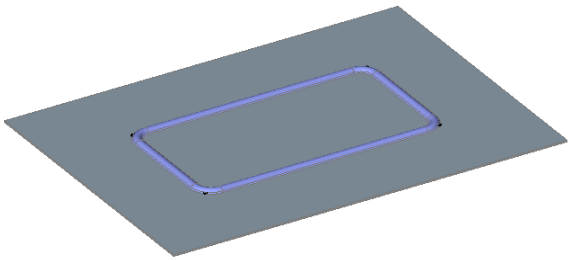


Figura 128: Configurações: Modelagem

Desdobrar

As configurações de desdobramento controlam o comportamento do **valor do Fator-K**, **modo de desdobramento do Recurso de forma**, e **tipo/valor da extensão da linha de Dobra**.

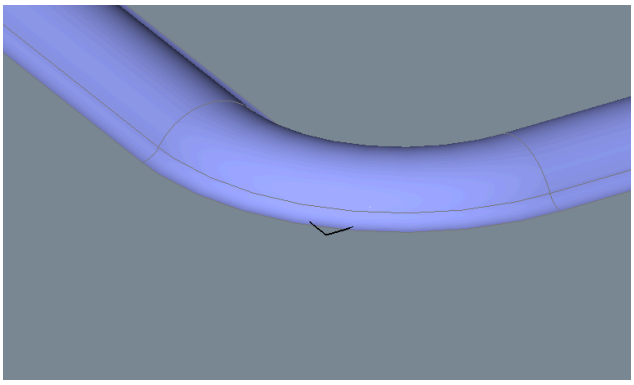


Figura 129: Configurações: Desdobramento

21.5.9.5 Modelagem

A configuração de Modelagem inclui a preferência do usuário *SMSMARTFEATURES* que controla o local e a atualização inteligente da geometria, depois que os comandos de Sheet Metal são executados.

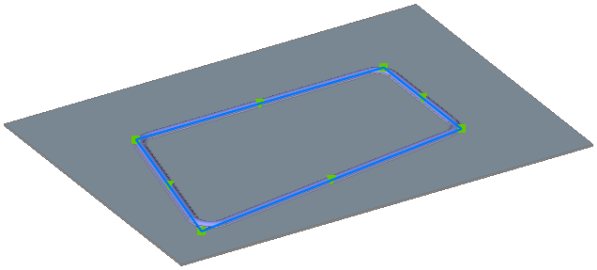


Figura 130: Configurações: Modelagem

Tutorial: Parametrizar peças de Sheet Metal

21.5.10 Criar Flanges

21.5.10.1 Visão Geral

Uma flange é o principal recurso da maioria das peças de chapa metálica. No BricsCAD você pode criar flanges usando várias técnicas diferentes:

- Criar uma flange de base a partir de um perfil 2D fechado, que defina seu limite.
- Criar uma ou várias flanges de borda puxando as bordas das flanges existentes.
- Criar várias flanges ao mesmo tempo extrudando uma polilinha 2D (cada borda linear da polilinha vai se tornar uma flange).

Dica: Existem também outras técnicas que você pode usar para criar flanges. Estes são descritos em:

- Flanges podem ser criadas como elementos de peças em loft de sheet metal.
- Flanges podem ser reconhecidas na geometria existente.

Em contraposição aos softwares MCAD baseados em histórico, no BricsCAD se aplica o mesmo conjunto de ferramentas para modificar qualquer flange - independentemente.

21.5.10.2 Criar uma Flange Base

Em geral, você cria uma flange de base, para iniciar o projeto da peça de sheet metal. Para criar um flange base, selecione um perfil plano fechado e execute o comando *SMFLANGEBASE*.

Siga um dos procedimentos:

- Clique o botão de ferramenta **Criar Flange Base** ( ) na aba **Sheet Metal**.
- Escolha **Criar Flange Base** no menu **Sheet Metal**.
- Escolha **Criar Flange Base** no grupo **Sheet Metal** do menu **Quad**:

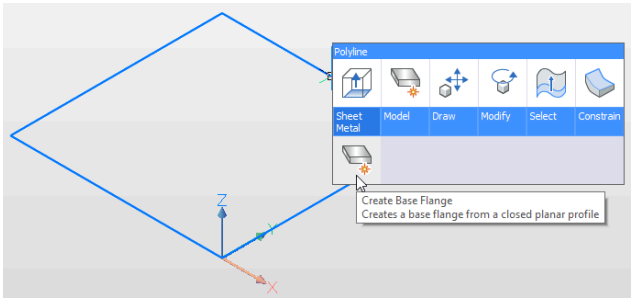
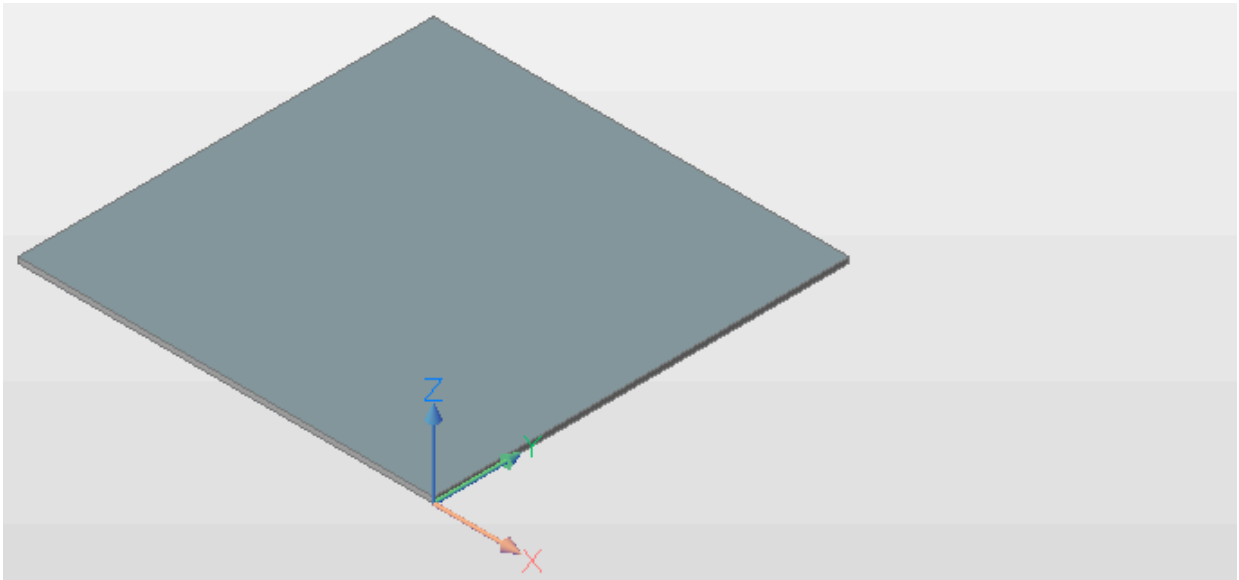


Figura 131: Criar uma Flange Base



Para criar uma flange base, o comando *SMFLANGEBASE* vai extrudar um perfil.

A altura da flange de base é a propriedade de **espessura** padrão da peça de chapa metálica. Para alterar a espessura de sua peça de chapa metálica, digite o valor apropriado no campo **Espessura/Thickness** no Navegador de Mecânica.

21.5.10.3 Criar Flanges de Borda

Você pode adicionar uma flange adicional à sua peça de chapa metálica:

1 Selecione uma borda linear.

2 Selecione o comando *SMFLANGEEDGE* na seção **Sheet Metal** do menu de cursor **Quad**.

3 Mova o ponteiro do mouse para definir o comprimento desejado da flange de borda, e o ângulo entre as duas flanges. Você também pode usar os campos de dimensão dinâmica para inserir os valores necessários.

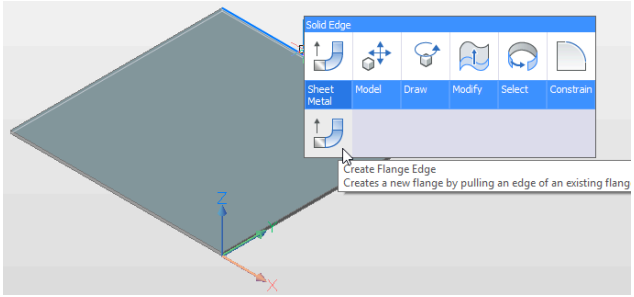
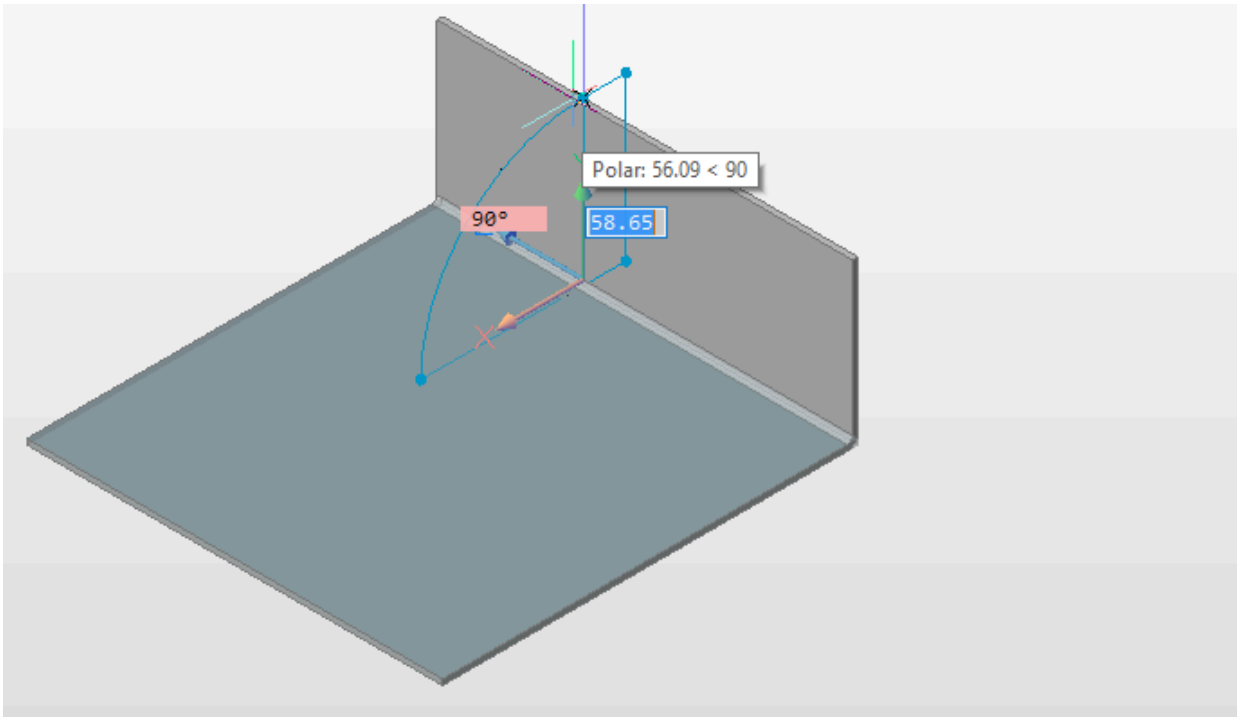
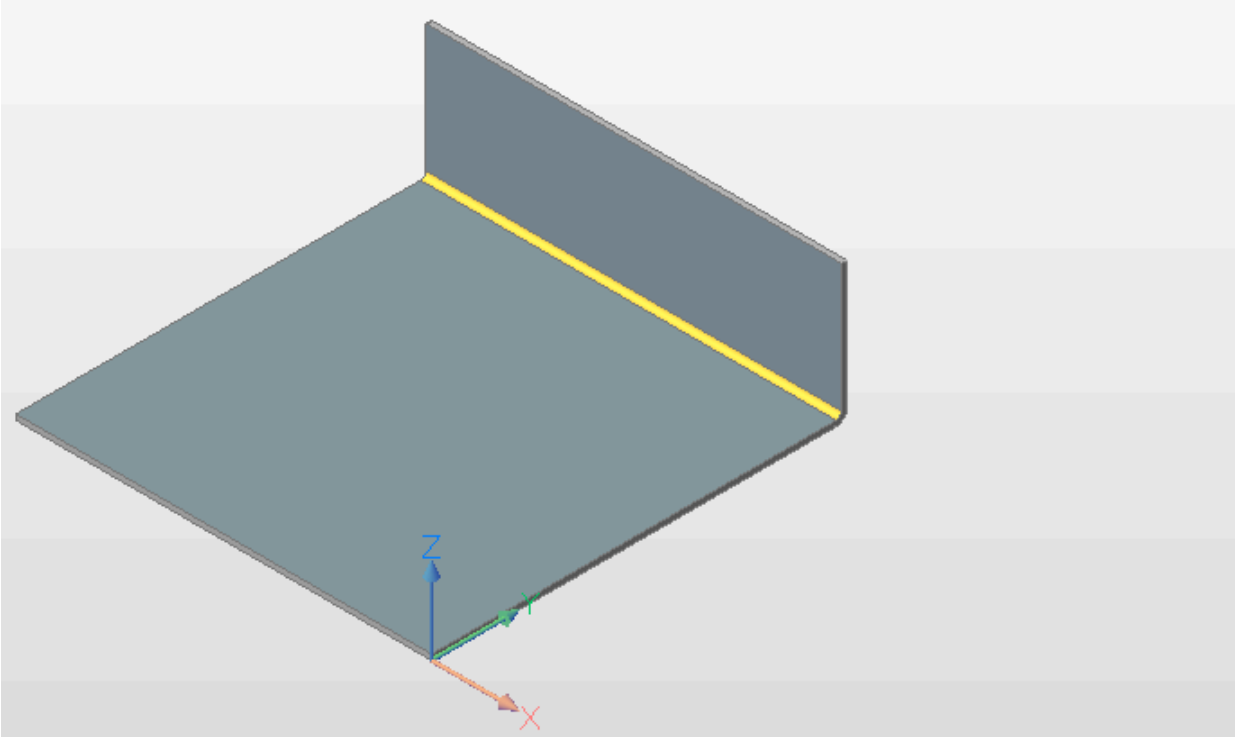
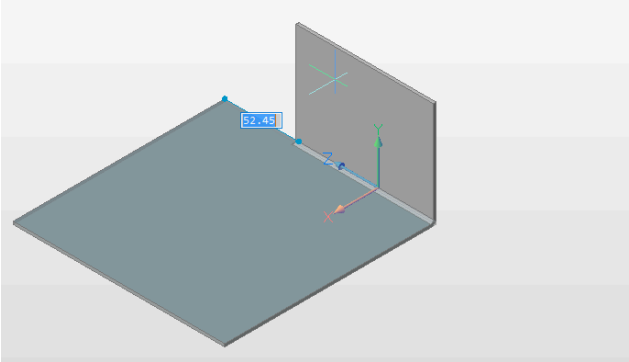


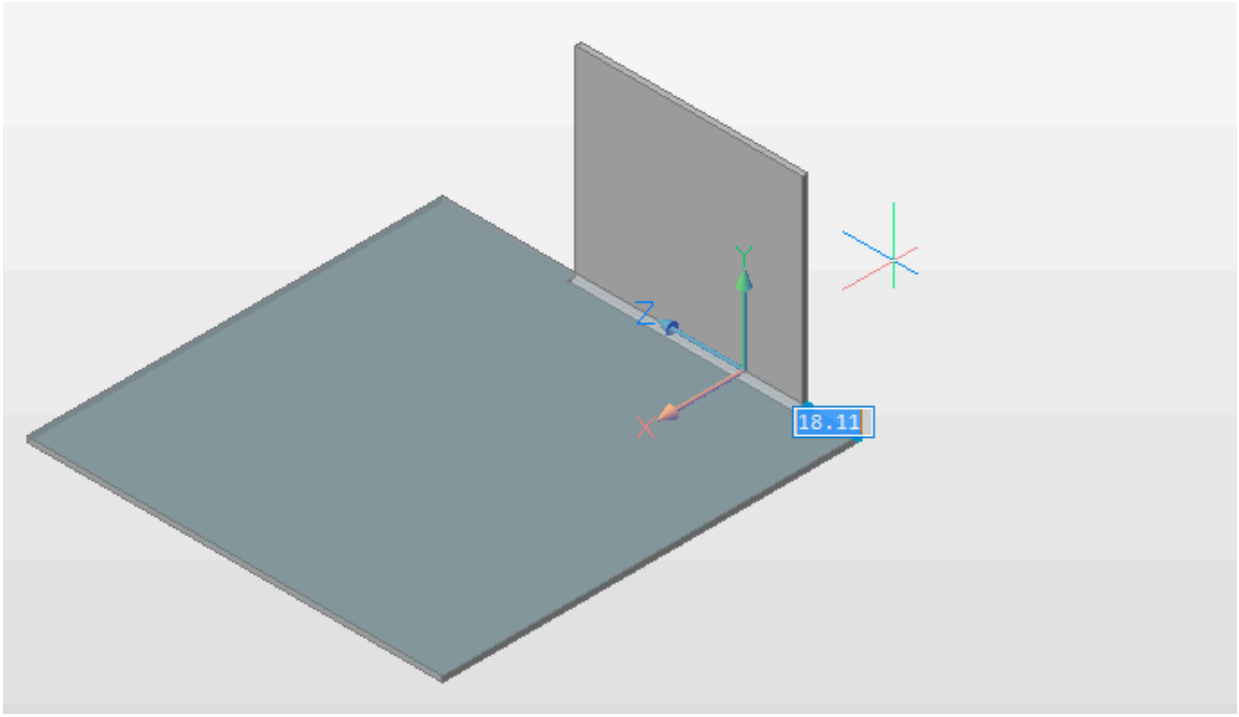
Figura 132: Criar Borda de Flange

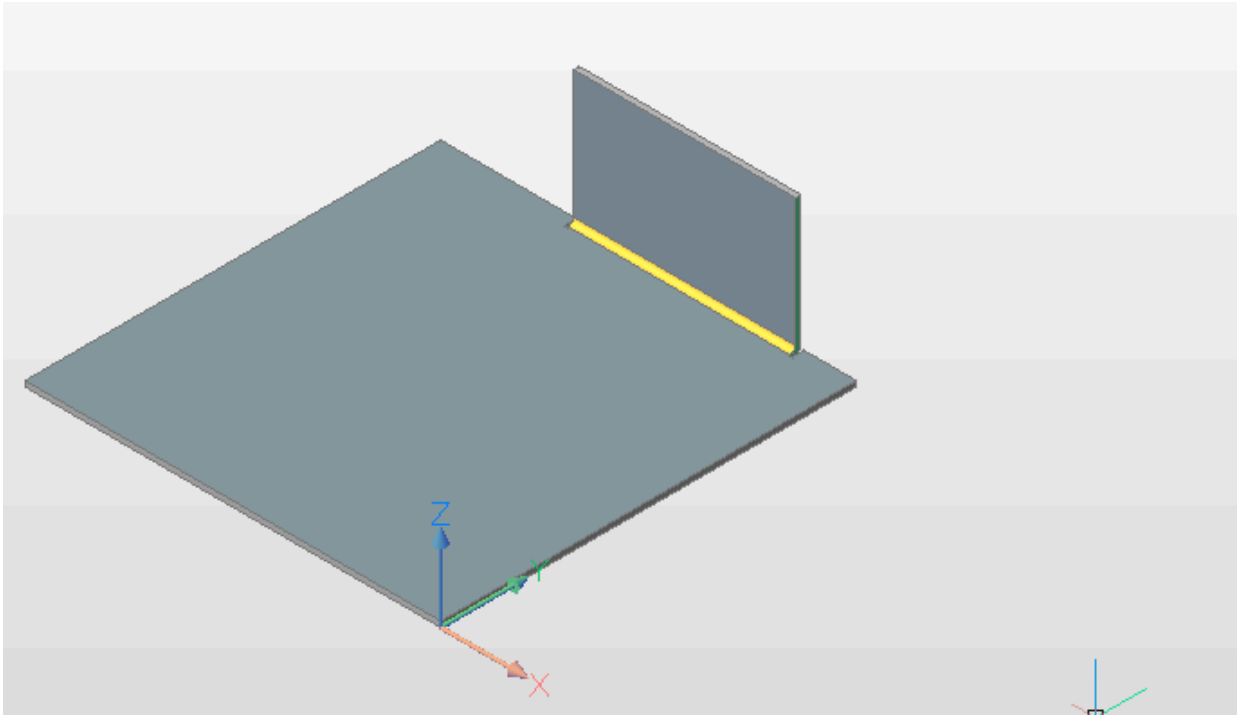




Para criar uma flange de borda cuja largura seja diferente da largura da borda, selecione a opção **Width** (Largura) do comando *SMFLANGEEDGE* e defina a distância do deslocamento de um ou de ambos os lados da borda.



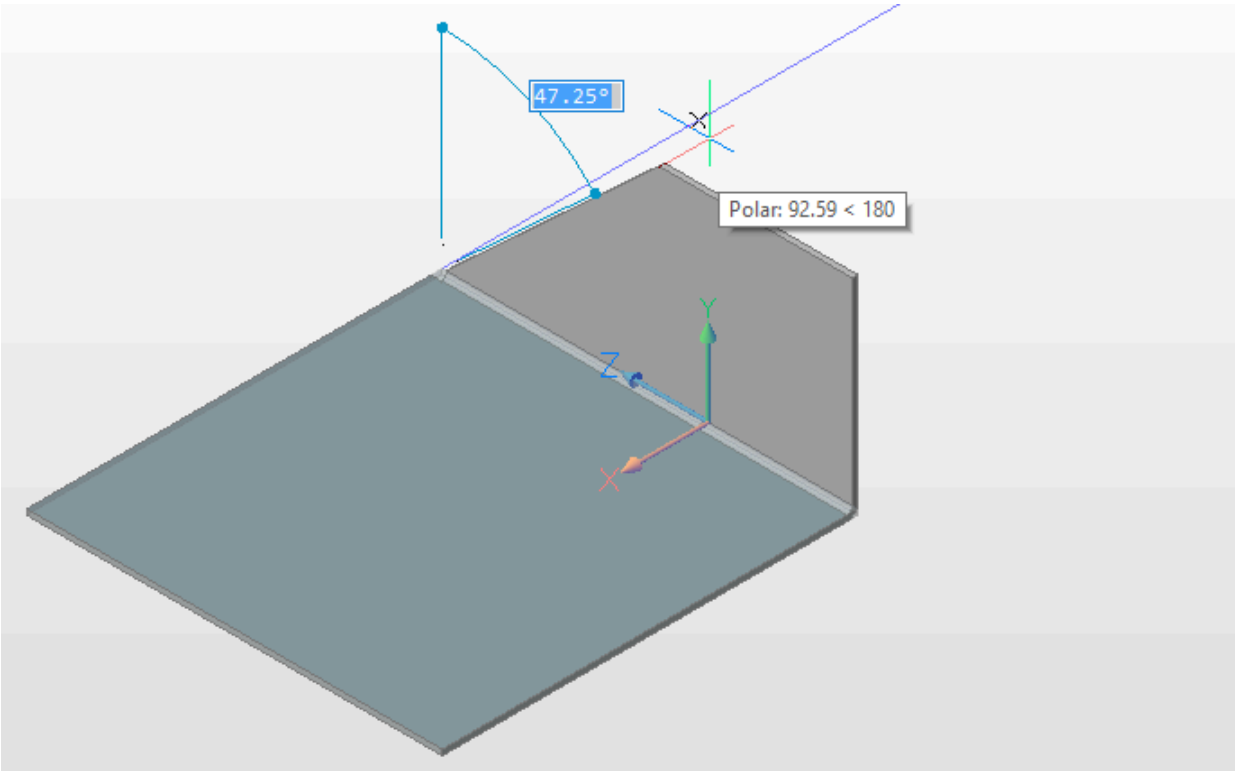


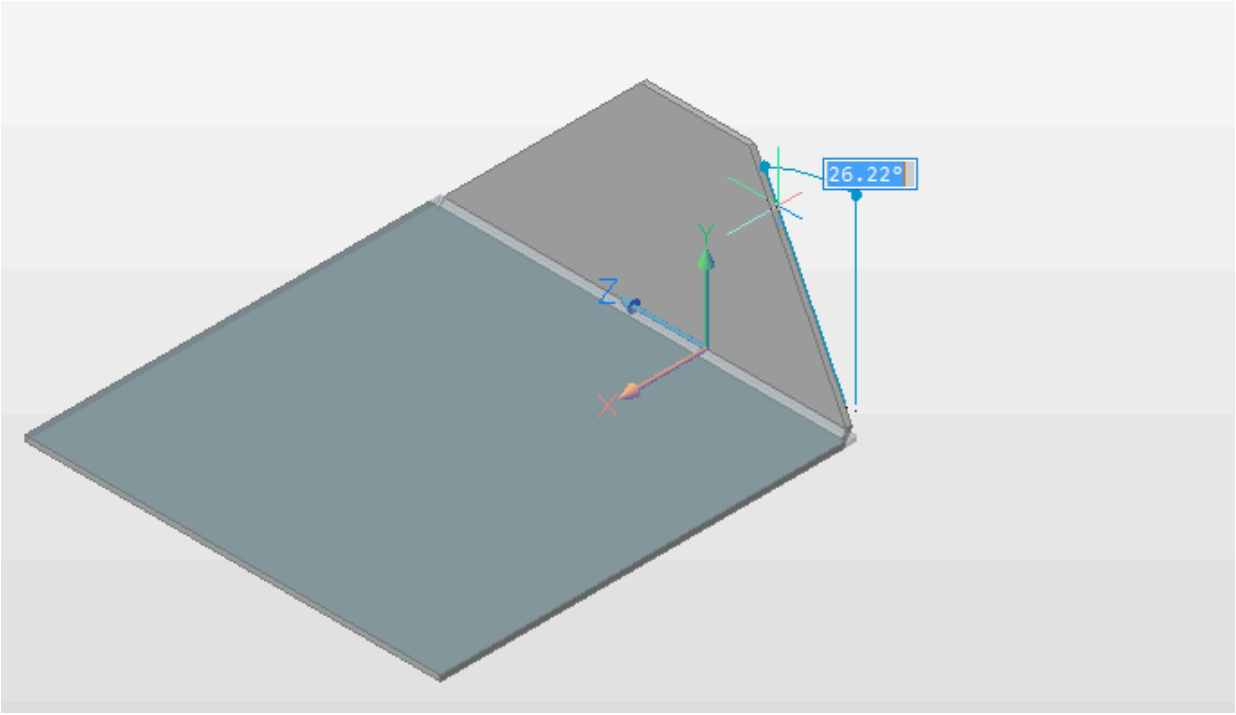


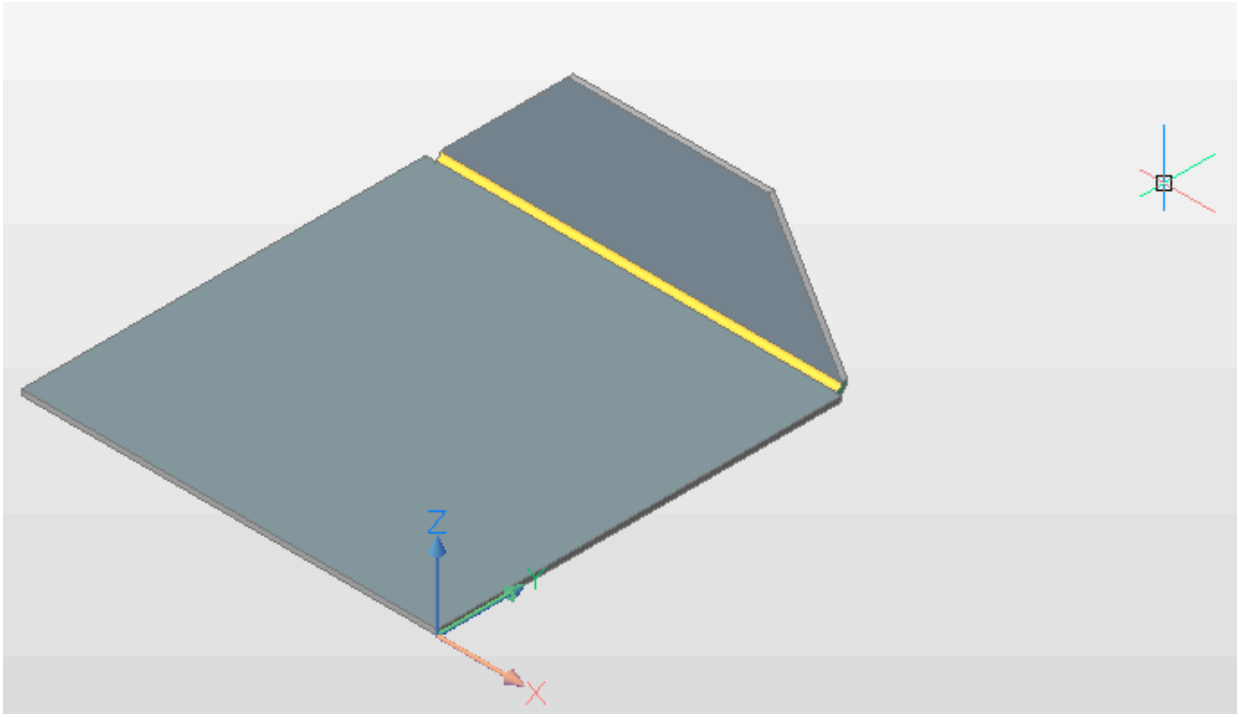
Nota: Quando você cria uma flange de borda de largura diferente os alívios de dobra são criados automaticamente.

21.5.10.4 Criar Flange de Borda Com Angulo(s) de Conicidade

Você pode criar flanges de borda trapezoidal com a opção **Angulo de afunilamento** do comando *SMFLANGEEDGE*:

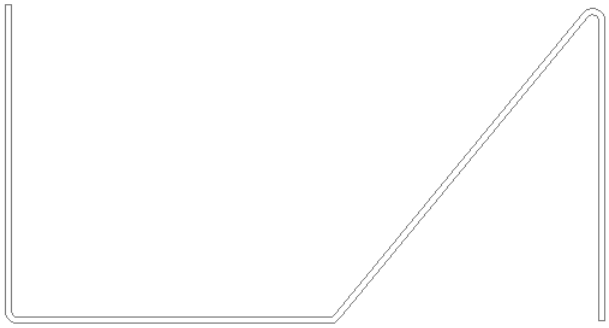






21.5.10.5 Criar Múltiplas Flanges por Extrusão de uma Polilinha

Existe uma classe simples, mas importante, de peças de chapa metálica que são vistas como polilinha engrossadas (espessadas), se nós observarmos a peça a partir de seu lado "frontal":

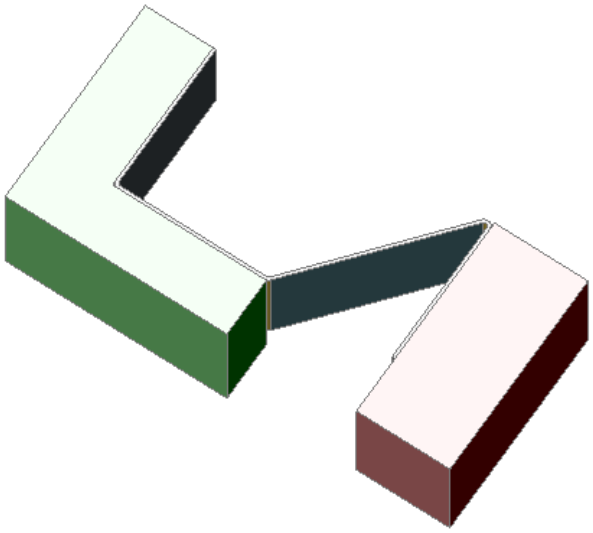


Para criar essa peça, você pode:

- usar chamadas sequenciais do comando *SMFLANGEEDGE*, ou
- desenhar uma polilinha e usar o comando *DMEXTRUDE*, seguido pelos comandos *DMTHICKEN*, *SMCONVERT*, e *SMBENDCREATE*.

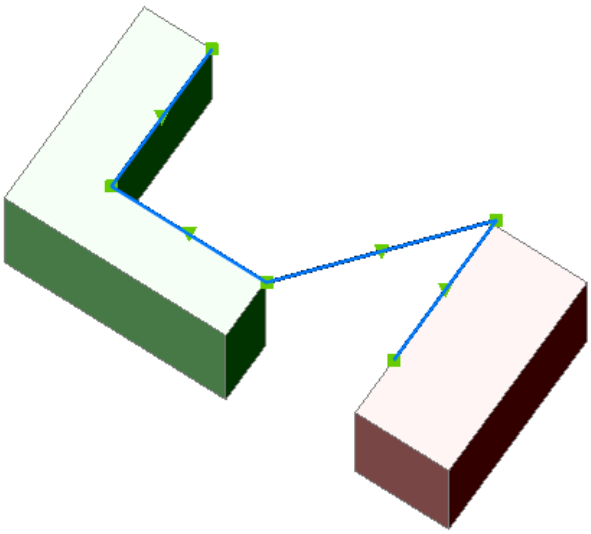
No entanto, recomendamos o seguinte procedimento:

O BricsCAD permite que você desenhe uma peça em uma montagem complexa com outras peças para que ela não interfira com outras entidades na montagem.



Você pode controlar a direção do espessamento de cada segmento de linha individualmente com *SMEXTRUDE*.

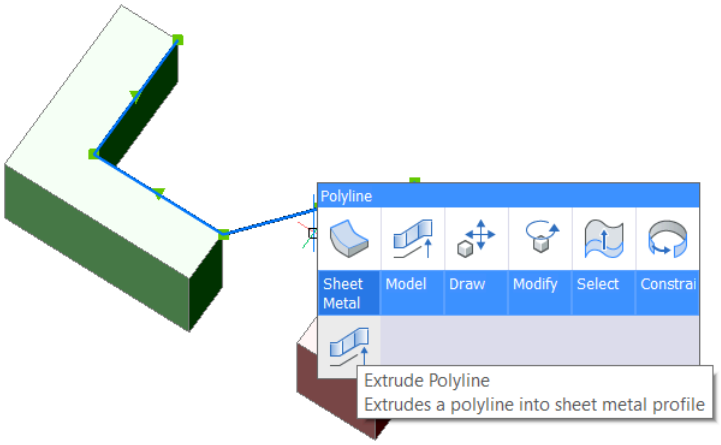
1 Desenhar uma polilinha que representa um esboço da futura peça de Sheet



Metal:

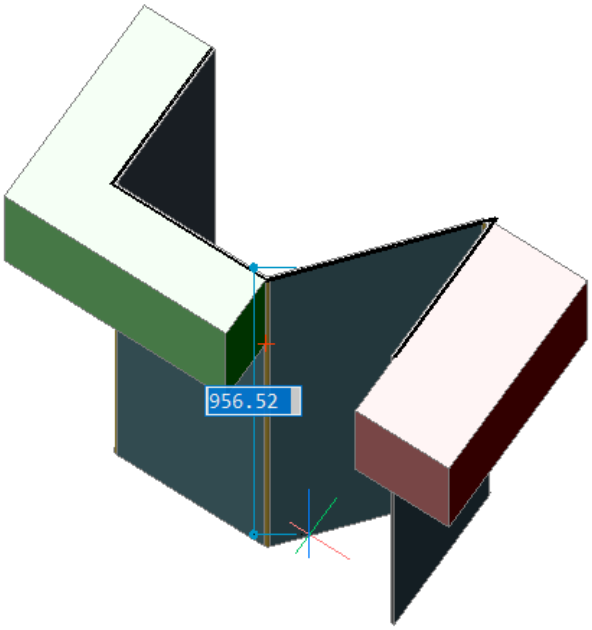
Faça um dos seguintes:

- Clique no botão de ferramenta **Extrudar Polilinha** () na barra **Sheet Metal**.
- Escolha **Extrudar Polilinha** no menu **Sheet Metal**.
- Escolha **Extrusao Polilinha** no grupo **Sheet Metal** do menu

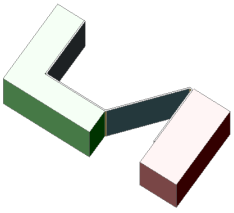
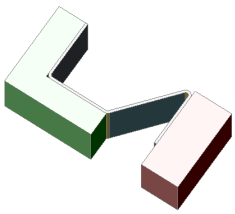
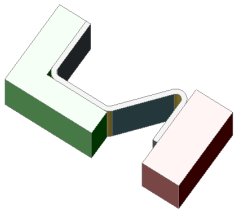


Quad:

2 Passe o cursor para selecionar a **altura da extrusão** ou entre o valor e pressione **Enter** para aceitar:



O corpo é criado e as extensões são respeitadas. A peça de chapa metálica não excede as dimensões da polilinha original. O comportamento é o mesmo, mesmo com espessuras diferentes:

Espessura 1 mm	Espessura 2 mm	Espessura 4 mm
		

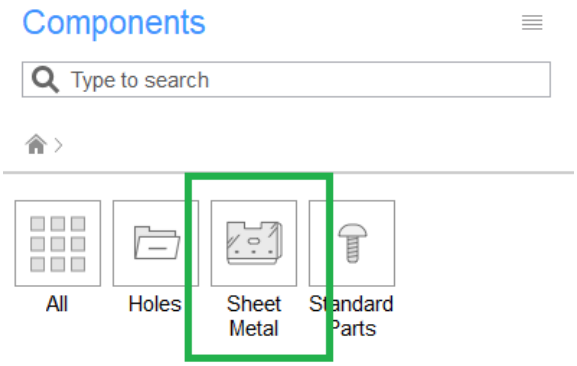
Nota: A política de preservar dimensões externas também funciona para peças de sheet metal criadas por outros fluxos de trabalho: construção de flange sequencial com *SMFLANGEEDGE*, dobras de peças com *SMFLANGEBEND* e reconhecimento da intenção de projeto de sheet metal com *SMCONVERT*. Se você quiser alterar a direção de extrusão de uma flange, aplique o comando *SMFLIP* a esta.

21.5.10.6 Extrusão de um Contorno para Criar uma Flange

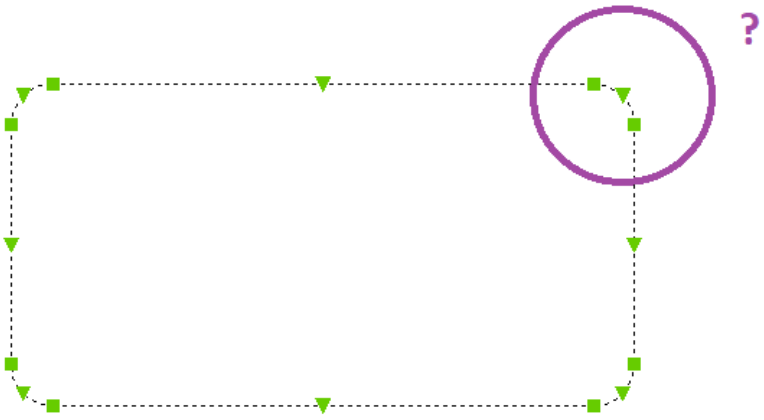
O comando *SMFLANGECONTOUR* permite extrusão de um determinado contorno para uma nova flange, e anexá-la ao modelo. Isto pode ser considerado uma sequência dos comandos *SMFLANGEBASE*, *SMFLANGECONNECT*, e *SMBENDCREATE*. Na realidade, este escolhe de forma inteligente as faces de espessura em flanges selecionadas e os conecta com as faces de espessura em uma nova flange.

Siga um dos procedimentos:

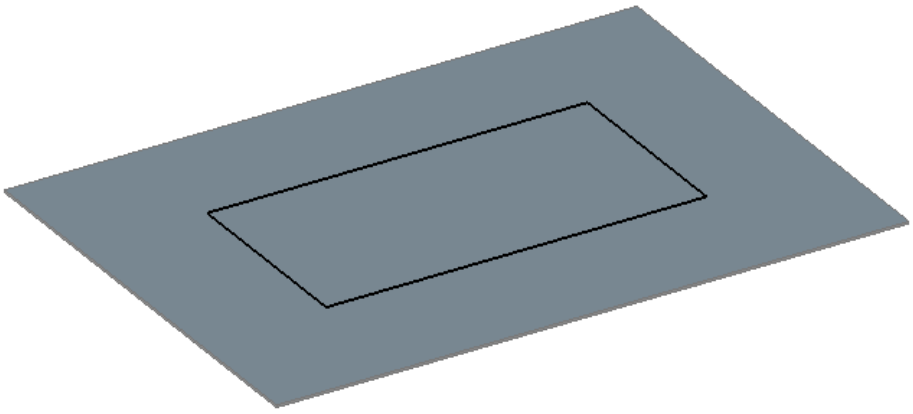
- Clique o botão **Criar Flange a partir do Contorno fechado** ( ) na barra **Sheet Metal**.
- Escolha **Criar Flange a partir do Contorno** no menu **Sheet Metal**.
- Escolha **Criar flange a partir do COMANDOSontorno** no grupo **Sheet Metal** do menu cursor **Quad**:



Uma vez todas as flanges necessárias estão selecionadas, clique no contorno:

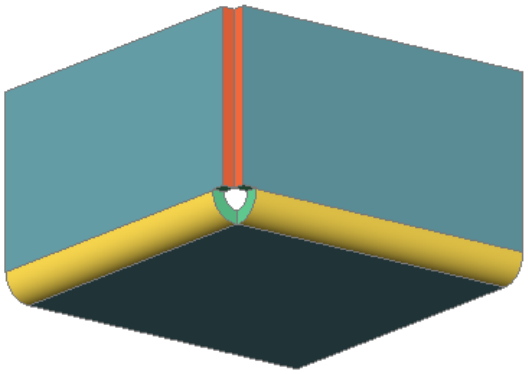


O comando cria uma nova flange e a mescla com as flanges selecionadas na peça:



21.5.10.7 Dobras, Alívios de Canto e Junções

Quando você puxa uma flange de borda com o comando *SMFLANGEEDGE*, é criado automaticamente um recurso de dobra entre a flange de borda e a flange base. Um recurso de alívio de canto é criado automaticamente em um canto onde três flanges se encontram. Esta operação também cria uma junção entre duas flanges que não estão conectadas por uma dobra.



Tutorial: Projetar a partir do zero no BricsCAD Sheet Metal

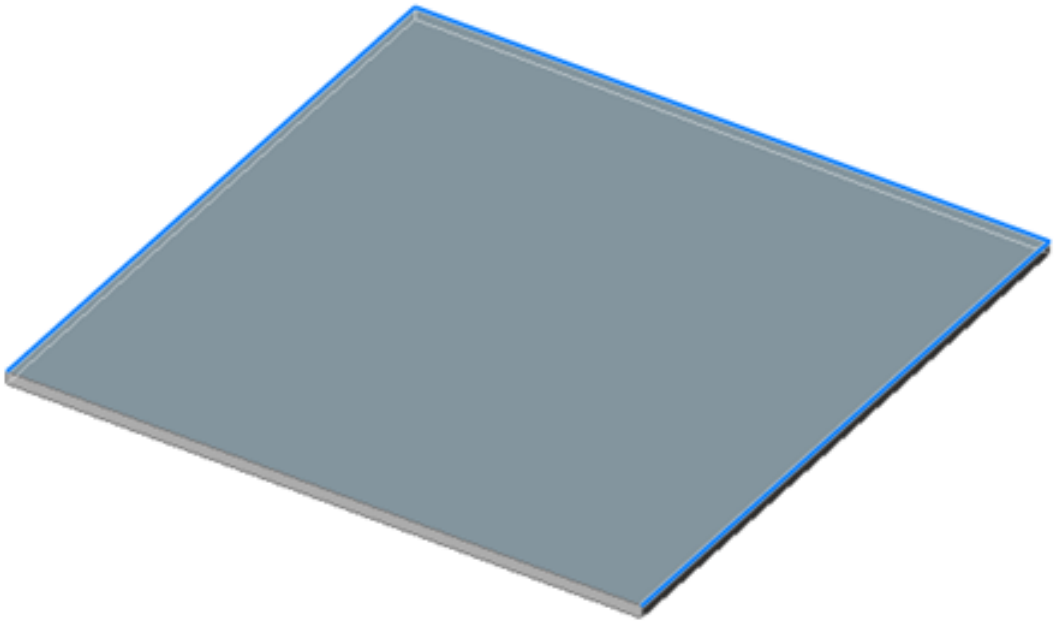
21.5.10.8 Criar Múltiplas Flanges

Você pode criar simultaneamente diversas flanges.

Siga um dos procedimentos:

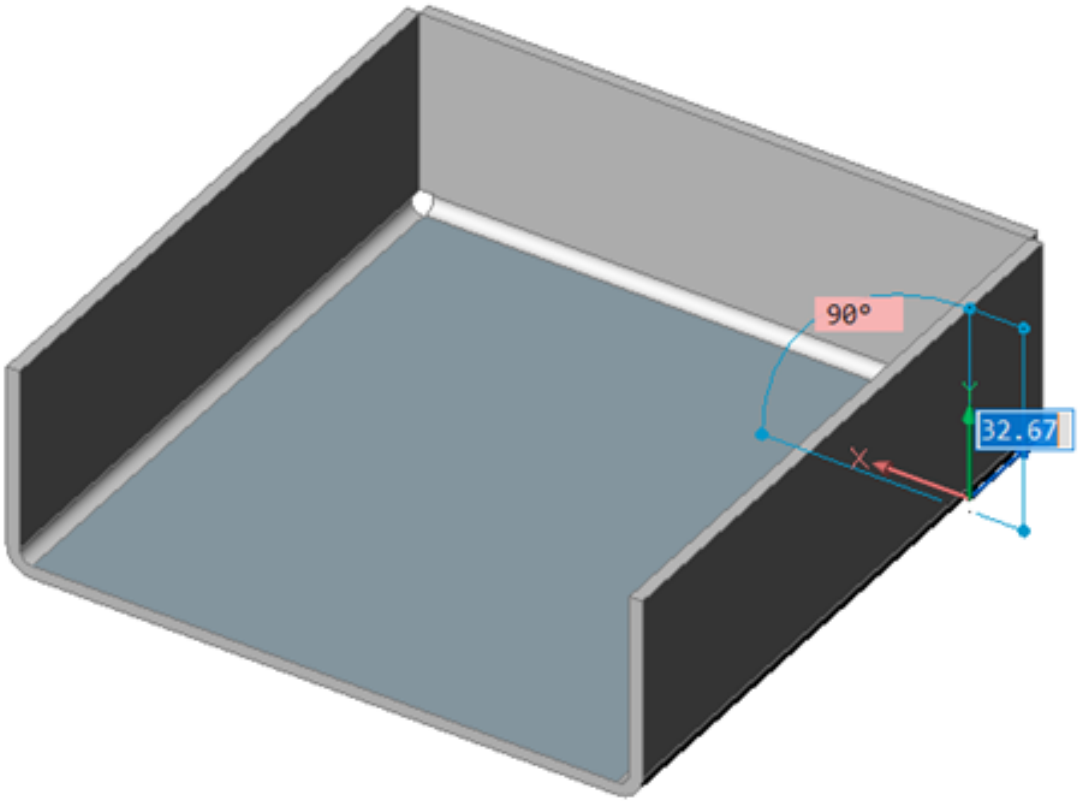
- Selecione várias bordas, e escolha **Criar Borda de Flange** ( ) no comando **Sheet Metal** grupo do menu de cursor **Quad**.

- Inicie o comando *SMFLANGEEDGE* e selecione múltiplas arestas.

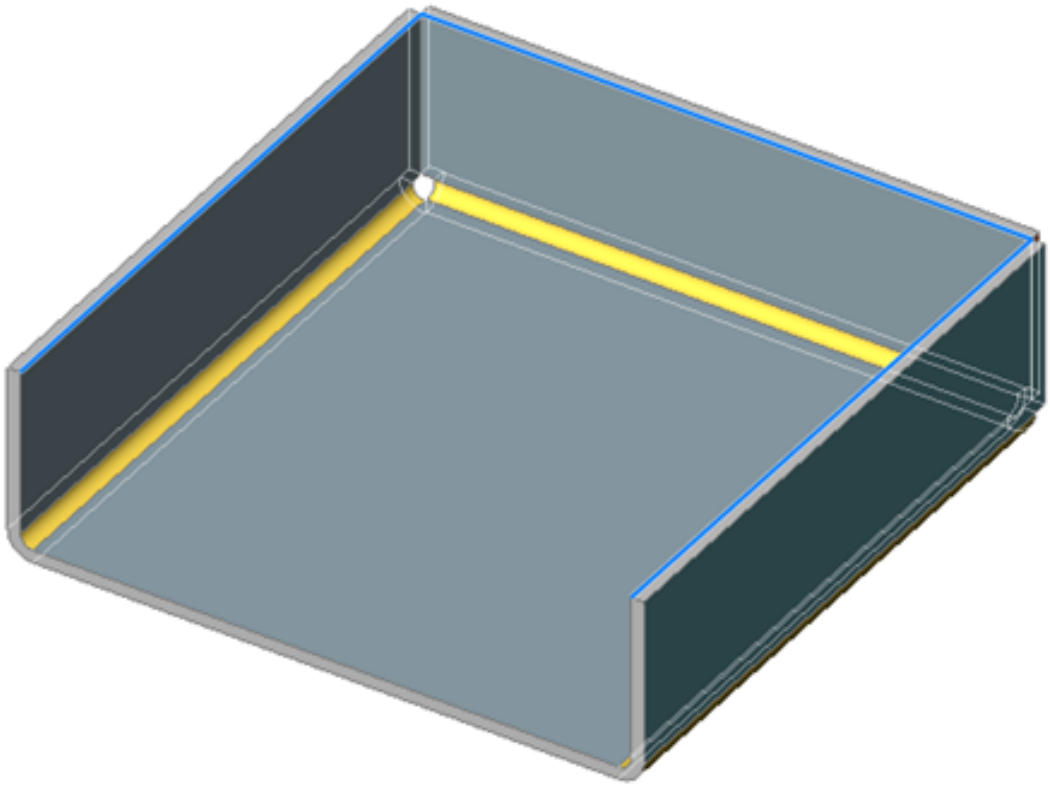


- Crie as bordas dinamicamente.

  Defina o ângulo e a altura do flange usando os campos de entrada dinâmicos. Pressione a tecla **TAB** para alternar entre o campo **Angulo** e o campo **Distância**.

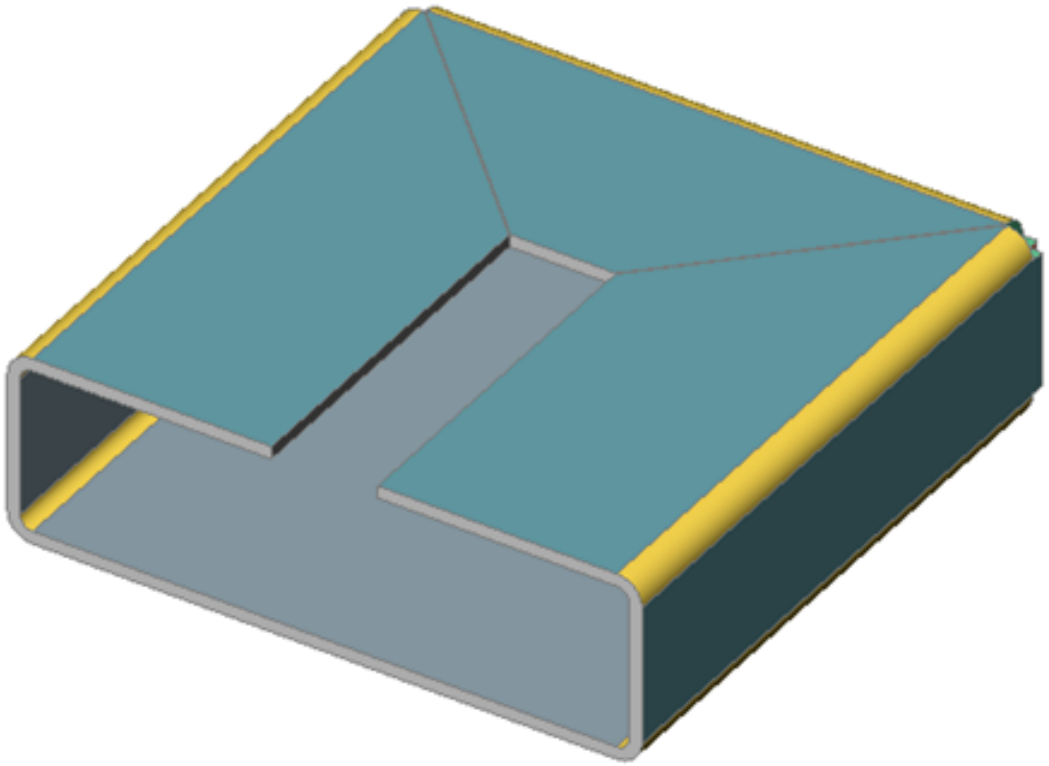

90°
32.67

- Selecione as arestas interiores das flanges recém-

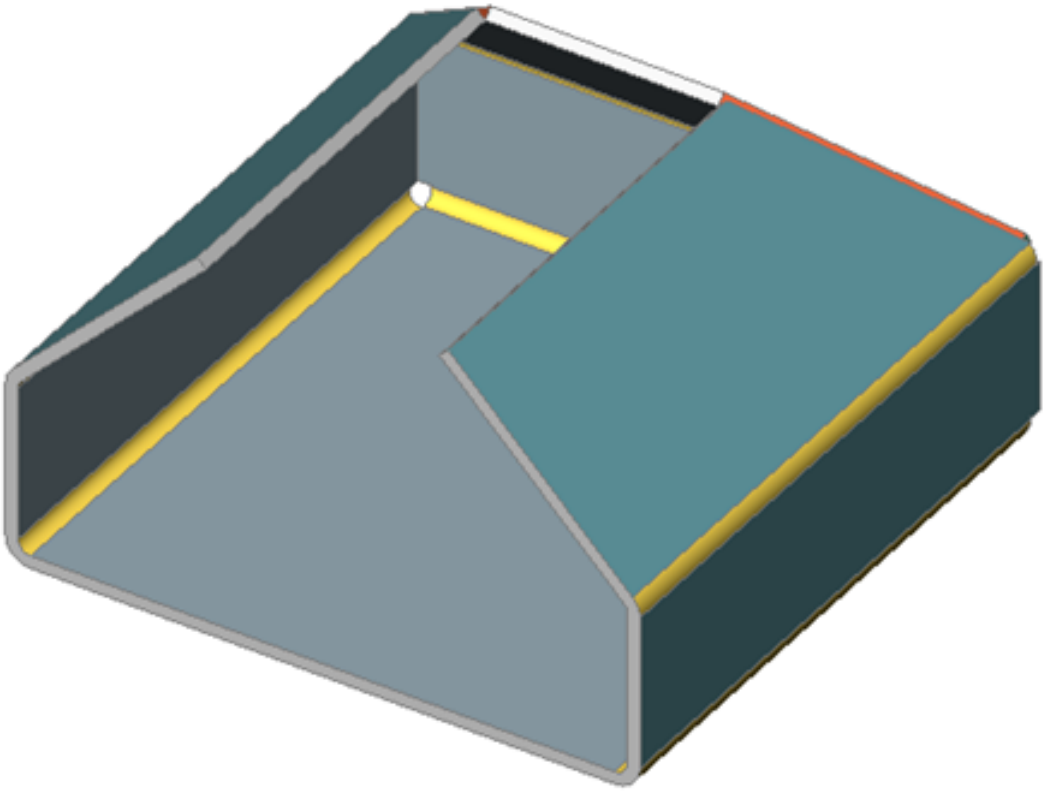


  criadas:
- Criar as flanges.

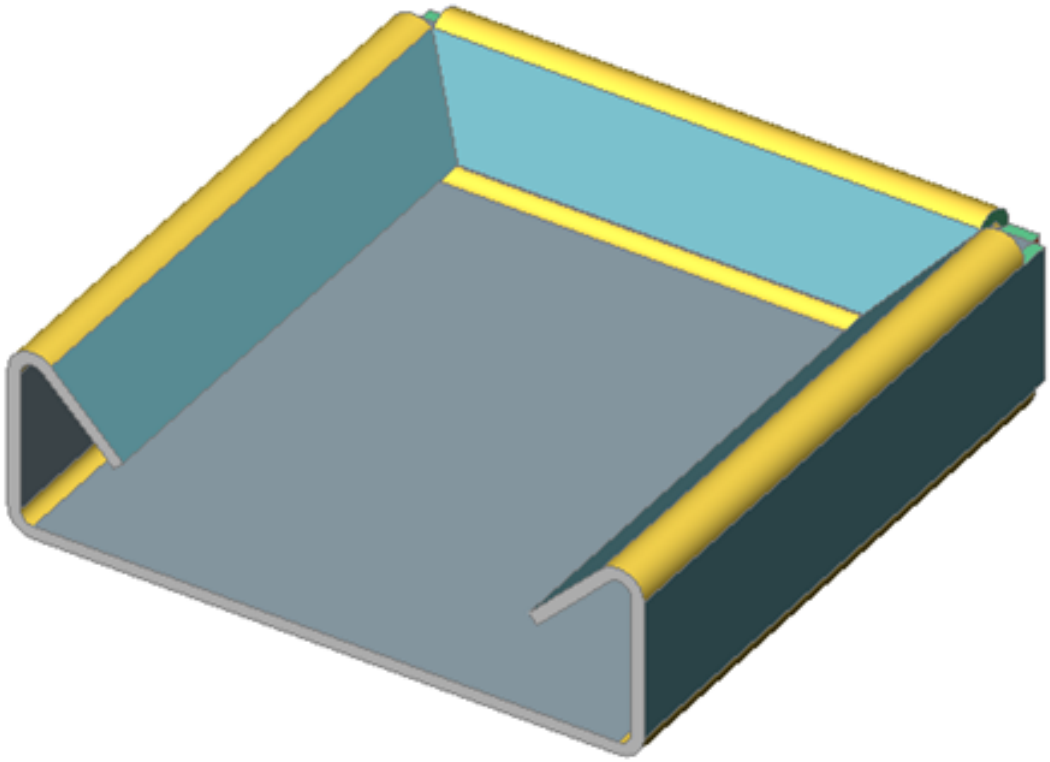
O comando calcula automaticamente a interseção das flanges e cria uma junção ou recursos de mitra. A 90 graus são criadas mitras:



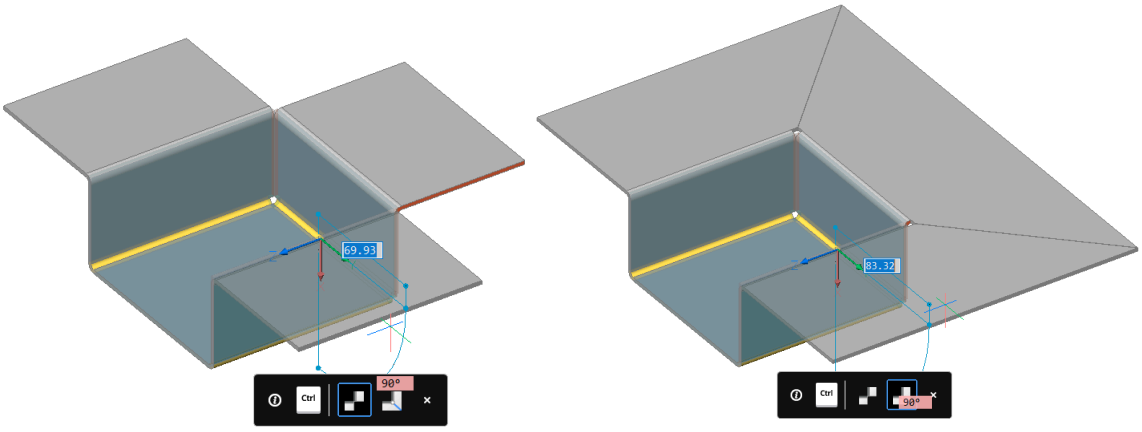
Para outros ângulos, são criadas junções:



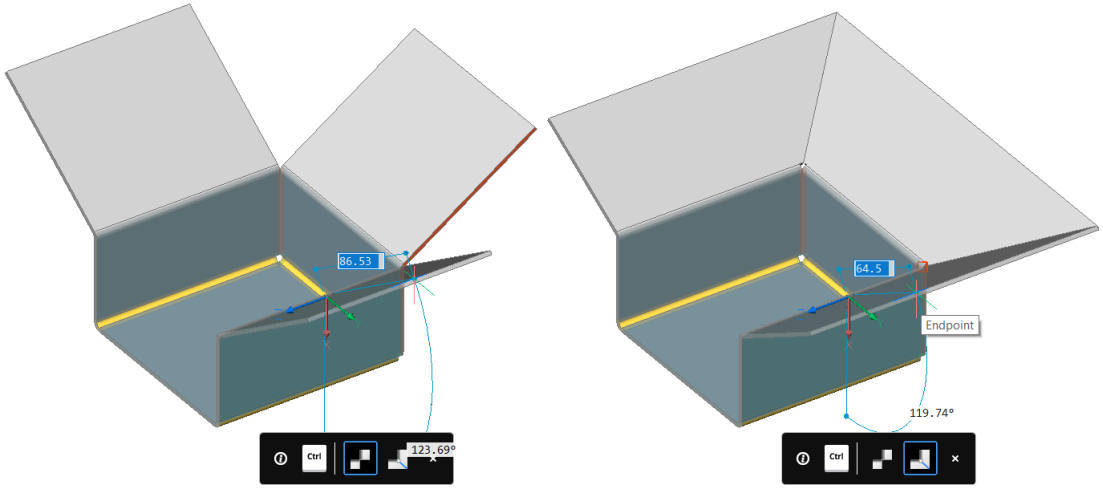
Em todos os casos, conflitos materiais são evitados:



Se as flanges forem criados externamente, use a opção **Alternar** (assim como **assistente de controle**) para controlar o tipo de conexão para mitras:



e junções:



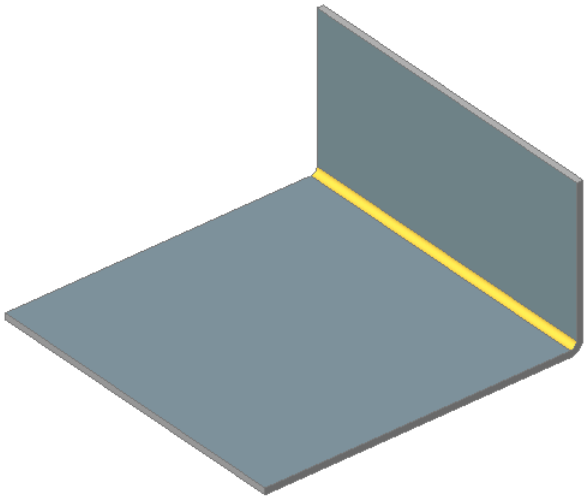
21.5.11 Modificar Flanges

O BricsCAD fornece várias ferramentas para modificar flanges em peças de chapa metálica. Essas ferramentas podem ser aplicadas de forma consistente a qualquer peça - independente de seu histórico de design. Neste artigo usamos ferramentas de Modelagem Sólida e ferramentas específicas de Sheet Metal. Quase todas as ferramentas de modelagem sólidas se aplicam a peças de chapa metálica, e ao mesmo tempo preservam a inteligência de projeto fornecida pelos Recursos de Sheet Metal.

21.5.11.1 Para alterar o tamanho e a forma da flange

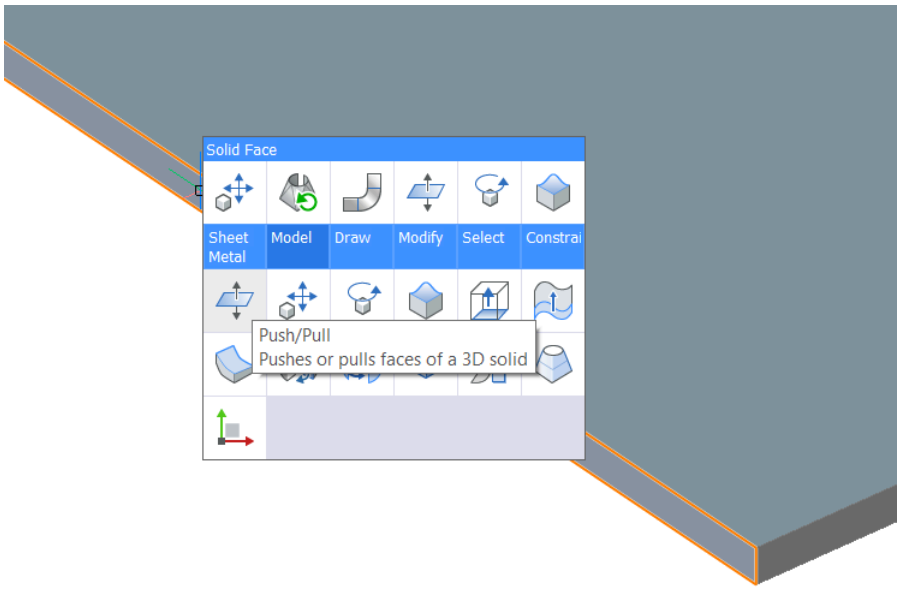
Em um determinado modelo simples, vamos aplicar várias operações de Modelagem Sólida para modificar a forma de suas flanges.

Nota: Trabalhamos apenas em faces de espessura do modelo



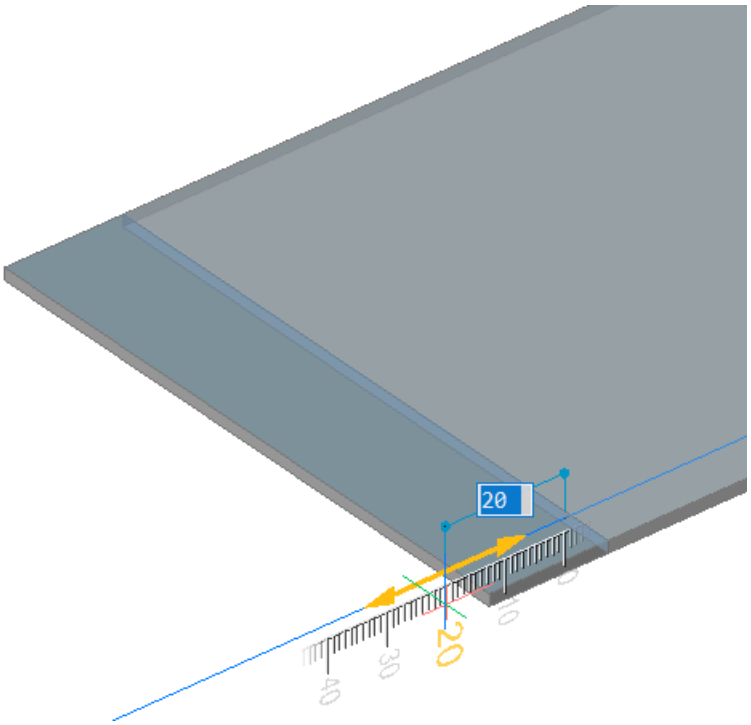
21.5.11.2 Para esticar flanges

1 Passe o mouse sobre a face de espessura e selecione **Empurrar/Puxar** no grupo de comandos



Modelo no menu **Quad**.

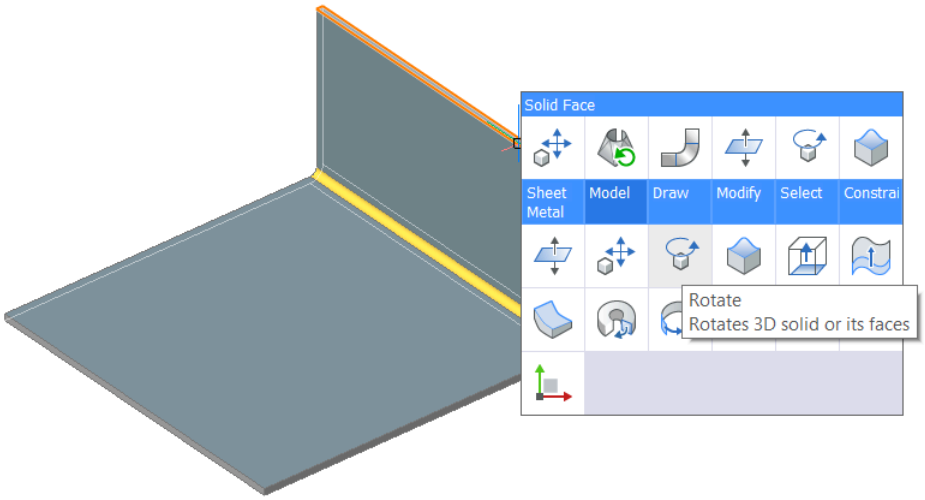
2 Arraste a face, e clique o botão-direito.



As mudanças serão aplicadas à peça, a flange é esticada ao longo da distância especificada.

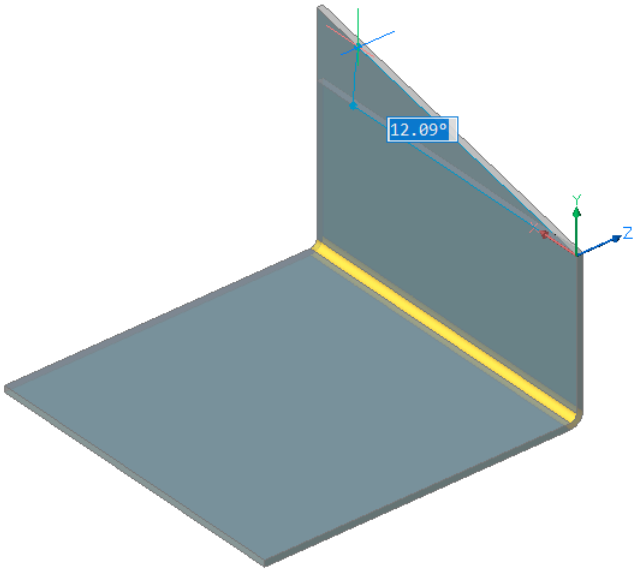
21.5.11.3 Para rotacionar uma face de espessura

1 Passe o mouse sobre a face e escolha **Rotacionar** no grupo de comando **Modelo** no menu



Quad.

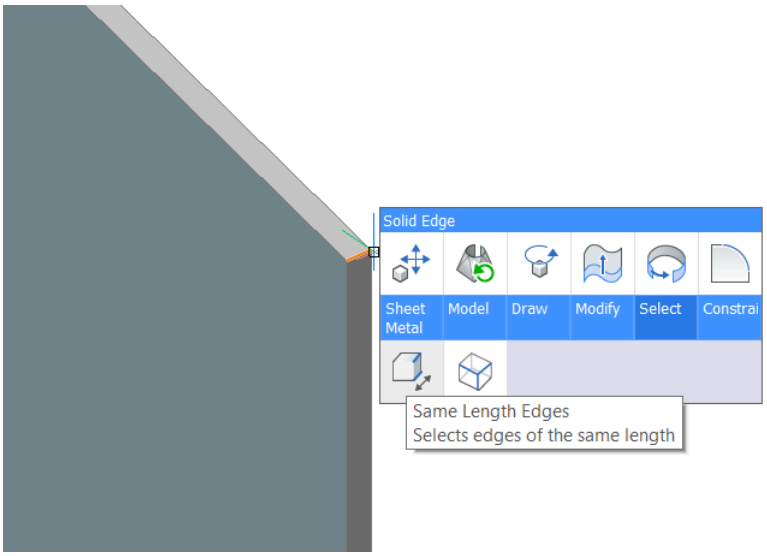
2 Especifique o eixo de rotação, o ponto base e o ângulo (para obter mais detalhes, consulte DMRotate



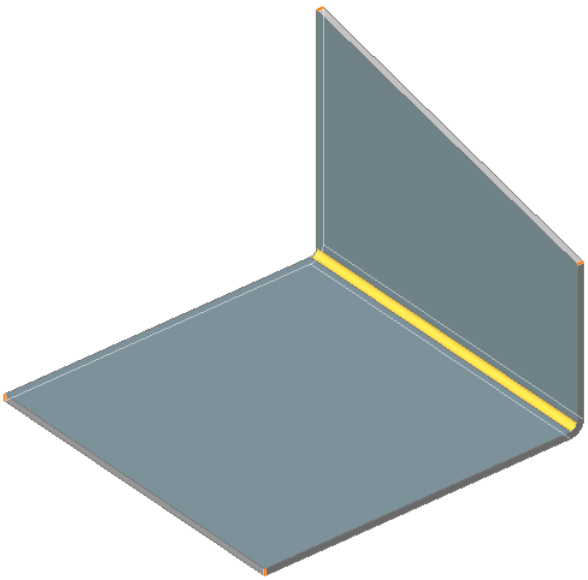
na Referência de Comando):

3 Preencha as bordas afiadas.

Passe o mouse sobre uma borda de espessura e escolha **Mesma borda de comprimento** ( ) no grupo de comando **Selecionar** no menu **Quad**.



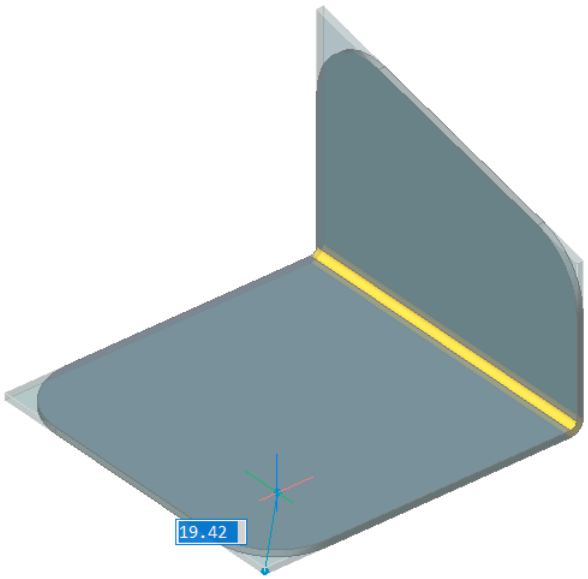
4 Pressione **Enter**. Bordas com o mesmo comprimento são

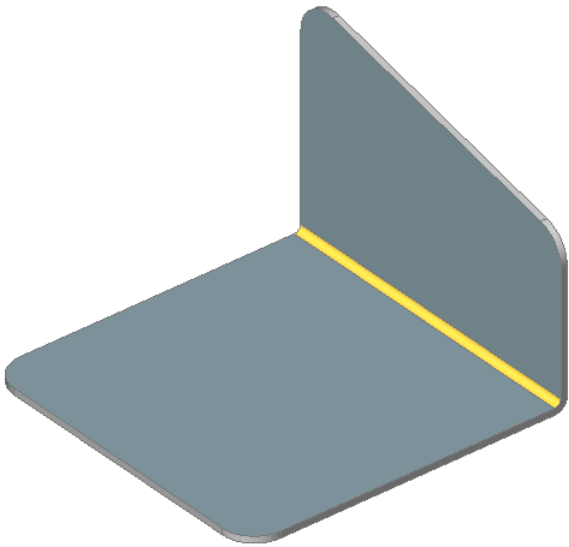


selecionadas.

5 Escolha **Concordar** no grupo de comandos **Modelo** no **Quad**.

6 Especificar o raio de concordância

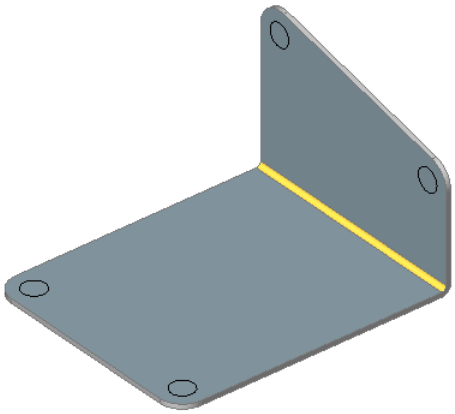




desejado:

21.5.11.4 Para fazer um furo em uma flange

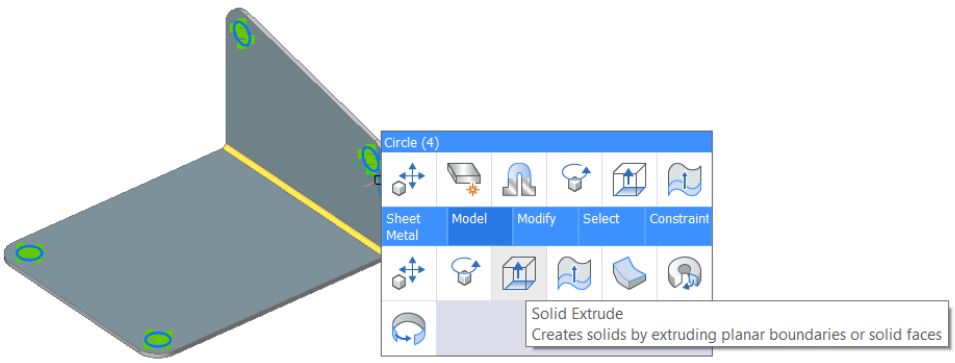
1 Crie 4 círculos conforme indicado na imagem abaixo. Use o snap de entidade Snap ao Centro para



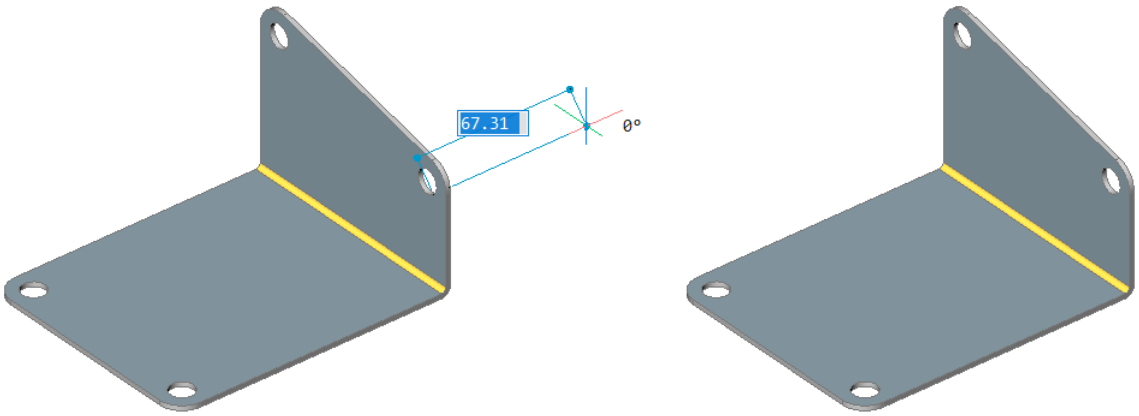
definir os pontos centrais.

2 Selecione os círculos.

3 Escolha **Solid extrude** no grupo de comandos **Modelo** no menu **Quad**.



4 Mova o cursor para criar os

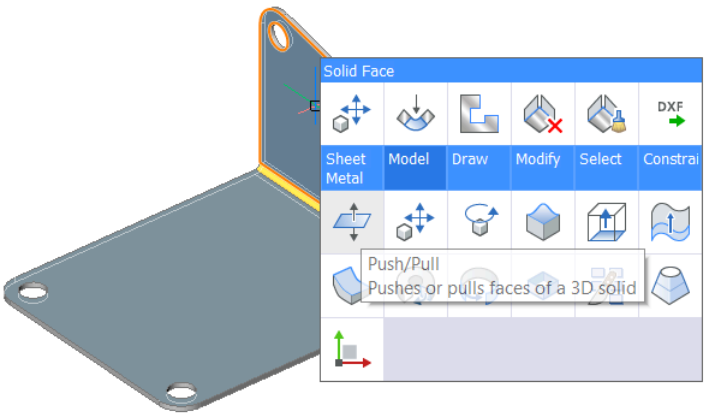


furos.

21.5.11.5 Para mover uma flange

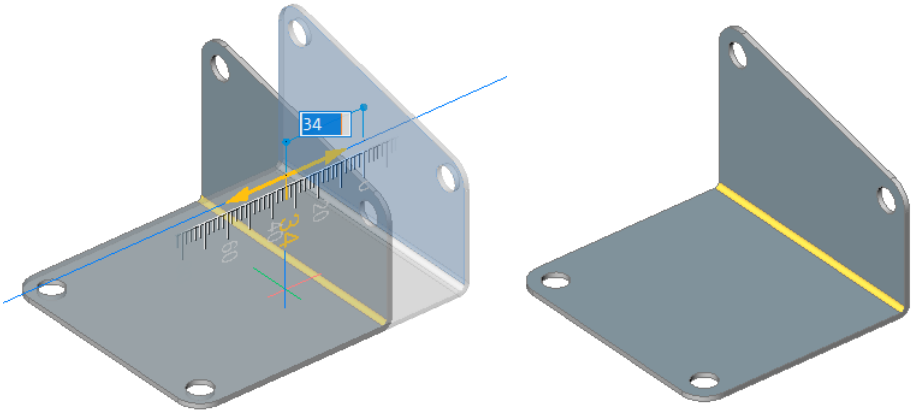
1 Siga um dos procedimentos:

- Clique o botão **Empurrar/Puxar** ( ) na barra **Modelagem Direta**.
- Escolha **Modelagem Direta > Empurrar/Puxar** no menu **Modelo**.
- Escolha **Empurrar/Puxar** ( ) no painel **Editar** da aba **Sheet Metal** na Faixa de opções.
- Execute o comando *DMPUSHPULL* para determinada face.
- Passe o mouse sobre a face que você deseja mover, e escolha **Empurrar/Puxar** no grupo de comandos **Modelo** no menu **Quad**.



2 Selecione a nova posição da flange.

Nota: Devido à presença de **Recursos de Sheet Metal**, a espessura da peça é preservada.



21.5.11.6 Para rotacionar uma flange

Embora seja possível rotacionar uma flange utilizando o comando *DMROTATE* (), é recomendado o uso de *SMFLANGEROTATE* () em vez disso, porque este comando seleciona automaticamente o eixo de rotação, para respeitar a intenção de projeto da peça de Sheet Metal.

1 Inicie o comando *SMFLANGEROTATE* ().

Você é solicitado: Selecione uma face de flange para rotacionar:

Face de flange sob o cursor se destacam.

2 Clique para selecionar uma face de flange.

A flange rotaciona dinamicamente.

Uma dimensão dinâmica exibe o ângulo atual, em relação à flange base.

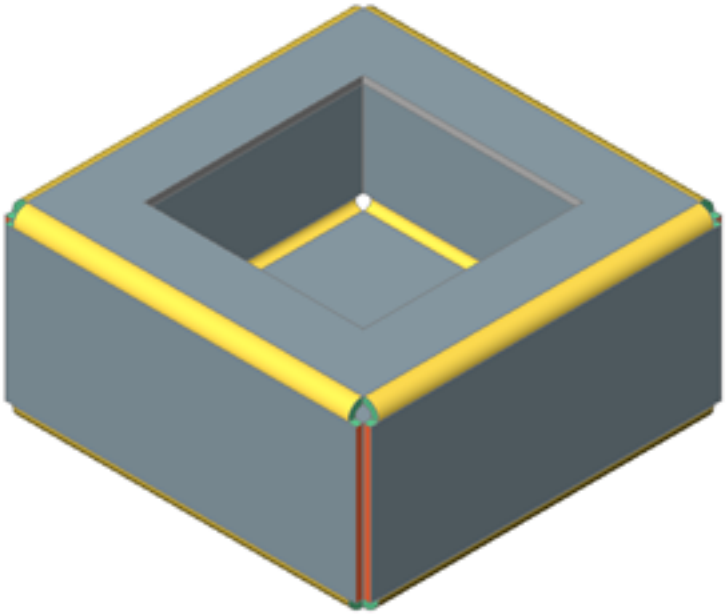
3 Siga um dos procedimentos:

- Selecione um ponto.
- Digite um valor no campo de dimensão dinâmica.
- Pressione a tecla **TAB**, e especifique o ângulo absoluto de rotação no campo da dimensão dinâmica.

Nota: Quando o comando *DMROTATE* é usado para rotacionar um flange, a qual está conectada a outras flanges por recursos de junção, esses recursos são automaticamente dissolvidos antes da rotação.

21.5.11.7 Para dividir uma flange

Em alguns casos é melhor dividir uma flange para minimizar o consumo de material, por exemplo, quando uma flange tem um grande furo, como é mostrado na imagem abaixo:



1 Inicie o comando *SMSPLIT* ( ).

Você é solicitado: Selecione o flange ou a face de dobra em loft:

2 Selecione uma face de flange.

Você é solicitado:

Selecione linhas ou bordas para dividir a flange ou [usar Divisão Inteligente/desenhar uma Nova linha]<desenhar uma Nova linha>:

3 Se nenhuma linha existir ainda na face da flange, clique o botão-direito.

Você é solicitado: Ponto inicial da linha:

4 Especificar o ponto inicial da linha.

Você será solicitado: Extremidade da linha:

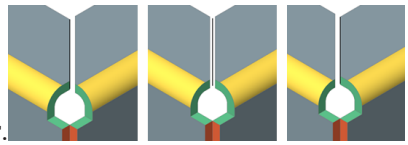
5 Especificar o ponto final da linha:

Você será solicitado: Dividir pelo Centro/Esquerda/Direita/<Aceitar modelo>:

6 Para ajustar a posição de corte em relação à linha, faça um dos seguintes:

- Pressione a tecla **CTRL** para alternar as opções **Centro**/**Esquerda**/**Direita**.

  Se as **DICAS** estiverem Ligadas, a opção atualmente selecionada é indicada no Assistente de Dicas.
- Selecione **Centro**, **Esquerda** ou **Direita** no menu Prompt.
- Digite uma opção na barra de Comando, e pressione **Enter**.



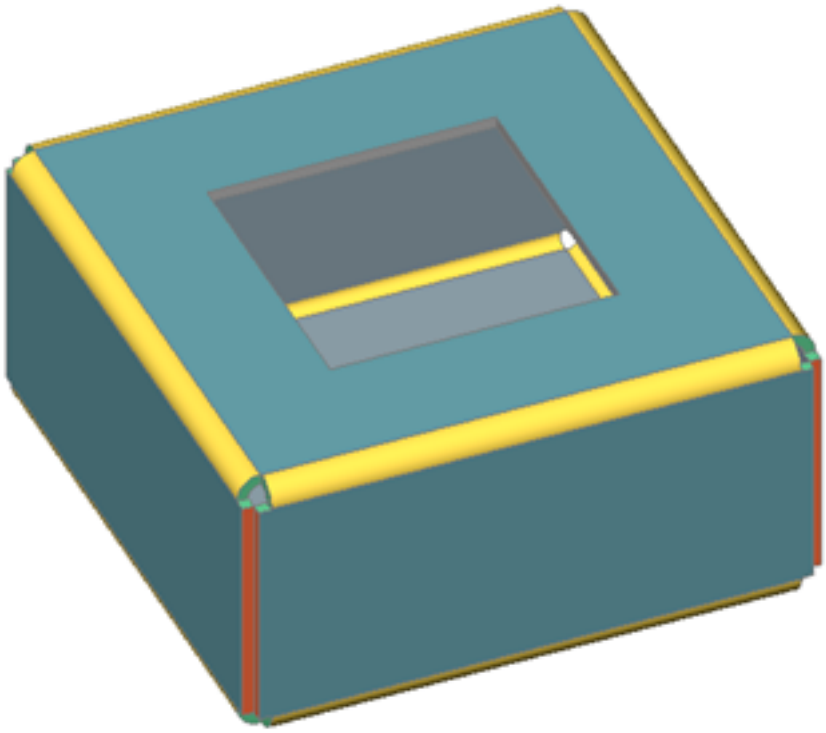
A flange é cortada pela linha: esquerda, centro ou direita.

Nota: uma operação de divisão cria um recurso de mitra. Você pode restaurar a flange inicial removendo o recurso de mitra da lista de recursos no **Navegador de Mecânica**.

21.5.11.8 Ferramentas inteligentes para dividir uma flange

O cenário abaixo ilustra os outros recursos do comando *SMSPLIT*:

- Divisão inteligente ao selecionar um único vértice no canto.
- Dividir para fora a dobra e converter a peça adjacente ao canto, em uma



junção.

1 Siga um dos procedimentos:

- Clique a ferramenta **Usar Divisão Inteligente da Flange** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Usar Divisão Inteligente da Flange** no menu **Sheet Metal**.
- Digite *SMSPLIT* na barra de Comando.

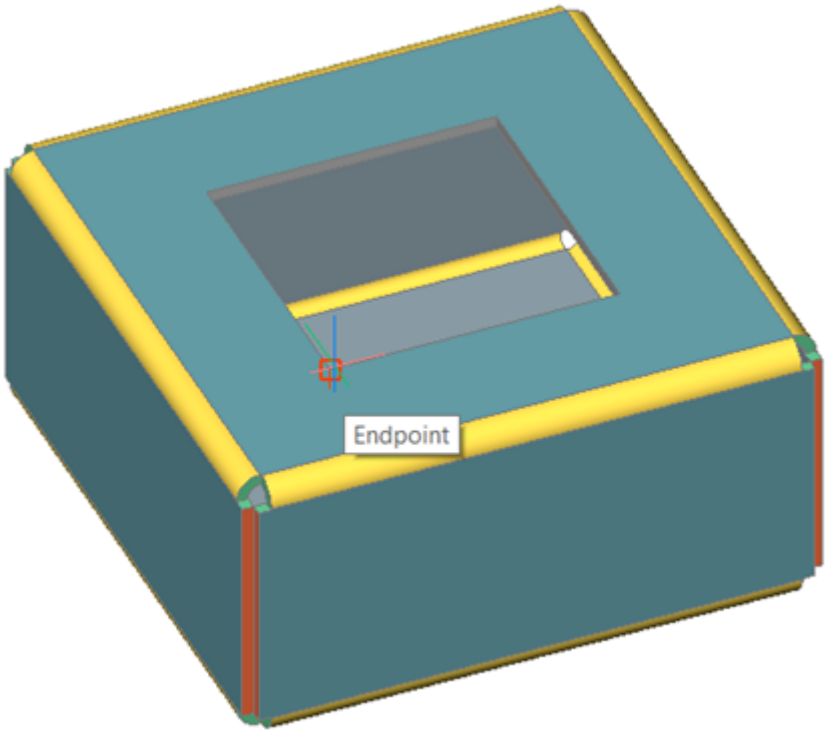
  Você é solicitado: Selecione o flange ou a face de dobra em loft:

2 Selecione uma face de flange.

Você é solicitado:

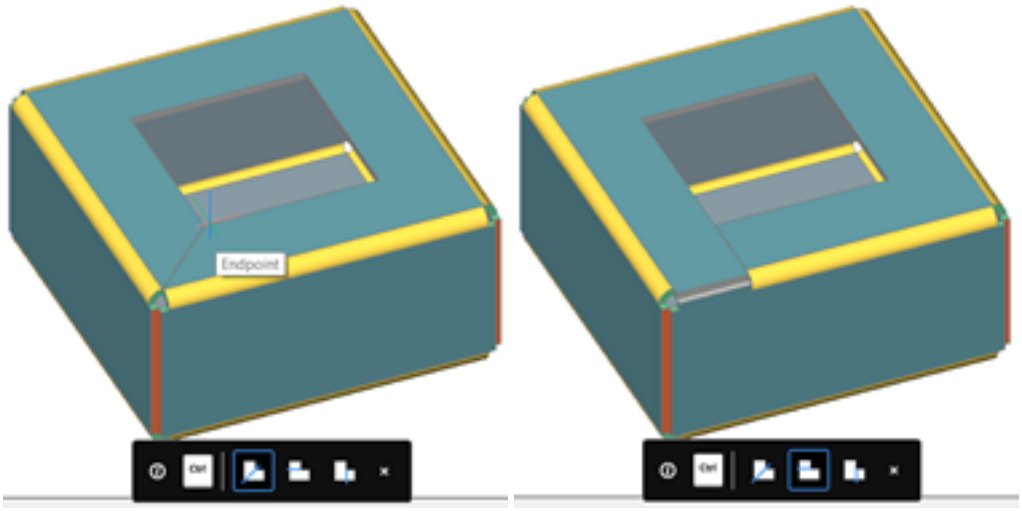
Selecione linhas ou bordas para dividir a flange ou [usar Divisão Inteligente/desenhar uma Nova linha]<desenhar uma Nova linha>:

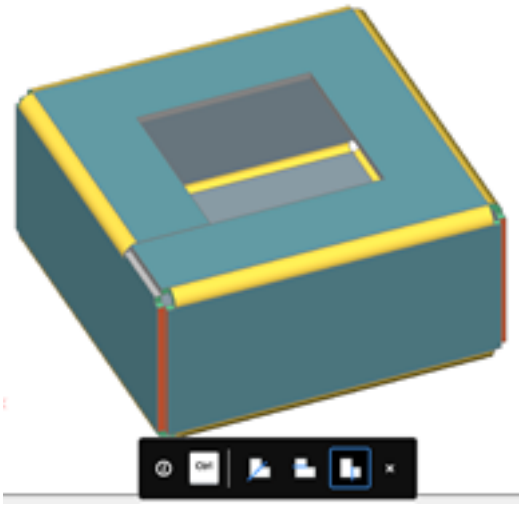
3 Selecione a opção **Usar divisão inteligente**, então selecione um vértice de canto:



Você será solicitado: Faça divisão [Centro/Esquerda/Direita/Aceitar]<Aceitar>:

As opções são:





Nota: A conversão da peça da dobra em uma junção não está relacionada ao modo **Smart Split** (Divisão Inteligente).

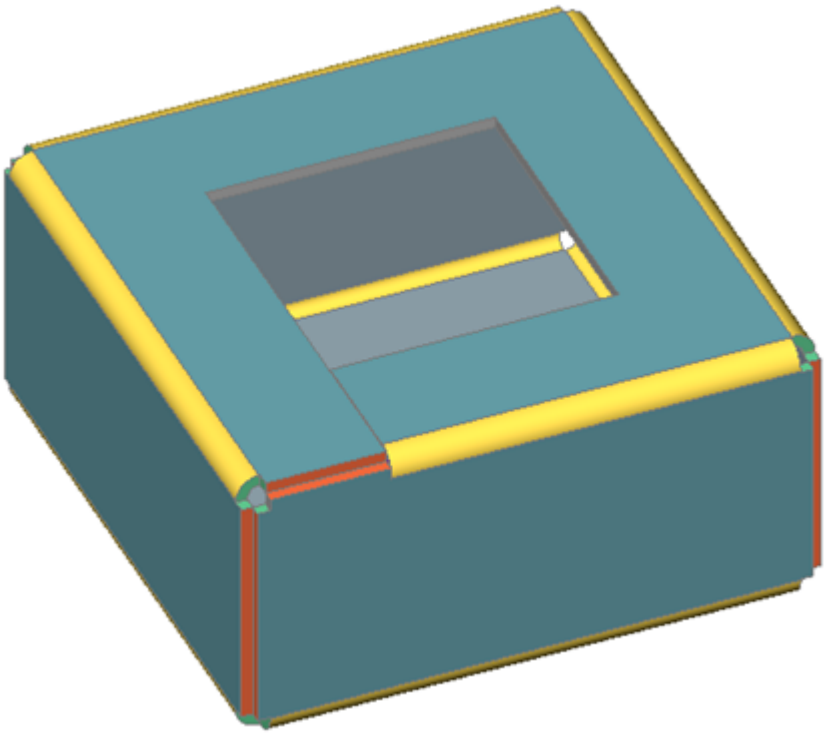
4 Para alternar as opções, siga um destes procedimentos:

- Pressione repetidamente a tecla **CTRL**.

  Se as **DICAS** estiverem Ligadas, a opção atualmente selecionada será indicada no Assistente Dicas.

- Selecione **Centro**, **Esquerda** ou **Direita** no menu Prompt.
- Digite uma opção na barra de Comando, e pressione **Enter**.

  A flange é cortada por uma linha, a dobra é parcialmente convertida na junção:



No caso da opção **Centro** se os **Recursos Inteligentes** estiverem ligados, os alívios de canto adjacentes à divisão serão convertidos em alívios de dobra.

21.5.11.9 Para dobrar uma flange

O comando *SMFLANGEBEND* dobra uma ou mais flanges usando uma linha. Você pode tanto desenhar a linha primeiro, e depois iniciar o comando, como usar a opção **Nova Linha** do comando.

Para dobrar várias flanges simultaneamente

1 Inicie o comando *SMFLANGEBEND*.

Você é solicitado: Selecione uma face de flange:

2 Selecione a flange que deseja dobrar.

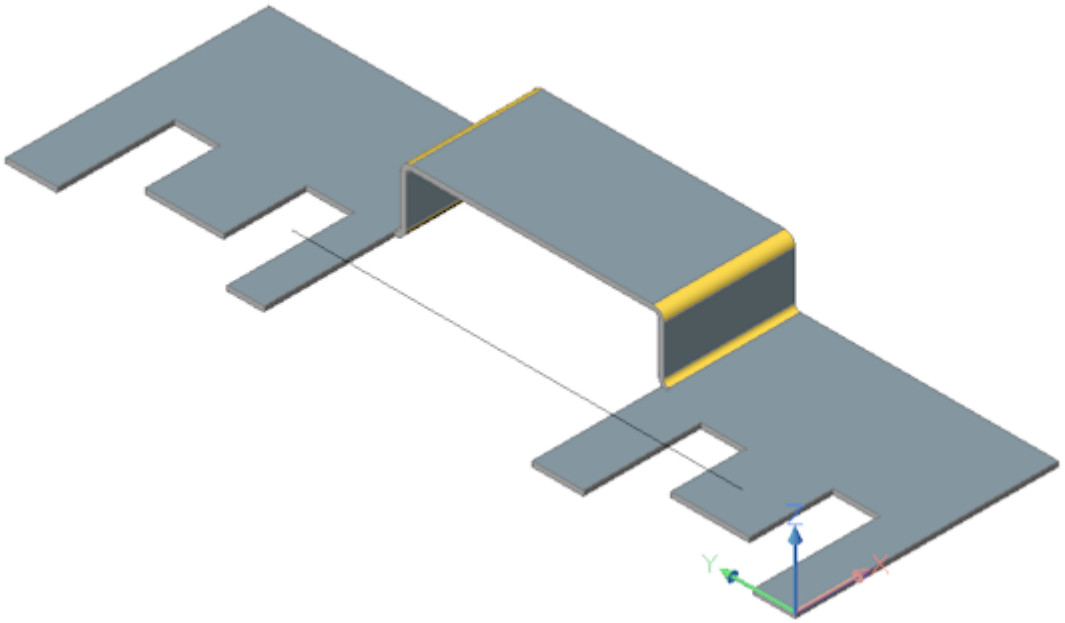
Você é solicitado: Selecione linha ou borda para dobrar o sólido ou desenhar uma [Nova linha] <Nova linha>:

3 Siga um dos procedimentos:

- Selecione uma linha que se sobreponha à(s) flange a ser dobrada.
- Escolha a opção **Nova Linha**.

  Você é solicitado a desenhar uma linha na parte superior da(s) flange a ser dobrada.

4 Você é solicitado: Escolha um ponto no lado que será movido:



Escolha um ponto no lado da linha que você deseja mover.

A peça selecionada é dobrada dinamicamente.

Você é solicitado: Entre o ângulo de dobra ou defina [Raio/Trocar lado]:

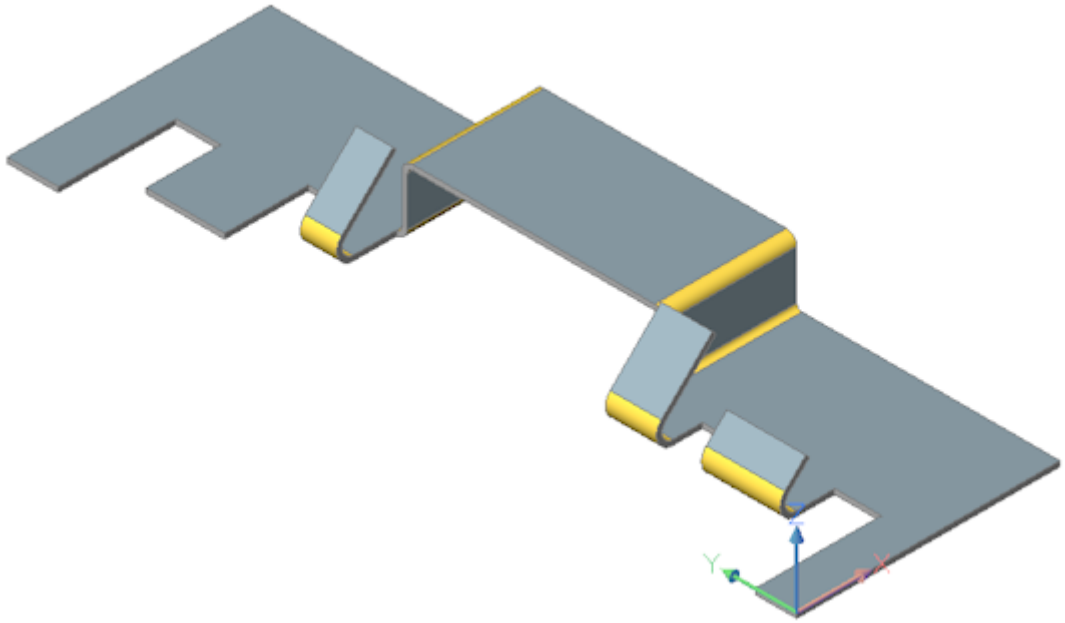
5 (opção) Digite um valor para definir o ângulo de dobra, usando o raio de dobra padrão.

6 (opção) Selecione a opção **Raio**.

- Você é solicitado: Entre o raio de dobra [Voltar] <Voltar>:

  Digite um valor para substituir o raio de dobra padrão, e pressione **Enter**.
- Você é solicitado: Entre o ângulo de dobra ou defina [Raio/Trocar lado]:

  Digite um valor para definir o ângulo de dobra.



21.5.11.1 Para conectar flanges

0

O comando *SMFLANGECONNECT* fecha lacunas entre 2 flanges orientadas arbitrariamente.

1 Inicie o comando *SMFLANGECONNECT*.

Você é solicitado: Selecione faces de espessura planar de duas flanges:

Faces de flange sob o realce realçam.

2 Selecione a face de espessura da primeira flange.

Você é solicitado:

Entidades/subentidades no conjunto: 1

Selecione faces de espessura planar de duas flanges:

3 Selecione a face de espessura da segunda flange.

As faces de espessura selecionadas são conectadas.

Nota: Depois de conectar 2 flanges, você pode converter o recurso de junção em uma dobra.

21.5.12 Trabalhar com Dobras e Junções

Duas flanges adjacentes, de peças de sheet metal são conectadas por uma dobra ou uma junção. Diferentemente de muitos outros sistemas CAD de Mecânica, no BricsCAD, dobras e junções são recursos

separados. Uma dobra pode ser trocada para uma junção, e vice-versa, o que permite a você corrigir ou otimizar facilmente uma representação desdobrada de uma peça de Sheet Metal.

21.5.12.1 Converter Bordas Rígidas em Dobras ou Junções

Se as flanges adjacentes de uma peça de Sheet Metal não estiverem conectadas por uma dobra ou junção, a peça não poderá ser desdobrada ou exportada para um sistema CAM. Portanto, tais arestas rígidas devem ser convertidas em dobras ou junções.

Converter bordas rígidas em dobras

1 Siga um dos procedimentos:
   - Passe o mouse sobre uma borda rígida e clique na ferramenta **Criar Dobra** () no menu **Quad**;
   - Clique no botão **Criar Dobra** () na aba **Sheet Metal** da Faixa de opções;
   - Escolha **Criar Dobra** no menu **Sheet Metal**;
   - Digite *SMBENDCREATE* na barra de Comando.

     Você é solicitado:

     Selecione bordas rígidas ou junções ou flanges ou sólidos 3D ou [Modelo inteiro] <Modelo inteiro>:

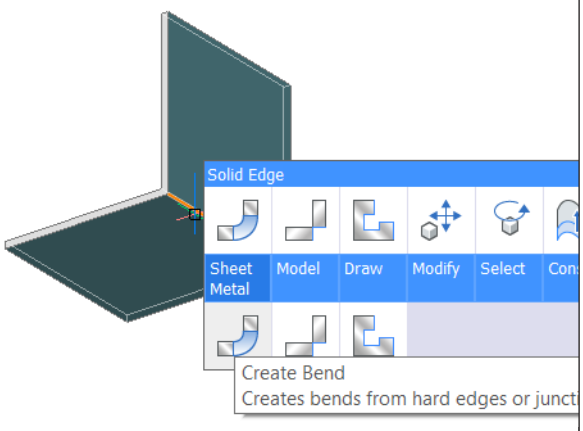
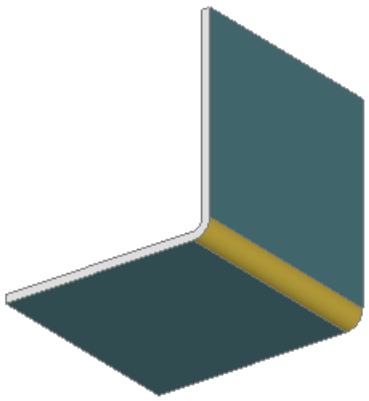
2 Selecione uma face de flange ou uma borda rígida.

   Você é solicitado:

   Entidades no conjunto: 1

   Selecione faces de flange ou bordas rígidas:

3 Siga um dos procedimentos:
   - Selecione mais faces ou bordas rígidas.
   - Clique o botão-direito ou a tecla **Enter** para criar as dobras ou junções nos itens selecionados.

	
Borda rígida selecionada	Uma dobra é criada

Converter Bordas Rígidas em Junções

1 Siga um dos procedimentos:

- Passe o mouse sobre uma borda rígida e clique na ferramenta **Criar Junção** (⌟) no menu **Quad**.
- Clique no botão **Criar Junção** (⌟) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Junção** no menu **Sheet Metal**.
- Digite *SMJUNCTIONCREATE* na barra de Comando.

  Você é solicitado: Selecione bordas rígidas ou dobras ou flanges ou sólidos 3D [Modelo inteiro] <Modelo inteiro>:

2 Selecione uma face de flange ou uma borda rígida.

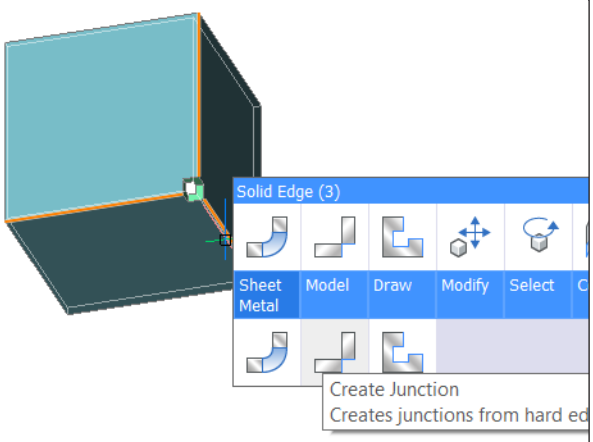
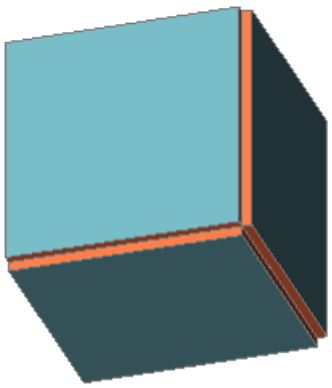
Você é solicitado:

Entidades no conjunto: 1

Selecione faces de flange ou bordas rígidas:

3 Siga um dos procedimentos:

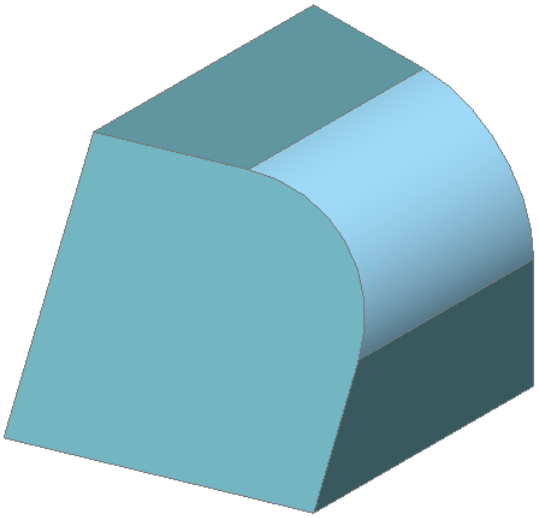
- Selecione mais faces ou bordas rígidas.
- Clique o botão-direito ou a tecla **Enter** para criar as dobras ou junções nos itens selecionados.

	
Bordas rígidas selecionadas. Observe que um alívio de canto deve ser criado primeiro	Junções criadas. Observe que cortes de material desnecessários foram removidos devido aos **Recursos Inteligentes**.

21.5.12.2 Criar Junções Curvas

Vamos dar uma olhada em um cenário mais complexo em que uma junção tem que ser criada sobre uma borda não linear. Para tornar a peça desdobrável, devemos:

- Cortar o alívio da dobra para uma das bordas rígidas lineares.
- Cortar as junções para as bordas remanescentes e bordas rígidas curvas.



- Crie uma dobra na borda rígida restante.

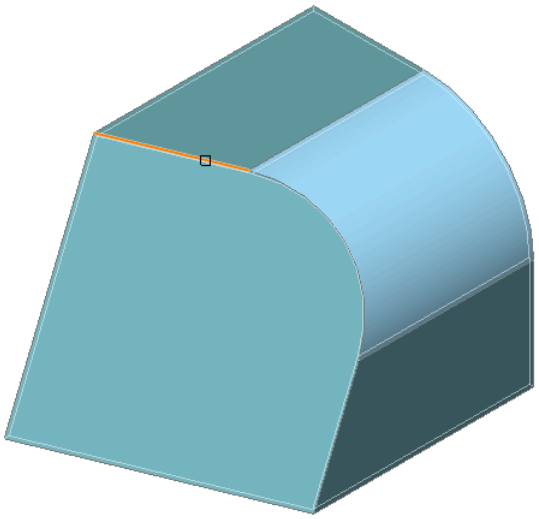
1 Crie alívios de dobra.

Faça um dos seguintes:

- Clique no botão **Criar Alívio** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Alívio** no menu **Sheet Metal**.
- Digitar *SMRELIEFCREATE* na barra de Comando.

Você é solicitado:

Selecione uma borda rígida, face de dobra, face de flange, ou sólido 3D [Modelo inteiro]:



2 Selecione uma borda rígida.

Você é solicitado:

Selecione outra borda rígida ou face de dobra para o alívio do canto:

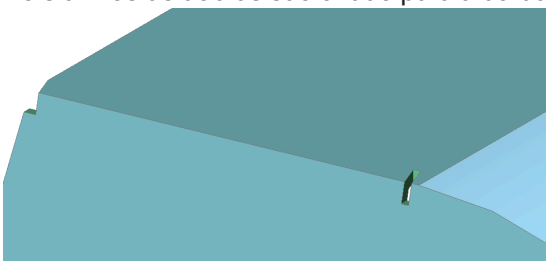
3 Pressione **Enter** ou botão-direito do mouse.

Você é solicitado:

Entre o tamanho do alívio pela relação do raio de dobra ou [forçar alívio de Dobra/Auto] <Auto>:

4 Pressione **Enter** ou clique botão-direito.

Dois alívios de dobras são criado para a borda:



Nós criamos dois alívios de dobra porque um lado da borda rígida com um vértice "livre" precisa de um alívio para fazer uma face de espessura na ortogonal.

Criamos um alívio de dobra no lado adjacente à dobra em loft porque este recurso é diferente de uma dobra regular: criar um alívio de canto ali seria muito grande.

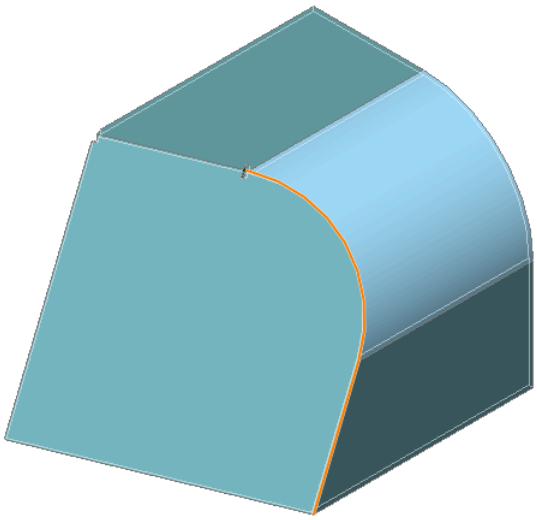
5 Crie 2 junções.

Faça um dos seguintes procedimentos:

- Clique no botão **Criar Junção** () na aba **Sheet Metal** da Faixa de opções;
- Escolha **Criar Junção** no menu **Sheet Metal**;
- Digite *SMJUNCTIONCREATE* na barra de Comando.

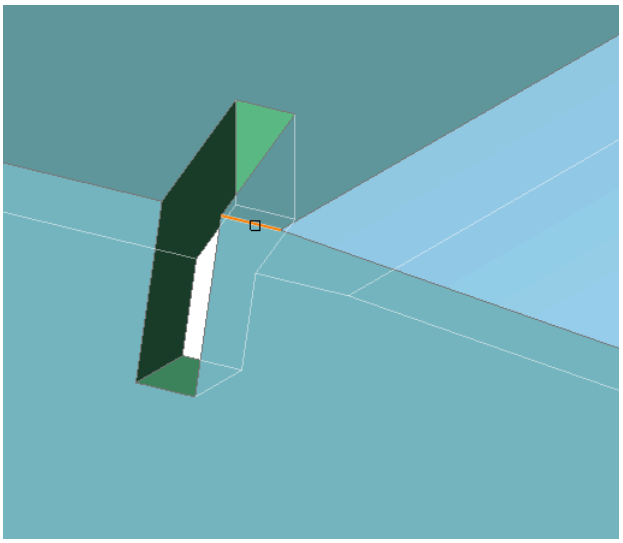
  Você é solicitado:

  Selecione bordas rígidas ou dobras ou flanges ou sólidos 3D [Modelo inteiro] <Modelo inteiro>:



6 Selecione as 3 bordas rígidas a seguir:

Considere a pequena borda rígida:



O corte é criado. As peças que correspondiam à borda linear obtêm recursos de junção regulares.

7 Criar uma dobra na borda rígida. Por ser a única borda rígida que resta no modelo, aplique o comando a todo o modelo.

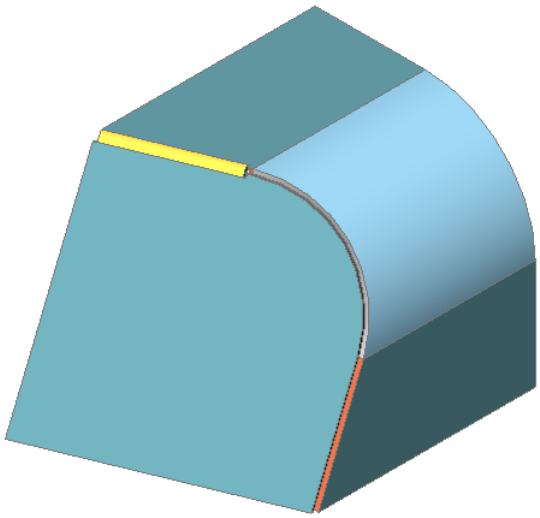
- Clique no botão de ferramenta **Criar Dobra** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Dobra** no menu **Sheet Metal**.
- Digite *SMBENDCREATE* na barra de Comando.

  Você é solicitado:

  Selecione bordas rígidas ou junções ou flanges ou sólidos 3D ou [Modelo inteiro] <Modelo inteiro>:

8 Pressione **Enter**.

A dobra é criada.



9 Desdobrar a peça.

- Clique no botão **Desobrar Corpo** ( ) na aba **Sheet Metal** da Faixa de opções.

- Escolha **Desdobrar Corpo** no menu **Sheet Metal**.
- Digite *SMUNFOLD* na barra de Comando.

  Você é solicitado:

  Selecione uma flange ou face de dobra em loft para começar a desdobrar [Associativo]:

10 Selecione alguma face de flange.

   Você é solicitado:

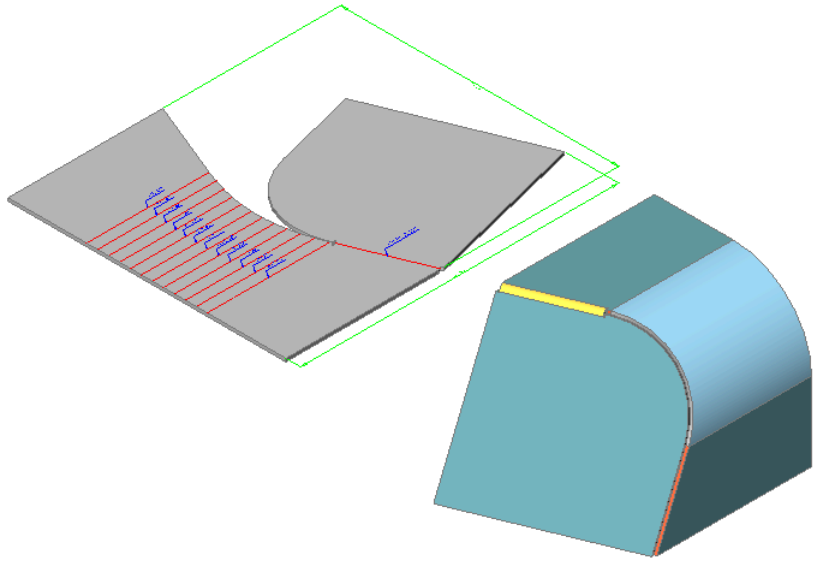
   Selecione a posição do corpo desdobrado:

11 Coloque a peça desdobrada.

   Você é solicitado:

   Valide o corpo desdobrado e selecione uma opção [salvar geometria 2D/salvar geometria 3D/Otimizar anotações de dobra/Manter] <Manter>:

12 Pressione **Enter** para manter o resultado.



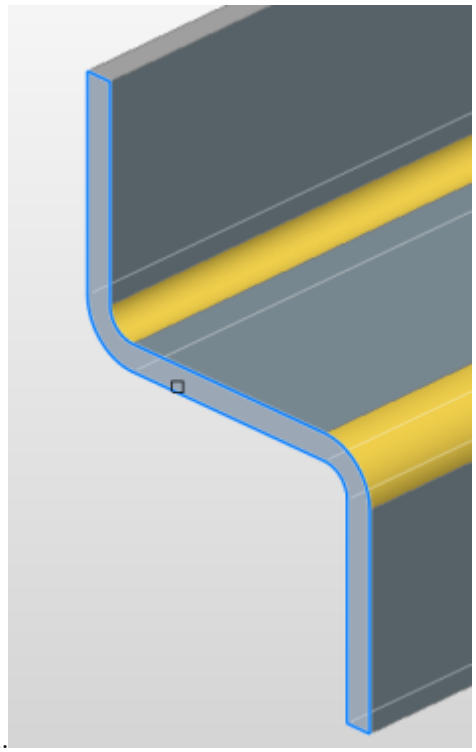
21.5.12.3 Para Dividir uma Face de Espessura

Para estender uma flange que compartilha a mesma face de espessura com outras flanges, a face de espessura precisa ser dividida.

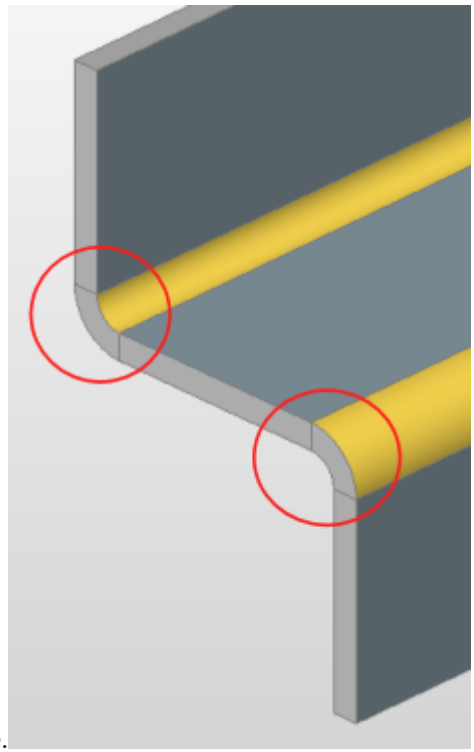
1 Inicie o comando *SMIMPRINT* ( ).

  Solicita a você: Selecione uma face de espessura:

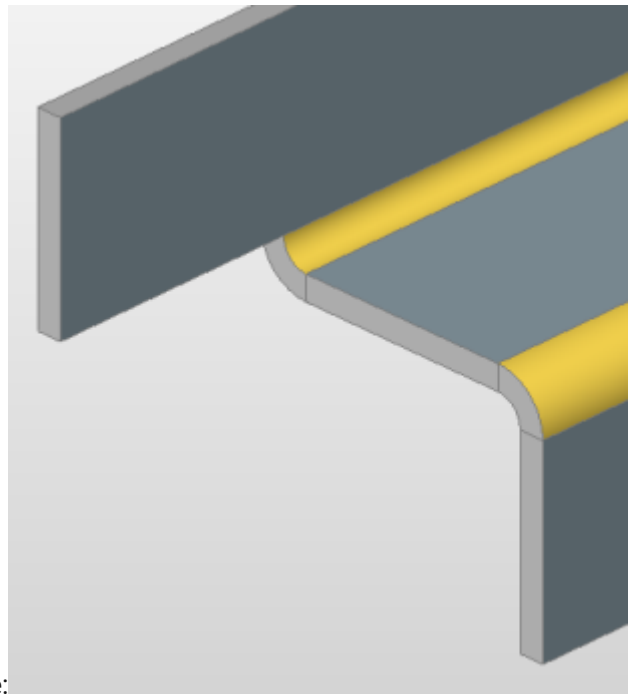
2 Selecione a face de espessura compartilhada.



3 Clique o botão-direito ou pressione **Enter** para dividir a face.



4 As flanges podem agora ser estendidas

separadamente: 

5 Executar o Comando *DMSIMPLIFY* para remover as divisões não necessárias, unificando assim as faces de espessura coplanares.

Tutorial: Dividir faces de espessura em regiões

21.5.12.4 Mudar o Raio de Dobra

Para alterar o raio geral de dobra:

1 Selecionar o nó raiz no **Navegador de Mecânica**.

2 Encontre o grupo **Raio de dobra** na parte inferior do **Navegador de Mecânica**.

3 Siga um dos procedimentos:

- Defina o **Tipo** como **Valor Absoluto**, então digite o valor de raio no campo **Valor**.

  O campo **Raio de Dobra** muda de acordo.

- Defina o **Tipo** para **Razão de espessura**, e digite um valor no campo **Valor**.

  O campo **Raio de Dobra** é calculado como o produto da **Espessura** e **Razão de espessura**.

Para alterar o raio de dobra de uma dobra em especial:

1 Selecionar o nó raiz no **Navegador de Mecânica**. A dobra selecionada se destaca no modelo. As Propriedades de Dobra são exibidas na parte inferior do **Navegador de Mecânica**.

2 Siga um dos procedimentos:

- Defina o **Tipo** como **Valor Global**.

  O raio global de dobra atual é aplicado à dobra selecionada.

- Defina o **Tipo** como **Valor Absoluto**, então digite o valor de raio no campo **Valor**.

O campo **Raio de Dobra** muda de acordo.

- Defina o **Tipo** para **Razão da Espessura** e digite um valor no campo de valor.

  O campo **Raio de Dobra** é calculado como o produto da **Espessura** pela **Razão da espessura**.

21.5.12.5 Alterar a Lacuna da Junção

Para alterar a lacuna geral de junção:

1 Selecionar o nó raiz no **Navegador de Mecânica**.

2 Encontre o grupo **lacuna de junção** na parte inferior do **Navegador de Mecânica**.

3 Siga um dos procedimentos:

- Defina o **Tipo** como **Valor Absoluto** e digite o valor da lacuna no campo **Valor**.

  O campo **Lacuna da junção** vai muda de acordo.

- Defina o **Tipo** para **Razão da espessura**, então digite um valor no campo **Valor**.

  O campo de **Lacuna da junção** é calculado como o produto da **Espessura** pela **Razão da espessura**.

Para alterar a lacuna da junção para uma determinada dobra:

1 Selecionar o nó raiz no **Navegador de Mecânica**. A dobra selecionada se destaca no modelo. As Propriedades de Dobra são exibidas na parte inferior do **Navegador de Mecânica**.

2 Siga um dos procedimentos:

- Defina o **Tipo** para **Valor Global**.

  A atual lacuna da junção global é aplicado à dobra selecionada.

- Defina o **Tipo** como **Valor absoluto**, então digite o valor da lacuna no campo **Valor**.

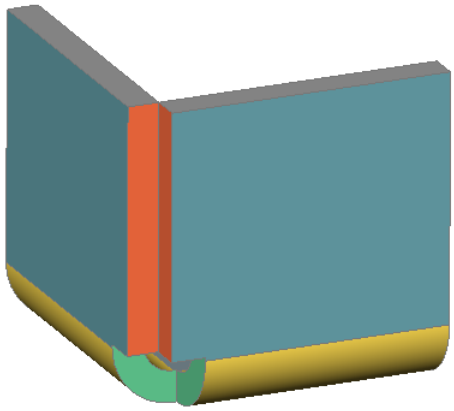
  O campo de **Lacuna da junção** muda de acordo.

- Defina o **Tipo** para **Razão da espessura**, então digite um valor no campo **Valor**.

  O campo de **Lacuna da junção** é calculado como o produto da **Espessura** pela **Razão da espessura**.

21.5.12.6 Alterar o Tipo de Junção

Por padrão, BricsCAD cria junções simétricas como é mostrado na imagem abaixo:

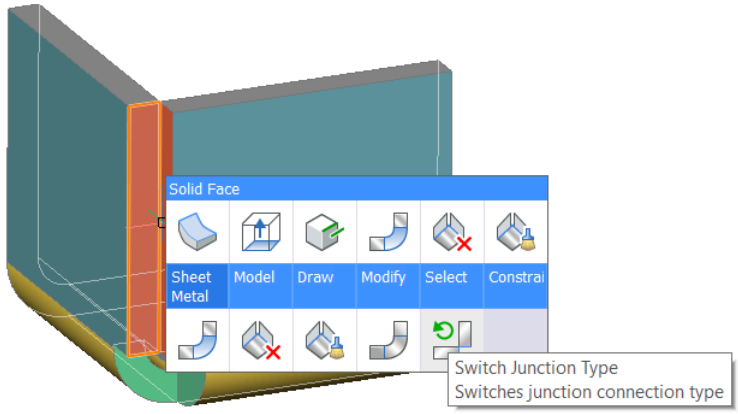


O comando *SMJUNCTIONSWITCH* permite alterar um recurso de junção simétrica para um com faces sobrepostas.

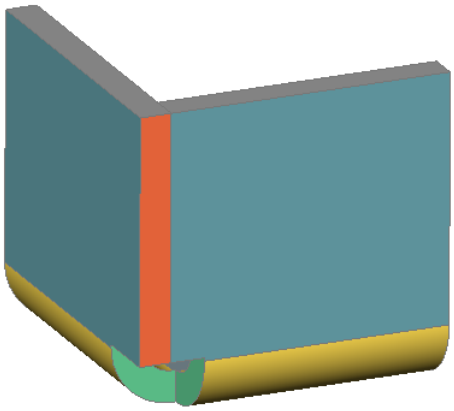
Para alterar o tipo de junção usando o menu Quad:

1 Sobrevôe o cursor sobre uma face de espessura dessa junção.

2 Selecione **Trocar Tipo de Junção** no grupo de comandos **Sheet Metal** no menu **Quad**.



A face de espessura selecionada está alinhada com a face da flange.



Para alterar o tipo de junção no Navegador de Mecânica:

Método 1:

1 Selecione a junção correspondente na árvore.

As propriedades da junção são exibidas na parte inferior do **Navegador de Mecânica**.

2 Selecione a propriedade **Tipo de conexão**.

3 Escolha o tipo de junção desejada.

As opções são: **Simétrica**, **Alinhada à Flange_X** ou **Alinhada à Flange_Y**.

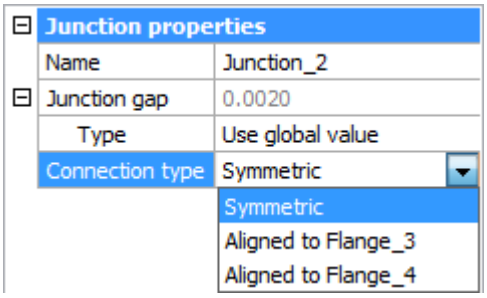


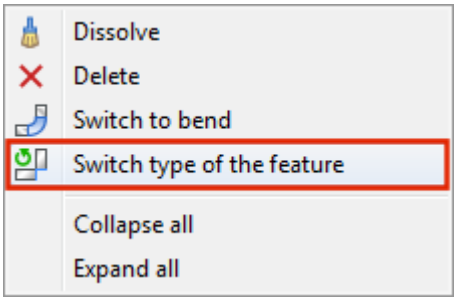
Figura 133: Propriedades da junção

A junção é atualizada em conformidade.

Método 2:

1 Selecione a junção correspondente na árvore.

Um menu de contexto é exibido:



2 Clique o botão-direito do mouse e escolha **Trocar o tipo de recurso** no menu de contexto.

Nota: Ambas junções simétricas e alinhadas mantêm o mesmo valor de lacuna entre flanges conectadas.

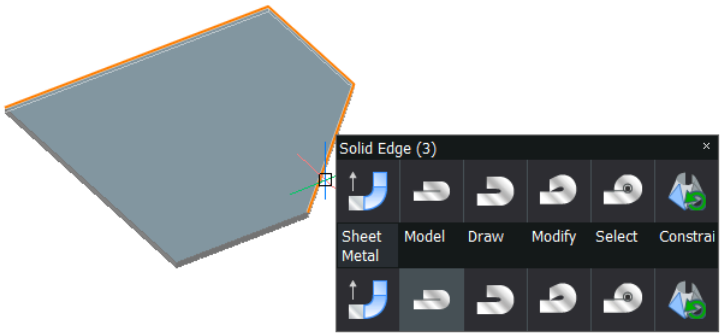
21.5.13 Trabalhar com Bainhas

Um recurso de bainha é produzido quando a borda de uma flange de Sheet Metal é dobrada de volta sobre si mesma. *SMHEMCREATE* simplifica a criação de bainhas de chapa metálica. No BricsCAD bainhas têm seu próprio conjunto de propriedades dependendo do tipo, pelo tipo de geometria "flange" e "dobra". Os recursos de Bainha são reconhecidos pelo comando *SMCONVERT* e têm um símbolo especial quando desdobrado com o comando *SMUNFOLD*.

21.5.13.1 Criar uma Bainha

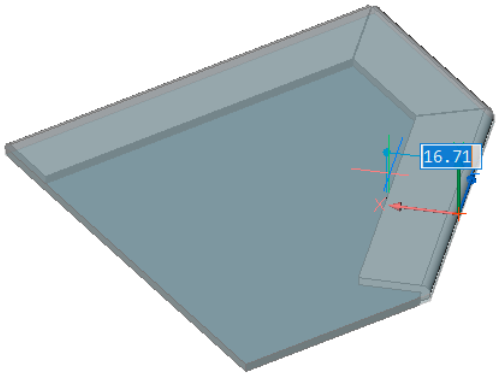
1 Siga um dos procedimentos:
   - Selecione várias bordas lineares na flange.
   - Passe o cursor sobre uma borda linear da flange.

2 Siga um dos procedimentos:
   - Selecione a ferramenta **Criar uma Bainha Fechada** () no menu **Quad**.
   - Clique os botões **Criar uma Bainha Fechada** () na aba **Sheet Metal** da Faixa de opções.
   - Escolha **Criar uma Bainha Fechada** no menu **Sheet Metal**.
   - Digite *SMHEMCREATE* na barra de

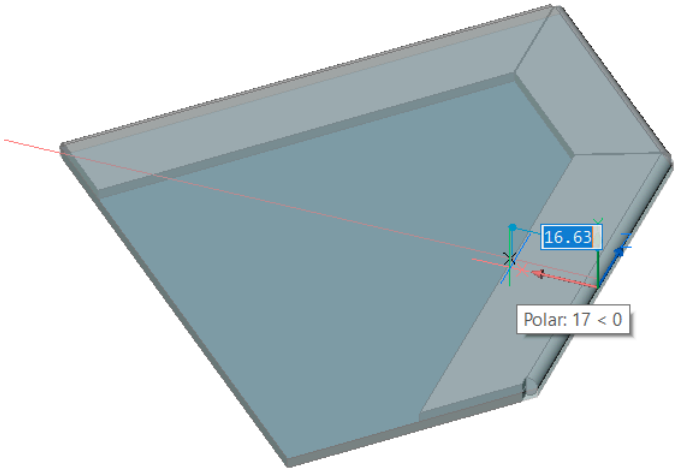


   Comando.

3 Passe o cursor para ajustar o comprimento da bainha:

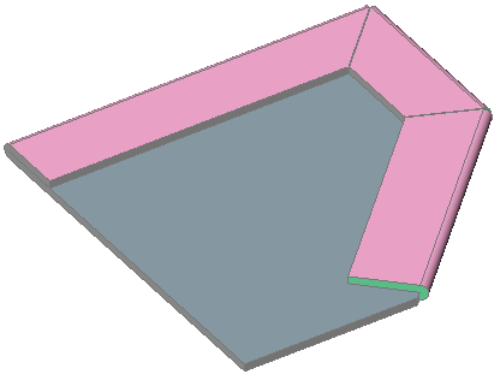


4 Use o assistente **Ctrl** ou a opção **Alternar afunilamento** para inverter o ajuste de uma bainha até a



borda:





5 Quando terminar, pressione **Enter**:

Bainhas e recursos auxiliares (alívios e meia-esquadrias) são criados.

21.5.13.2 Modificar Parâmetros da Bainha

Para modificar parâmetros da Bainha, selecione o recurso e use o **Navegador de Mecânica**. Diferentes parâmetros estão disponíveis, dependendo do tipo da bainha.

Tabela 43: Parâmetros de Bainha

Tipo de bainha	Parâmetros disponíveis
Fechado	Raio Comprim

Tabela 43: Parâmetros de Bainha

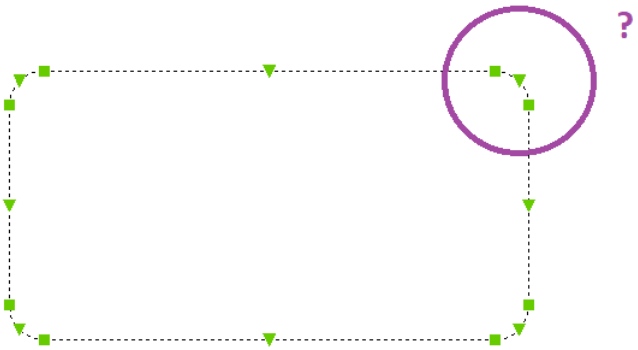
Tipo de bainha	Parâmetros disponíveis
Abrir	Raio Comprim
Lágrima	Raio - altera o ângulo da bainha Comprimento - alterar o ângulo da bainha
Redondo	Raio - altera a parte "planar" da bainha

No caso de qualquer alteração paramétrica, o tipo de bainha é preservado e a geometria é movida para dentro, para manter as dimensões externas.

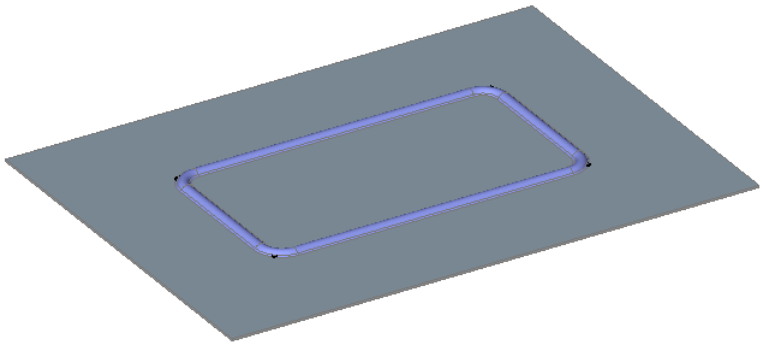
21.5.13.3 Apagar uma Bainha

Quando um recurso de bainha é apagado, o BricsCAD restaura de forma inteligente as dimensões e alívios da geometria.

A peça de flange abaixo tem dimensões externas iguais a 100.

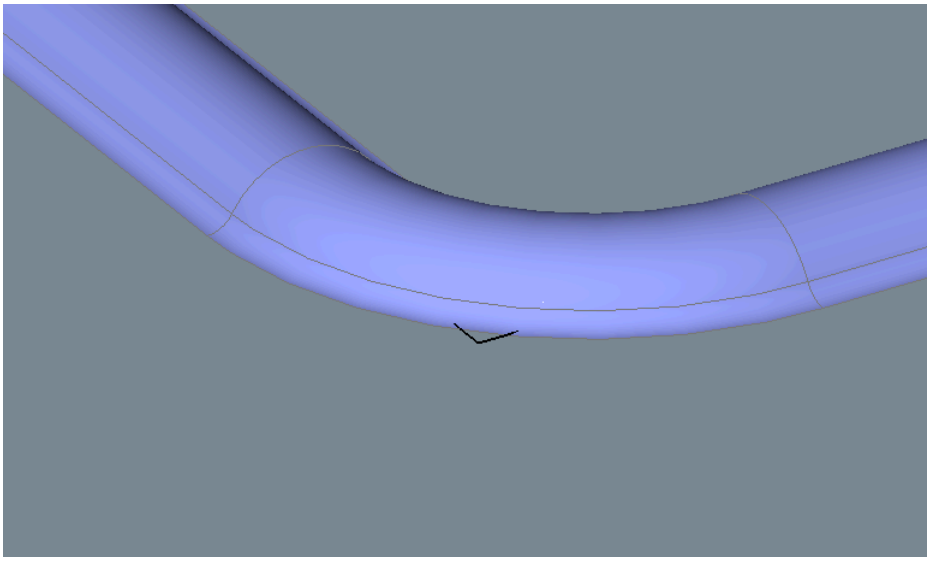


1 Use a opção Largura, do comando *SMHEMCREATE* para criar uma bainha parcial em uma borda

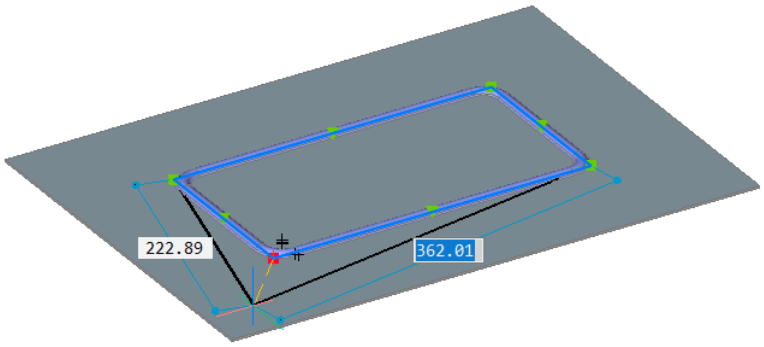


única:

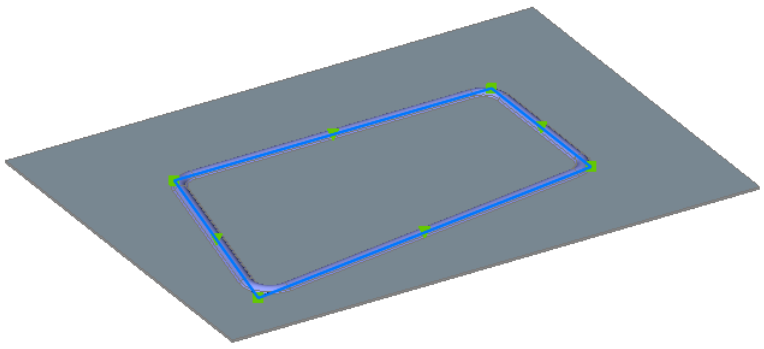
2 Crie bainhas em outras 2 bordas:



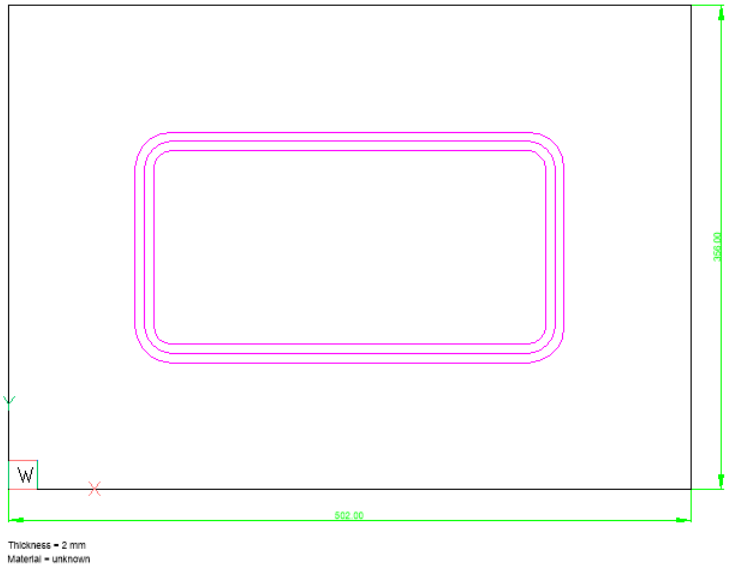
Essas operações levam ao aparecimento de geometria de alívio complexa:



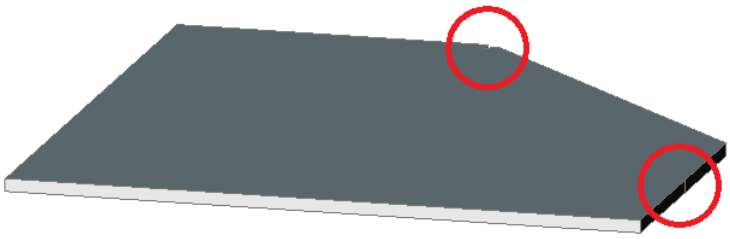
Nota: As dimensões externas da peça são preservadas.



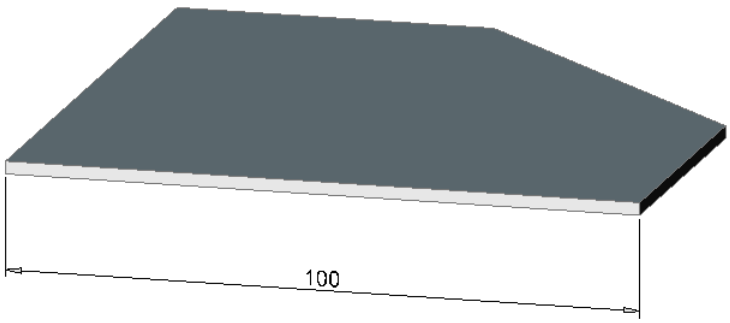
3 Selecione os recursos da bainha e exclua-os.



Algumas imperfeições são deixadas no recurso de alívio do canto e na borda estampada:



4 Exclua o recurso de alívio de canto e use *DMSIMPLIFY* para obter a peça original com as dimensões



originais.

21.5.13.4 Comandos Relacionados

O comando *NAVEGADORMECANICAABRIR* exibe o Navegador de Mecânica.

O comando *SMDELETE* restaura uma borda rígida entre 2 flanges, para remover uma dobra ou junção - remove uma flange e todas as dobras adjacentes. As flanges adjacentes são estendidas até uma configuração de junção, com a flange sendo excluída. Restaura a geometria removida pela mitra.

O comando *SMHEMCREATE* cria uma ou mais bainhas para uma peça de sheet metal, puxando uma ou mais bordas de uma flange existente.

21.5.14 Trabalhar com Abas

Os recursos de abas são usados para criar conexões fáceis e robustas entre peças de sheet metal.

Os recursos de aba são criados a partir de junções.

21.5.14.1 Criar uma Aba única

Uma única aba é criada por padrão pelo comando *SMTABCREATE*. Esse recurso só tem um único slot.

1 Siga um dos procedimentos:

- Passe o mouse sobre uma face de junção e clique na ferramenta **Criar Aba única** ( ) no menu **Quad**.
- Clique o botão **Criar Aba única** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Aba única** no menu **Sheet Metal**.
- Digite *SMTABCREATE* na barra de Comando.

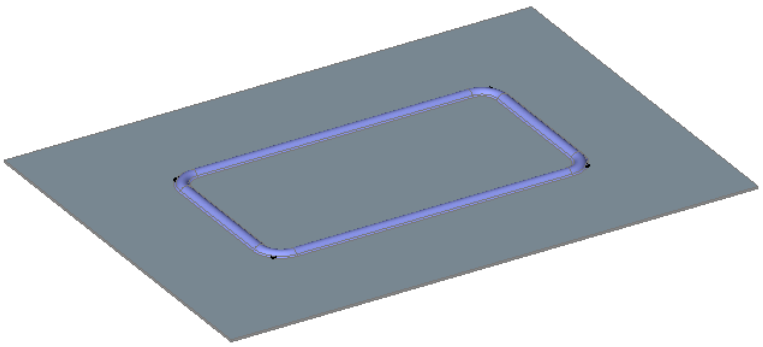
  Você é solicitado: Selecione a junção para criar uma aba única ou [Matriz]:

2 Selecione uma face de junção.

  Você é solicitado: insira o comprimento da aba ou [Ponto central]:

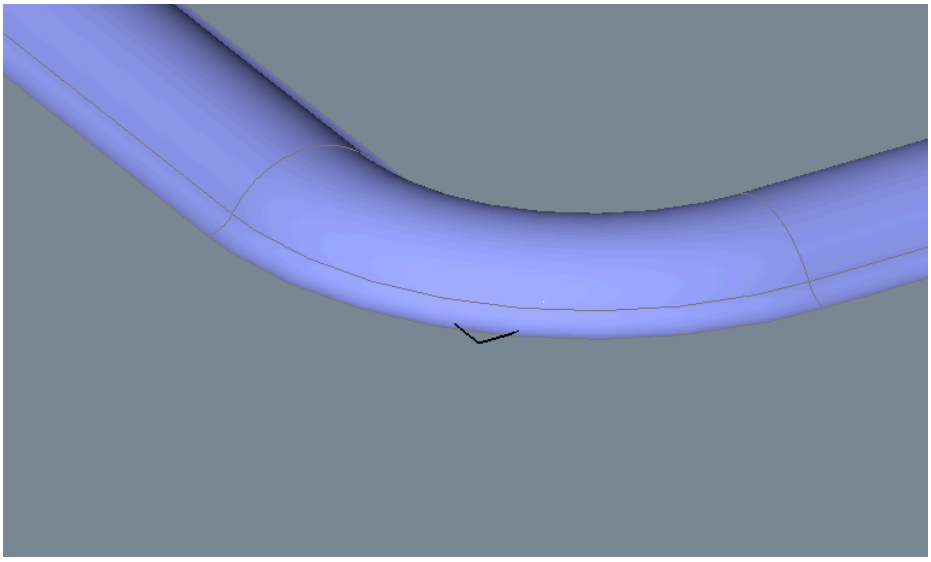
3 Define o comprimento da aba.

  Uma aba é colocada simetricamente em relação a um ponto central. Digite valores nos campos de entrada dinâmica de comprimento, para definir o comprimento de uma aba.



4 Selecione a opção **Ponto central** para posicionar a aba.

  Você é solicitado: Selecione o ponto central ou [Comprimento]:

2 Dimensões dinâmicas controladas por tabulação são usadas para criar uma guia relativa a uma junção: 

5 Selecione a opção **Comprimento** para especificar o comprimento, ou pressione **Enter** para concluir o comando.

21.5.14.2 Criando uma matriz de Abas

Um recurso de matriz de abas possui várias ranhuras, em contraste com Abas individuais.

1 Para criar matriz de abas, siga um destes procedimentos:

- Passe o mouse sobre uma face de junção, e clique na ferramenta **Criar matriz de Abas** ( ) no menu **Quad**.
- Clique no botão **Criar matriz de Abas** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar matriz de Abas** no menu **Sheet Metal**.
- Digite *SMTABCREATE* na barra de Comando.

  Você é solicitado: Selecione a junção para criar uma aba única ou [Matriz]:

2 Selecione a opção **Matriz**.

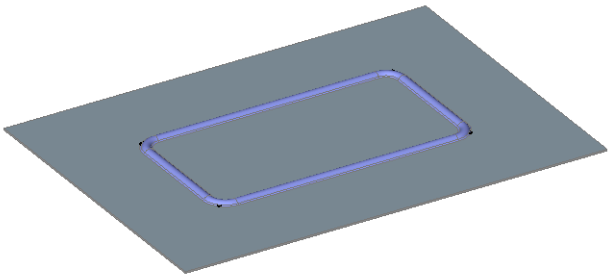
Você é solicitado: Selecione junção ou borda rígida curva, para criar uma aba de matriz ou [Aba única/ Opções de seleção (?)]:

3 Selecione uma face de junção.

Você é solicitado: Aceite a configuração da aba ou [Inverter/Número da ranhura/Distância/ Comprimento/Raio redondo/Distância do chanfro/Aceitar]<Aceitar>:

4 Você pode usar opções para modificar a aparência de uma aba de matriz (veja o exemplo abaixo).

- **Número de ranhuras = 3** - observe que a imagem não é simétrica. Se você clicar na opção **Inverter**, 3 ranhuras vão aparecer no outro lado da junção.
- **Comprimento = 8** - o comprimento da ranhura.
- **Distância = 40** - a dimensão entre os centros das

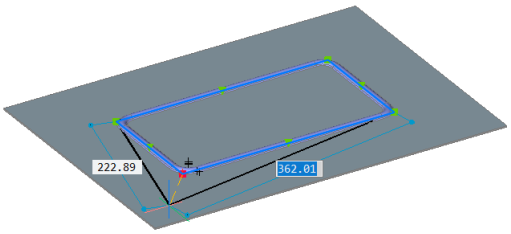
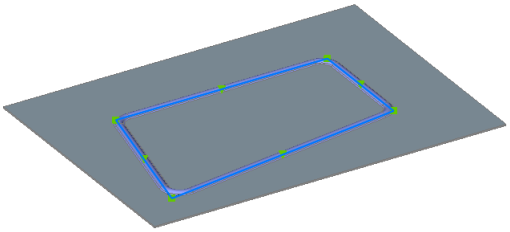


ranhuras.

5 Altere o **Raio redondo** e **Distância do chanfro** para definir concordâncias e chanfros para as bordas.

Dica: Se você definir por engano concordâncias ou chanfros, use valores de 'zero' para desativá-los.

6 Depois de definir todas as opções, escolha **Aceitar** ou pressione **Enter** para criar a aba e terminar o comando.

Concordar	Chanfros
	

21.5.14.3 Criar uma Matriz de Abas em Curva

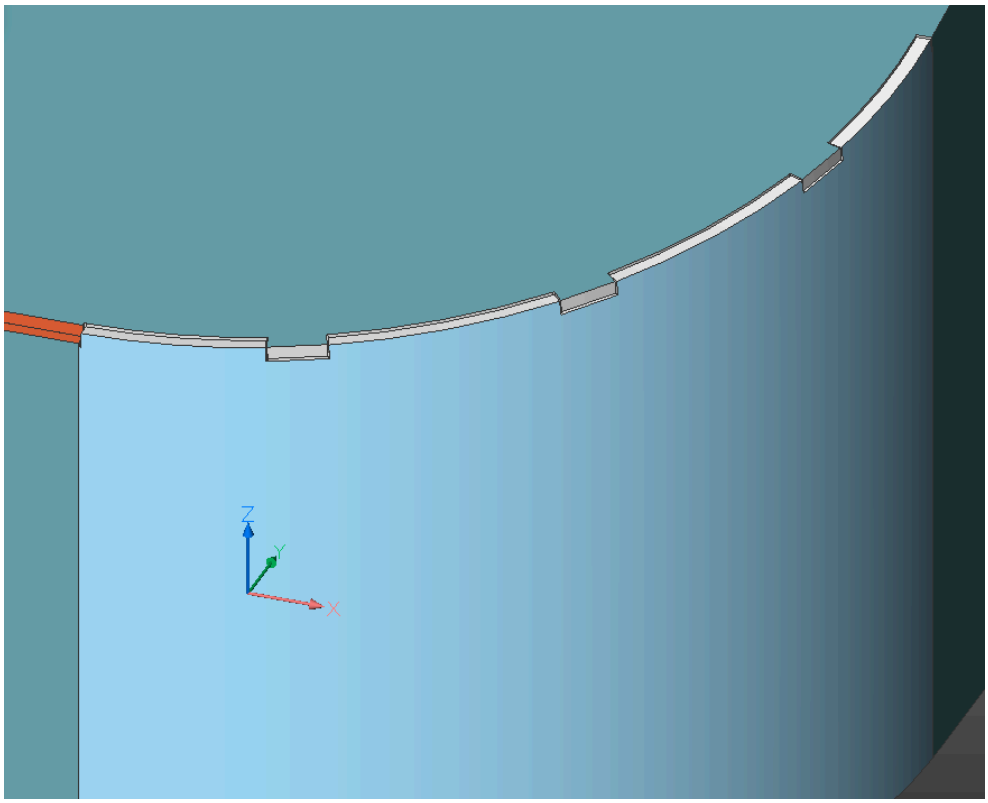
Este caso é semelhante à criação da Matriz de Abas reta, mas as abas são colocadas ao longo de um caminho curvo.

1 Para criar uma matriz de abas em curva, siga um destes procedimentos:

- Passe o mouse sobre uma borda rígida curvada e clique na ferramenta **Criar matriz de Abas** () no menu **Quad**.
- Clique no botão **Criar matriz de Abas** () na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar matriz de Abas** no menu **Sheet Metal**.
- Digite *SMTABCREATE* na barra de Comando.

  Você é solicitado: Selecione junção ou borda rígida curva para criar uma aba de matriz ou [Aba única/Opções de seleção (?)]:

2 Selecione uma borda rígida curva.

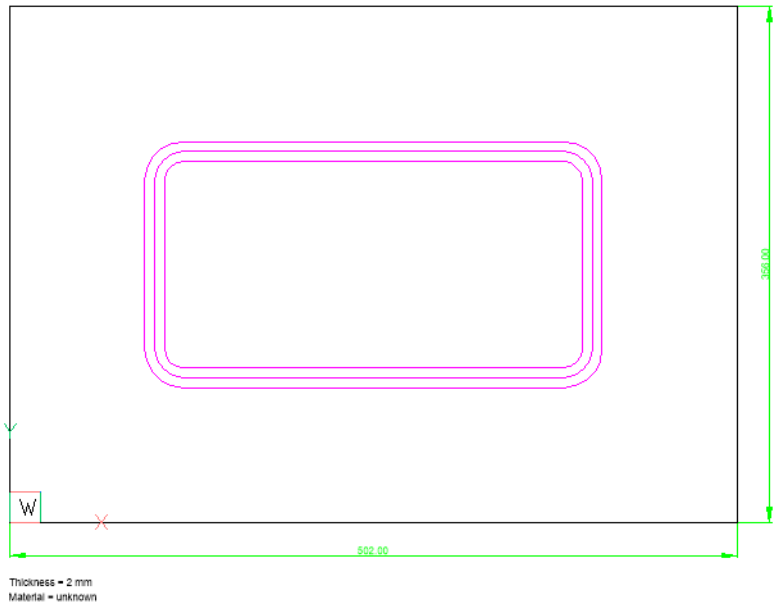


21.5.14.4 Modificar Abas

As abas, única ou em matriz, têm diferentes propriedades editáveis, que estão disponíveis no **Navegador de Mecânica**.

Clique no nome do recurso de aba para editar.

Uma aba única tem parâmetros de **Comprimento**, **Folga**, **Raio de concordância**, e **Distância do chanfro**. A **Folga da aba** define a lacuna entre a aba e as faces da ranhura.



Uma matriz de abas tem os parâmetros **Número de ranhuras**, **Distância**, **Comprimento**, **Folga**, **Raio de concordância**, e **Distância do chanfro**.

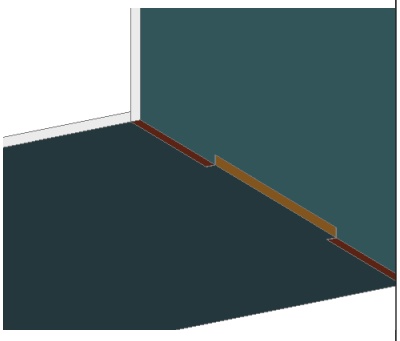
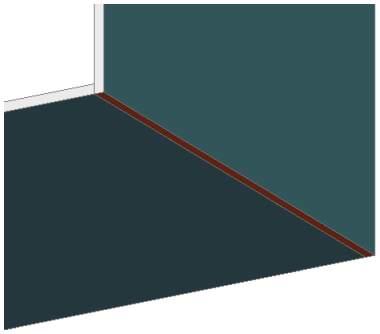
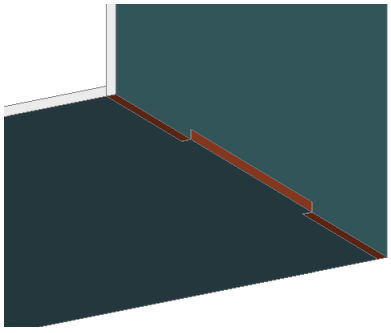
Nota: As Abas em Matriz em Curva não podem ser modificadas.

21.5.14.5 Excluir Abas

Use o comando *SMDELETE* para excluir abas (não aplicável para curvas). Um recurso de junção é criado no lugar das abas.

21.5.14.6 Explodir Abas

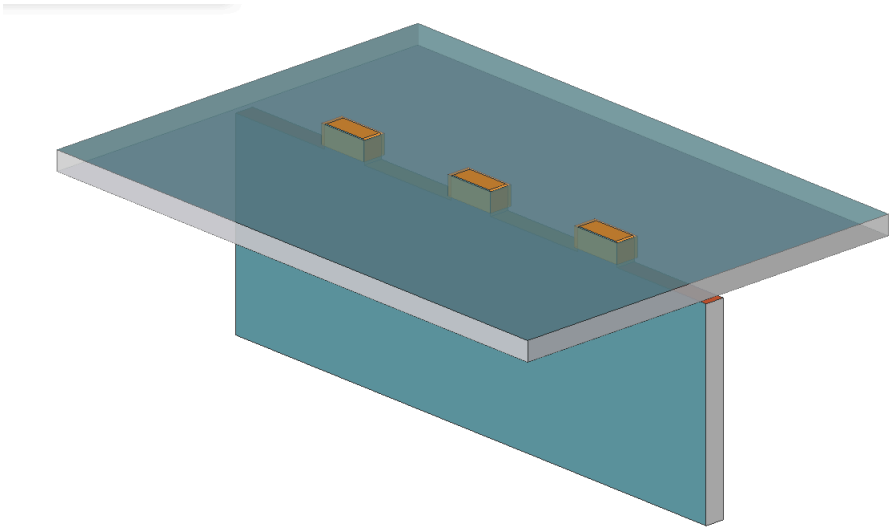
Use o comando *SMEXPLODE* para explodir uma junção com abas para um conjunto de recursos de abas. A geometria da aba permanece a mesma e suas faces são distribuídas entre vários recursos de junção.

Recurso original da aba	Resultado após excluir	Resultado após explodir
		

21.5.15 Trabalhar com Aba-T e Junção-T

21.5.15.1 Visão Geral

Você pode usar os recursos Aba-T para criar uma conexão com posicionamento robusto entre 2 flanges de sheet metal. Ao contrário das abas normais, a Aba-T forma uma configuração semelhante a um T, quando uma flange é conectada à outra.



Aba-T pode ser criada de Junção-T pelo comando *SMTabCreate*. Recursos de Junção-T são frequentemente usados como um estágio intermediário para criar uma Aba-T.

21.5.15.2 Criar uma Junção-T

Uma Junção-T pode ser criada em flanges separadas existentes, da seguinte maneira.

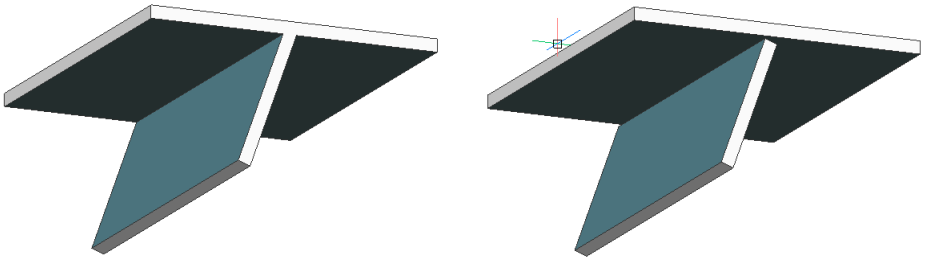
1 Selecione uma face de espessura na flange.

2 Siga um dos procedimentos:

- Clique no botão **Conectar Flanges** () na aba **Sheet Metal | Junção** na Faixa de opções.
- Escolha **Conectar Flanges** no menu **Sheet Metal | Junção**.
- Digite *SMFLANGECONNECT* na barra de Comando.

3 Pressione **Enter**.

Como alternativa, você pode criar um recurso de junção T no lugar de uma borda rígida semelhante a T entre 2 flanges (imagem à esquerda):



1 Passe o mouse sobre uma borda rígida entre as flanges e clique na ferramenta **Criar Junção** ( ) no menu **Quad**, ou siga um destes procedimentos:
   - Clique no botão **Criar Junção** ( ) na aba **Sheet Metal** da Faixa de opções.
   - Escolha **Criar Junção** no menu **Sheet Metal | Junção**.
   - Digite *SMJUNCTIONCREATE* na barra de Comando.

2 Selecione uma borda rígida.

21.5.15.3 Criar uma Aba-T Isolada

O comando *SMTABCREATE* cria uma única aba-T por padrão. Este recurso tem apenas uma única aba.

Para criar uma única aba-T:

1 Passe o mouse sobre uma face de junção T e clique na ferramenta **Criar Aba única** ( ) no menu **Quad** ou execute um dos seguintes procedimentos:
   - Clique o botão **Criar Aba única** ( ) na aba **Sheet Metal** da Faixa de opções.
   - Escolha **Criar Aba única** no menu **Sheet Metal | Aba**
   - Digite *SMTABCREATE* na barra de Comando.

2 Selecione uma face de junção-T.

21.5.15.4 Criar uma Aba-T em Matriz

Como diferente de abas-T isoladas, os recursos de Aba-T em matriz têm múltiplas ranhuras (slots).

Para criar uma matriz de aba-T:

1 Passe o mouse sobre uma face de junção-T e clique na ferramenta **Criar matriz de Aba** ( ) no menu **Quad** ou execute um destes procedimentos:
   - Clique no botão **Criar matriz de Abas** ( ) na aba **Sheet Metal** da Faixa de opções.
   - Escolha **Criar matriz de Abas** no menu **Sheet Metal | Aba**.
   - Digite *SMTABCREATE* na barra de Comando.

     Você é solicitado: Selecione a junção para criar uma aba única ou [Matriz]: selecione a opção **Matriz**.

2 Selecione uma face de junção-T.

21.5.15.5 Modificar Parâmetros de Junção-T e Aba-T

Uma junção-T tem apenas o parâmetro de **Lacuna da junção**.

Matriz de abas-T e abas-T isoladas têm as seguintes propriedades: **Distância**, **Comprimento**, **Folga**, **Raio de concord**, e **Distância do chanfro**. A Aba-T de matriz também têm o parâmetro **Número de ranhuras**.

Você pode definir o valor de todos os parâmetros no painel **Navegador de Mecânica**.

21.5.15.6 Excluir uma Junção-T ou Aba-T

Quando um recurso Aba-T é excluído, o BricsCAD restaura um recurso de Junção-T com a geometria correspondente. Apagar uma Junção-T junta as flanges, e cria uma borda rígida semelhante a um T entre estes.

21.5.15.7 Comandos Relacionados

O comando *NAVEGADORMECANICAABRIR* exibe o Navegador de Mecânica.

O comando *SMDELETE* apaga um recurso.

21.5.16 Trabalhar com Biséis

21.5.16.1 Visão Geral

Os recursos de Bisel representam uma operação de chanfro feita em peças de Sheet Metal. No nível do BricsCAD V21, cada Bisel deve estar exatamente em uma flange isolada. Não há operações específicas para criar um Bisel, portanto, o comando *DMCHAMFER* deve ser usado numa borda de uma flange, então usar o comando *DMROTATE* para ajustar o ângulo do Bisel. O comando *SMCONVERT* reconhece os biséis e é processado pelo comando *SMUNFOLD*. Os recursos de Bisel também são suportados pelos comandos *SMDELETE* e *SMREPAIR*.

Este procedimento se aplica a dois cenários:

- Trabalhar com geometria de importação, incluindo exportação em lote, pelo comando *SMASSEMBLYEXPORT*.
- Modificar as peças.

Nós distinguimos entre os seguintes tipos de bisel:

Tabela 44: Tipo de Bisel

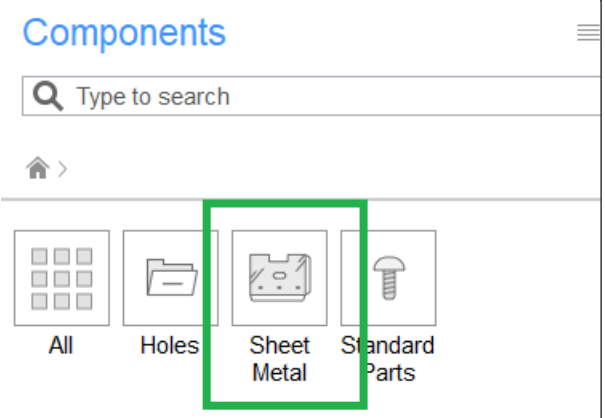
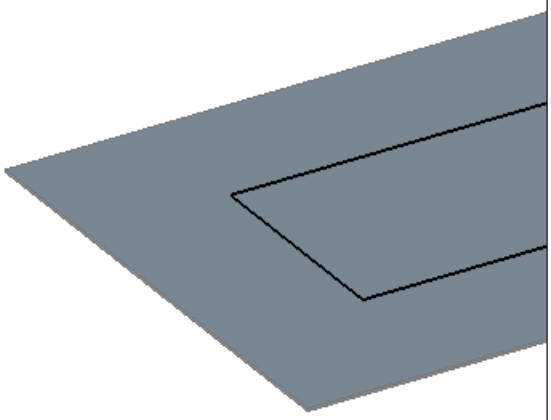
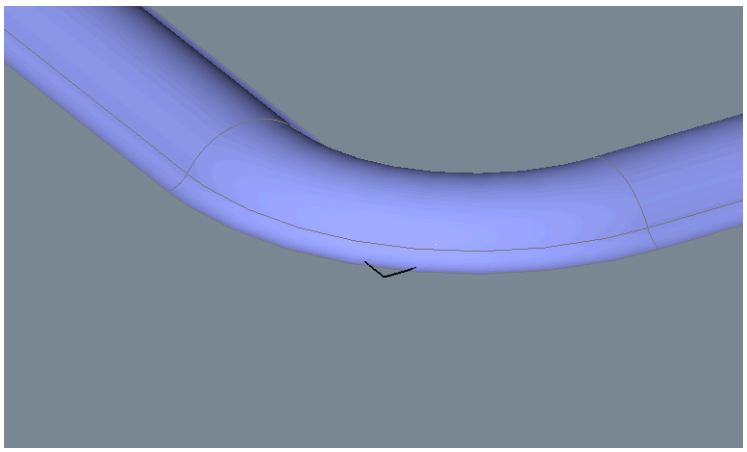
	Bisel de um lado
	Land bisel
	K-bisel

Tabela 44: Tipo de Bisel

	X-bisel

No **Navegador de Mecânica**, as propriedades de cada chanfro de um recurso de Bisel ficam disponíveis em separado. Estas propriedades são:



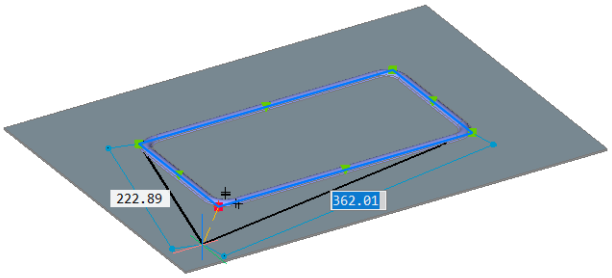
Altura (H): a profundidade da operação de chanfro, assim como o comando *DMCHAMFER* faz.

Angulo (A): a inclinação da face chanfrada.

Nota: Nas propriedades do Navegador de Mecânica, os termos: Superior ou Inferior não são usados para descrever os recursos de Bisel.

21.5.16.2 Reconhecendo os biséis

Vamos começar com uma peça de folha metálica com geometria de Bisel (bordas de flange chanfradas).

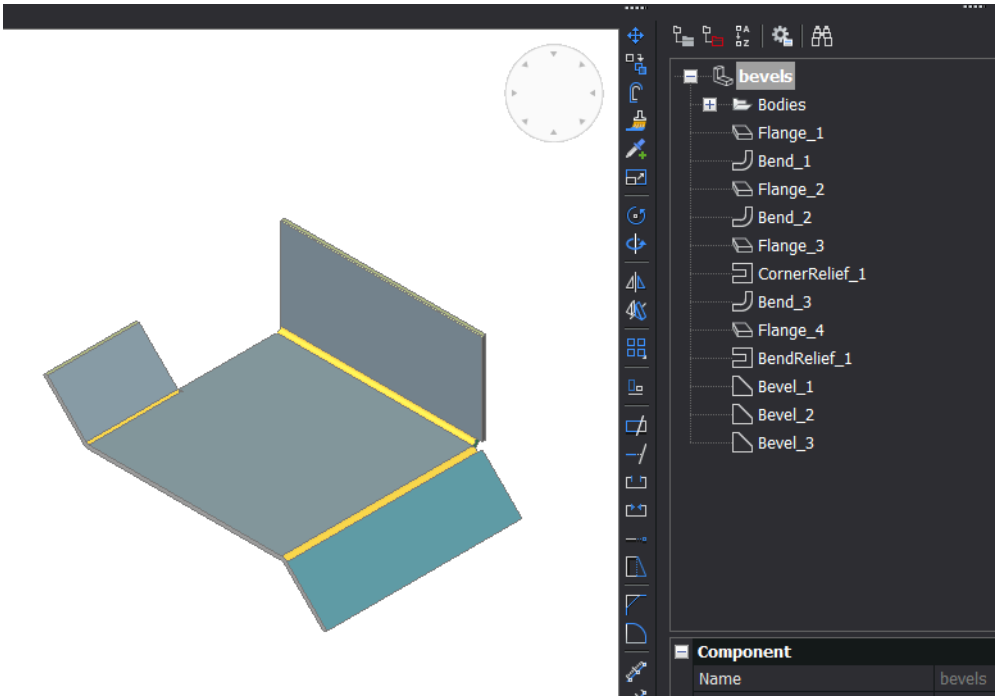


1 Clique no botão **Converter em Sheet Metal** ( ) no painel **Sheet Metal/Criar** Ribbon.

Você é solicitado: Selecione sólidos 3D ou faces iniciai (seed) ou [Modelo Inteiro/Opções de seleção (?)] <Modelo inteiro>:

2 Pressione **Enter** para converter o sólido 3D em uma peça de sheet metal.

Os recursos de bisel são reconhecidos e aparecem no **Navegador de Mecânica** e mostram na peça:



21.5.16.3 Excluir e Reparar

A seguinte operação de ida-e-volta não altera as dimensões gerais da peça: Crie a geometria e o recurso Bisel, usando os comandos *DMCHAMFER*/*SMCONVERT*, então exclua o Bisel usando comando *SMDELETE* e reinicie o procedimento.

Ao usar *SMDELETE* para excluir um Bisel, o volume é **sempre adicionado** à peça, porque as dimensões da peça não podem ser reduzidas.

O suporte no comando *SMREPAIR* também é importante, já que esse comando faz parte do fluxo de trabalho Importar e Retrabalhar.

Nota: Se você aplicar o comando *SMREPAIR* a uma peça com chanfros, mas sem recursos de Bisel, os chanfros serão perdidos e as dimensões serão alteradas, dependendo do lado selecionado da peça.

21.5.16.4 Desdobrar

Os recursos do forma são desdobrados de acordo com a configuração **Modo de Desdobramento** no **Navegador de Mecânica**.

As opções são:

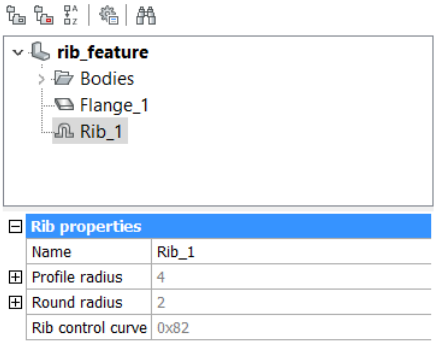
Manter: Não faz nada com geometria do Bisel, resultando em um sólido 3D com chanfros. Essas faces chanfradas provavelmente produzirão artefatos no formato *.dxf 2D.

Remover: mesmo resultado que aplicar *SMDELETE* em todos os recursos de Bisel: todos os talhes estão preenchidos.

Anotar: cada recurso de Bisel é removido, e a borda é anotada. Este modo é recomendado.

Este é o resultado para a peça de exemplo:

- Cada segmento de bisel foi anotado com um texto rosa na camada **Anotações Bisel**.
- UP/ACIMA e DOWN/ABAIXO são no que diz respeito ao lado selecionado para desdobramento. Quando escolher o lado oposto, todas as anotações UP/ACIMA tornam-se DOWN/ABAIXO e vice-versa.
- Para os biséis **Um-lado** e **Land** apenas é necessária uma única operação de chanfro: uma única anotação UP ou DOWN é criada.
- Para **Bisel-K** e **Bisel-X** são necessárias duas operações de chanfro: anotações UP/ACIMA e DOWN/ABAIXO são criadas.



21.5.17 Criar e Dividir Dobras em Loft

BricsCAD permite que você crie ou reconheça peças de sheet metal em loft com geometria curvilínea, que pode ser fabricada usando dobragem múltipla.

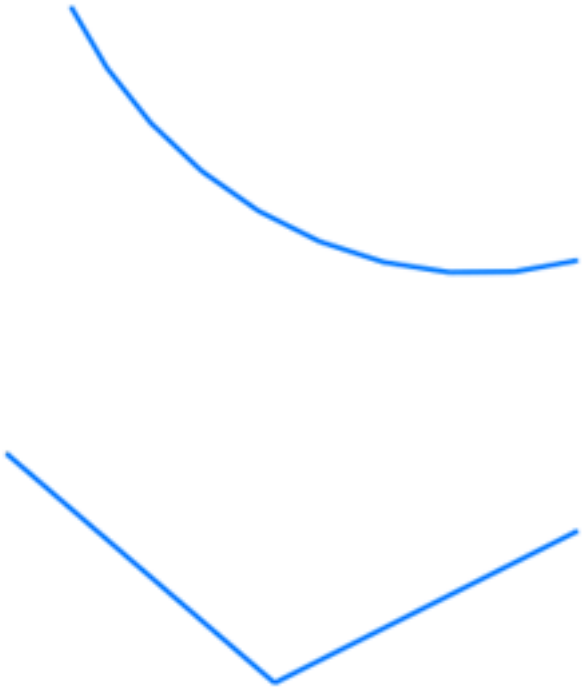
21.5.17.1 Procedimento Geral para Criar uma Peça de Sheet Metal com uma Dobra em Loft

1 Inicie o comando *SMLOFT*.

2 Siga um dos procedimentos:

- Clique o botão **Criar Dobra em Loft** (🔨) na barra **Sheet Metal**.
- Clique no botão **Criar Dobra em Loft** (🔨) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Loft em Sheet Metal** no menu **Sheet Metal**.

  Você é solicitado: Selecione duas seções transversais:



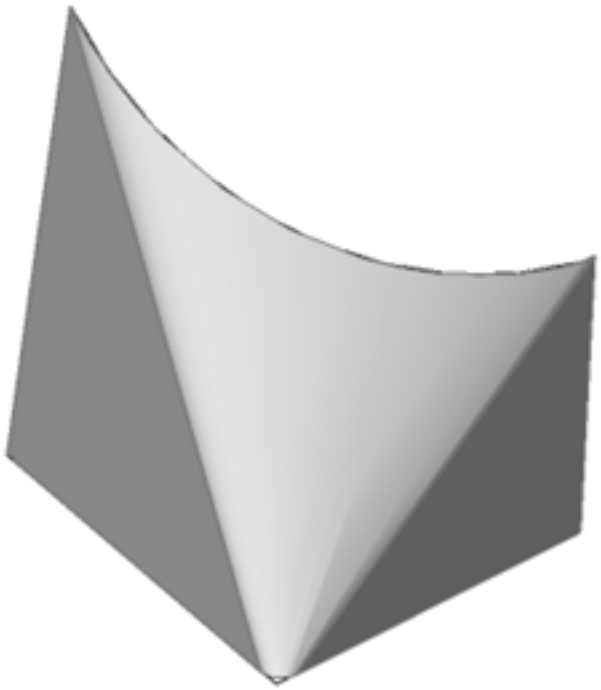
3 Selecione as duas entidades de perfil.

  Você será solicitado: Defina Raio de concordância/Espessura/Dobra simples/<Flanges com dobras>:

4 Pressione **Enter** para aceitar as configurações padrão.

  Uma prévia da peça de chapa metálica é criada.

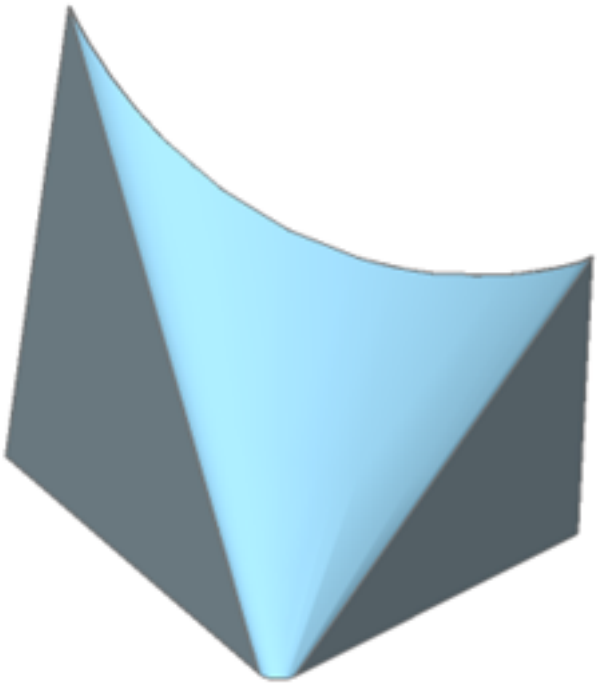
  Você é solicitado: Defina Raio de concordância/Espessura/Dobra única/Flanges com dobras ou engrossar Ambos os lados/Interno/<Externo>:





5 Pressione a tecla **CTRL** para escolher a posição da espessura da peça de sheet metal em relação aos perfis de origem: **Esquerda**, **Centro** ou **Direita**.

Se **DICAS** estiver Ligado, o assistente de dicas é exibido.



6 Pressione **Enter** para criar a peça se sheet metal.

Se **FEATURECOLORS** estiver Ligado, você poderá distinguir entre as 2 flanges ( ) e a dobra em loft ( ).

As propriedades da dobra em loft aparecem no **Navegador de Mecânica**:

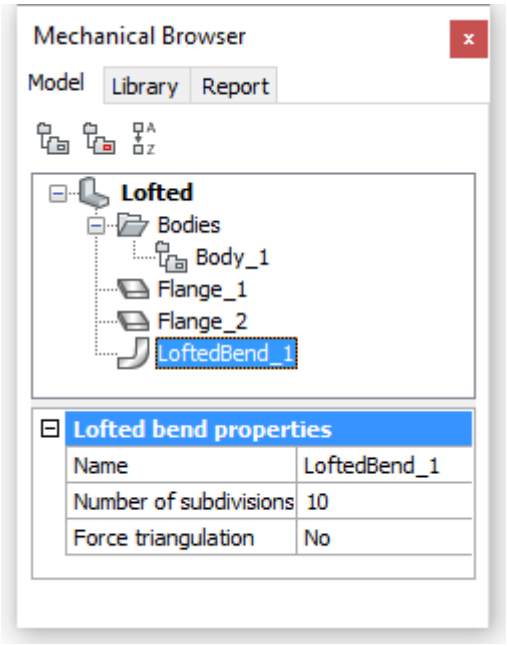


Figura 134: Propriedades da dobra em loft

Opcionalmente, selecione o recurso de dobra em loft para editar suas propriedades:

- **Número de subdivisões**: o número de regiões planas na representação desdobrada de um recurso de dobra em loft (quanto maior o número, mais linhas de dobra serão adicionadas para a visualização planificada para a dobra em loft).
- **Forçar triangulação**: a maioria das dobras em loft pode ser desdobrada usando um algoritmo inteligente de tesselação quadrática. No entanto, às vezes, pode produzir resultados não-ótimos. Nesses casos, configure essa propriedade para *Sim* para alternar para um algoritmo mais direto, que cria quadráticos similares a triângulos.

Tutorial: Dobras em loft no BricsCAD Sheet Metal

21.5.17.2 Para Dividir uma Dobra em Loft

Dobras em loft podem se dividir ao longo de uma linha na superfície em loft, em um ponto arbitrário:

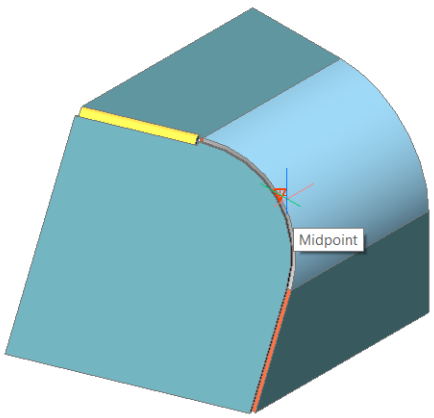
1 Siga um dos procedimentos:
   - Clique na ferramenta **Dividir** () na aba **Sheet Metal** da Faixa de opções.
   - Escolha **Dividir** no menu **Sheet Metal**.
   - Digite **SMSPLIT** na barra de Comando.

   Você é solicitado: Selecione a flange ou a face da dobra em loft:

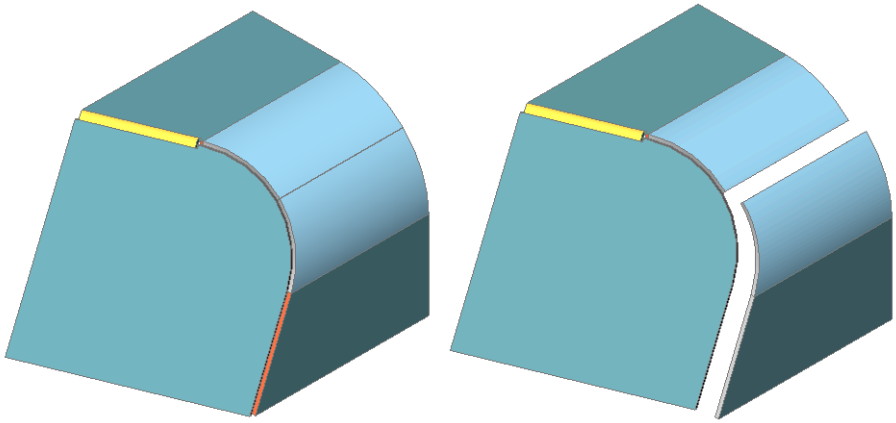
2 Selecionar uma face de dobra em loft.

   Você é solicitado: Selecione o ponto na face da dobra em loft:

3 Selecione um ponto na superfície da dobra em loft.



A dobra em loft é dividida. Se a junção de conexão for dissolvida, você poderá mover livremente a peça separada.



21.5.18 Operacoes dos Recursos: Alivio de Dobra e Alivio de Canto

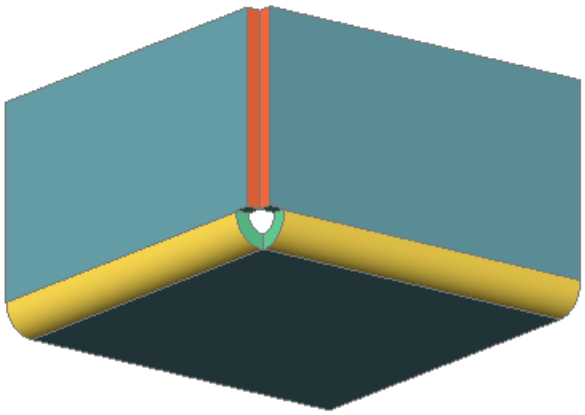
21.5.18.1 Visão Geral

Usar os comandos *SMFLANGEEDGE*, *SMRELIEFCREATE*, *SMJUNCTIONSWITCH*, e *SMRELIEFSWITCH*.

Alívios em peças de Sheet Metal são necessários para evitar que o material enrugue quando dobrado em operações de prensa. Basicamente, você não deve se preocupar com alívios, porque o BricsCAD cria automaticamente todos os alívios necessários quando você cria uma dobra (implícita ou explicitamente). No entanto, se você retrabalhar peças metálicas de baixo grau de precisão, pode ser útil criar alívios de canto e de dobra com as ferramentas dedicadas aqui disponíveis.

21.5.18.2 Alívios de Canto e Junções

Quando você cria uma flange usando o comando *SMFLANGEEDGE*, adjacente a uma borda em dobra, um alívio de canto é criado automaticamente no canto onde as 3 flanges se encontram. Além disso, uma junção é criada entre as 2 flanges que não estão conectadas com uma dobra.



Tutorial: Projetar a partir do zero no BricsCAD Sheet Metal

21.5.18.3 Criar Alívios Faltantes

1 Siga um dos procedimentos:

- Clique no botão **Criar Alívio** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Criar Alívio** no menu **Sheet Metal**.
- Digite *SMRELIEFCREATE* na barra de Comando.

  Você é solicitado: Selecione uma borda rígida, face de dobra, face de flange, ou sólido 3D [Modelo inteiro]:

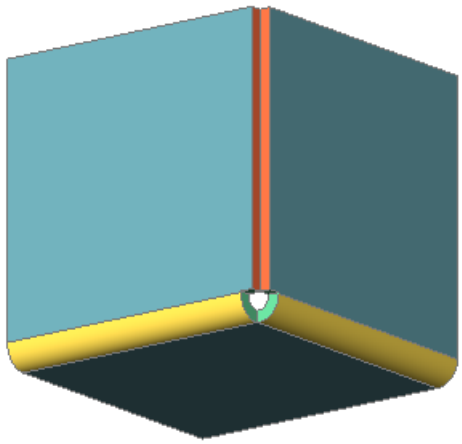
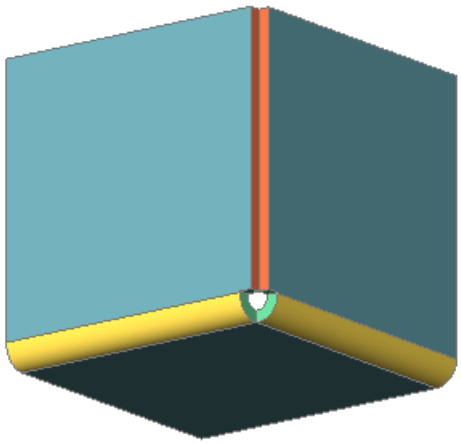
2 Selecione um item.

Entre o tamanho do alívio pela relação do raio de dobra, ou [forçar alívios de Dobra/Auto]< Auto>:

3 Siga um dos procedimentos:

- Pressione **Enter** ou clique o botão-direito para aceitar a razão de dobra padrão do raio.
- Digite um valor no prompt de Comando e pressione **Enter**.

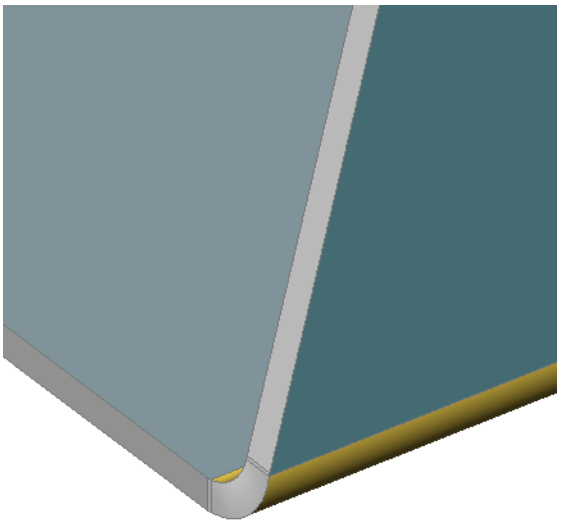
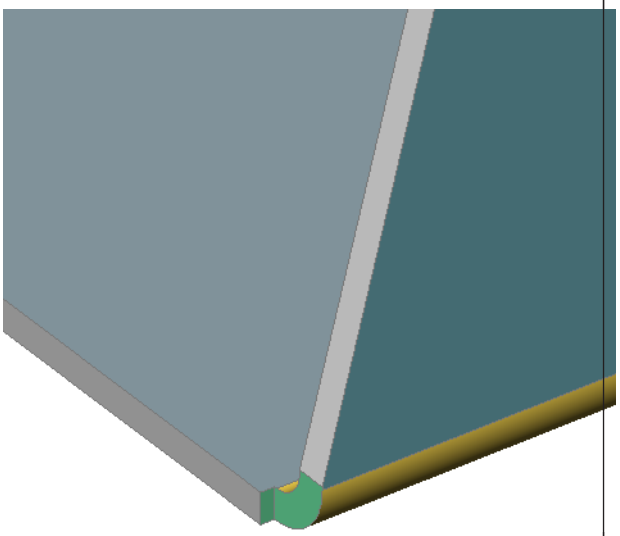
  O alívio é criado.

	
Faces dobradas selecionadas	Alívios criados

Nota:

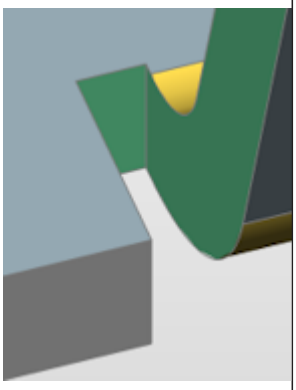
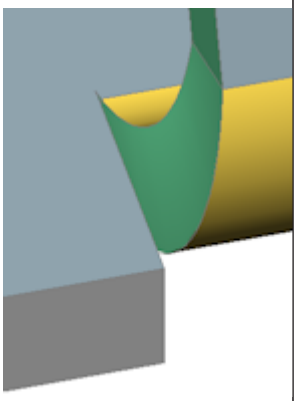
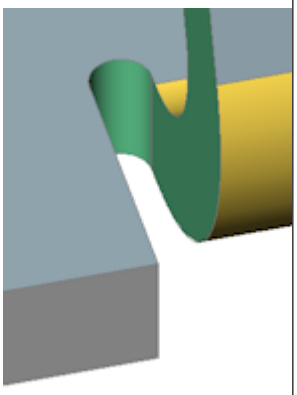
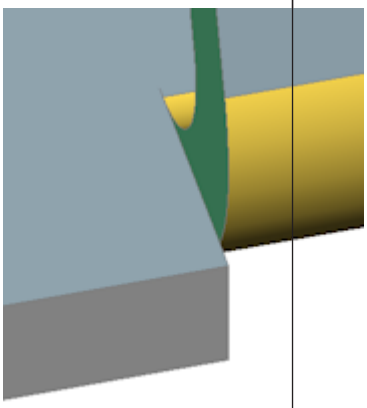
- Os itens selecionados podem ser um sólido 3D com recursos de chapa metálica, uma face ou uma borda rígida do sólido.
- Para criar localmente um alívio você pode selecionar 1 ou 2 bordas rígidas, 1 ou 2 dobras ou uma dobra e uma borda rígida.
- Se um sólido 3D for selecionado, o comando reconhece os lugares onde alívios de canto ou de dobra são necessários e os cria automaticamente.
- Se forem selecionadas 2 arestas rígidas ou 2 dobras ou uma dobra e uma borda rígida em um canto, o comando vai tentar criar um alívio de canto.

A opção **Forçar alívios de Dobra** força a criação do alívio de dobra mesmo se não houver necessidade de criar um alívio de dobra. Conforme mostrado na imagem abaixo, não há necessidade de criar um alívio de dobra, mas a geometria é diferente do corte retangular.

	
Um alívio de dobra não é necessário devido à transição suave da geometria.	Um alívio de dobra retangular é criado apenas se a opção **Forçar alívios de Dobra** estiver ativada.

21.5.18.4 Trocar o Tipo de Alívio de Dobra

O tipo de alívio de dobra pode ser alternado entre 4 tipos:

			
Retangular	Suave	Redondo	Rasgo

Nota:

- Um alívio **Rasgo** tem uma largura muito pequena de 0,001 * espessura. Se você precisar remover completamente o alívio, altere sua largura para 0. Esta operação remove a geometria de alívio da dobra e o recurso de alívio de dobra.
- Os tipos de alívios de dobra **Suave**, **Redondo** e **Rasgo** podem ser aplicados somente se o tipo de alívio de dobra atual for **Retangular**.

Usando a Faixa de opções, menu ou barra

1 Para escolher o tipo desejado de alívio, faça um dos seguintes:
   - Clique no botão **Alívio** de lista no painel **Sheet Metal / Modificar** na Faixa de opções e selecione **Mudar para Retangular**, **Mudar para Suave**, **Mudar para Redonda** ou **Mudar para Rasgo**.
   - Clique em um botão de ferramenta na barra **SM Alívio**: **Trocar para Alívio Retangular** (), **Trocar para Alívio Suave** (), **Mudar para Alívio Redondo** () ou **Trocar para Alívio de Rasgo** ().
   - Escolha o tipo de alívio no menu **Sheet Metal / Alívio**.

     Você é solicitado: Selecionar faces, sólidos 3d para trocar os alívios para o tipo-V [Retangular/Circular/Tipo-V/Suave/Rasgo/Redondo/Extensão de alívio/Modelo inteiro] <Modelo inteiro>:

2 Siga um dos procedimentos:
   - Pressione **Enter** para aplicar o tipo selecionado de alívio para o modelo inteiro.
   - Clique uma flange ou dobra vizinha, ou uma face do alívio de dobra.

     O alívio de dobra é convertido.

3 Se o comando foi chamado a partir da Faixa de opções, o comando repete o aviso principal até que você pressione a tecla **ESC**, para parar.

Usar o Quad

1 Paire o mouse sobre a flange vizinha, ou dobra, ou uma face do alívio de canto.

   Se a opção **Selecionar Faces** da variável de sistema **SELECTIONMODES** estiver selecionada, a flange ou a face se destaca; caso contrário, mantenha pressionada a tecla **CTRL** ao pairar até que a face se destaque.

2 Selecione o tipo desejado de alívio na aba **Sheet Metal** no **Quad** menu.

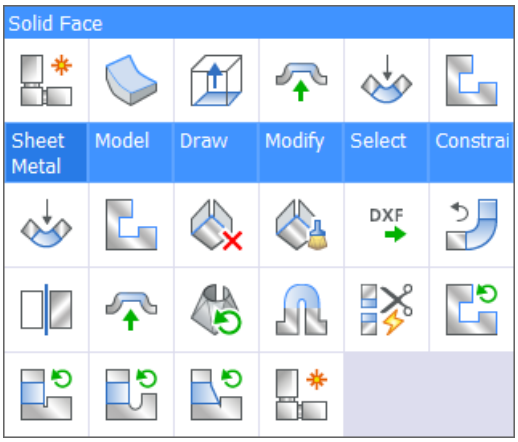
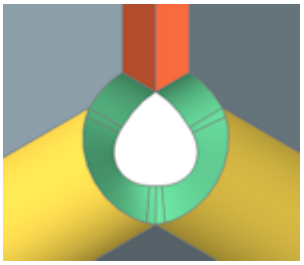
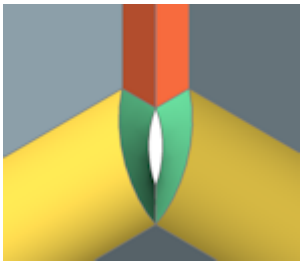


Figura 135: Sheet Metal

21.5.18.5 Trocando o Tipo de Alívio de Canto

O tipo de alívio de canto pode ser trocado entre 3 tipos:

		
Retangular	Circular	Tipo-V

Usando a Faixa de opções, menu ou barra:

1 Para escolher o tipo desejado de alívio, faça um dos seguintes:
- Clique no botão **Alívio** no painel **Sheet Metal / Modificar** na Faixa de opções, e selecione um tipo de alívio.
- Clique em um botão de ferramenta na barra **SM Alivio**.
- Escolha o tipo de alívio no menu **Sheet Metal / Alívio**.

  Você é solicitado: Selecionar faces, sólidos 3d para trocar os alívios para o tipo-V [Retangular/Circular/Tipo-V/Suave/Rasgo/Redondo/Extensão de alívio/Modelo inteiro] <Modelo inteiro>:

2 Siga um dos procedimentos:
- Pressione **Enter** para aplicar o tipo selecionado de alívio para o modelo inteiro.
- Clique em uma flange ou dobra vizinha, ou uma face do alívio de canto.

  O alívio de canto é convertido.

3 Se o comando foi chamado a partir da Faixa de opções, o comando repete o aviso principal até que você pressione a tecla **ESC**, para parar.

Usando o Quad:

1 Verifique se a opção **Selecionar Faces** da variável de sistema **SELECTIONMODES** está selecionada.

2 Paire o mouse sobre a flange vizinha, ou dobra, ou uma face do alívio de canto.

3 Selecione o tipo de alívio desejado no grupo de comandos **Sheet Metal** no menu **Quad**.

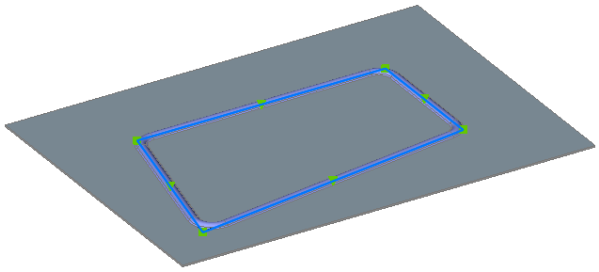


Figura 136: Sheet Metal

21.5.19 Trabalhando com Recursos de Forma e Nervura

21.5.19.1 Visão Geral

Os Recursos de Forma e Nervura são criados por uma ferramenta de molde. Os recursos de Nervura são produzidas ao estampar uma roda percorrendo a flange, seguindo uma trajetória arbitrária. Usaremos 'forma' para se referir a ambos os recursos de forma e nervura. Os recursos do forma podem ser reconhecidos na geometria importada, ou inseridos a partir de bibliotecas integradas ou definidas pelo usuário. Recursos de forma podem ser editados diretamente e parametricamente. Os recursos de forma são listados no **Navegador de Mecânica** com seus parâmetros. Os parâmetros dos recursos de forma são exibidos e podem ser editados no **Painel de Propriedades** quando selecionado.

Os seguintes comandos se aplicam aos recursos de forma:

Ícone	Comando	Descrição
	SMFORM	Converte um conjunto selecionado de faces em um recurso de forma. Um arquivo de desenho com um recurso de forma definido pelo usuário pode ser salvo, e inserido com o comando *BMINSERT*.
	SMCONVERT	Reconhece recursos de forma e de nervura na geometria importada.
	SMREPLACE	Substitui recursos de forma (incluindo aqueles reconhecidos) em peças de Sheet Metal com recursos de forma de uma biblioteca interna, ou do usuário.
	SMRIBCREATE	Cria recursos associativos de Nervura em peças de chapa metálica a partir de perfis 2D. O recurso é atualizado automaticamente quando o perfil definidor for modificado.

Ícone	Comando	Descrição
	BMINSERT	Insere um componente mecânico existente ou um recurso de forma de Sheet Metal no desenho atual
	SMSELECT	Seleciona recursos de forma comparáveis. O comando funciona em dois modos: • **Similar**: recursos que possuem o mesmo conjunto de faces, mas possivelmente tamanhos diferentes. • **Mesmo**: recursos idênticos.

Uma biblioteca de recursos de forma de sheet metal, parametrizados, está disponível na aba Recursos de Forma no painel **Paletas Ferramenta**. A biblioteca contém os tipos mais usados de recursos de forma. Para inserir um recurso de forma em um sólido 3D de sheet metal, arraste o ícone do recurso de forma do painel **Paletas de Ferramenta** ou do painel **Componentes**, para sobre uma flange de sólido.

A *SMDEFAULTFORMFEATUREUNFOLDMODE* (variável de sistema) controla a aparência do recurso de forma nas representações desdobradas em 2D e em 3D.

21.5.19.2 Para Inserir um Recurso de Forma

Recursos de forma são adicionados à peça de chapa metálica ao inserir esses nas flanges existentes. Os recursos de forma podem ser inseridos por meio do comando *BMINSERT* ou por arrastar-e-soltar na aba **Recursos do Forma** das **Paletas de Ferramentas**.

1 Certifique-se de que **Dimensões Dinâmicas** (DYN) e **UCS Dinâmico** (DUCS) estão ativos.

2 Siga um dos procedimentos:

- Clique no botão **Recurso de Forma** ( ) na barra **Sheet Metal**.
- Clique no botão **Recurso de Forma** ( ) na aba **Sheet Metal** da Faixa de opções.
- Escolha **Inserir Recurso de Forma** no menu **Sheet Metal**.
- Digite *BMINSERT* na barra de Comando, e pressione **Enter**.

  A caixa de diálogo *Selecionar arquivo para inserir* exibe a pasta **FormFeatures** [Pasta de instalação do BricsCAD]\Support\DesignLibrary\SheetMetal\FormFeatures:

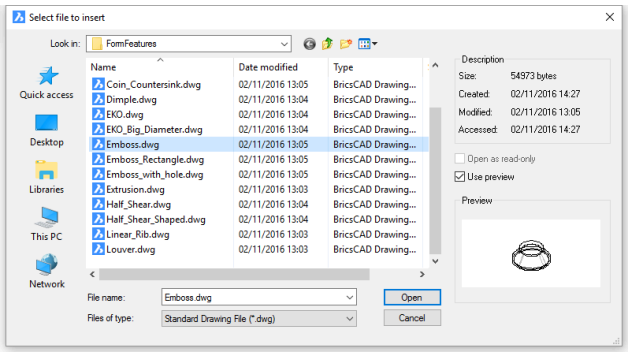


Figura 137: Selecione arquivo para inserir

Selecione o recurso de forma desejado, e clique duas vezes no recurso ou clique **Abrir**.

- Selecione o recurso de forma da aba Recursos de Forma do painel **Componentes**:

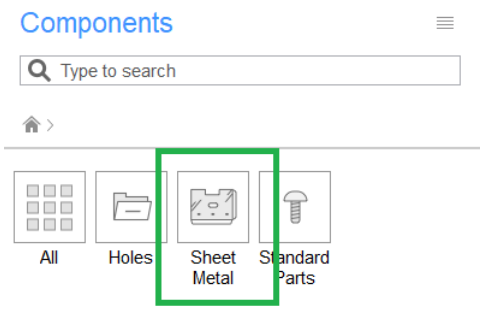


Figura 138: Componentes

- Selecione o recurso de forma na aba **Recursos de Forma** do painel **Paletas Ferramenta**:



Figura 139: Paletas Ferramenta

3 Mova o cursor sobre a face da flange na qual você deseja inserir o recurso.

O UCS dinâmico se alinha à face selecionada.

A orientação do UCS depende da borda em que o cursor entra na face.

As dimensões dinâmicas são exibidas da origem do UCS até o ponto de inserção do recurso de forma.

Você é solicitado:

Selecione o ponto de inserção ou [Editar entidade inserida/Rotacionar componente/definir ponto Base/Nome/Tipo de inserção/Inverter/Múltiplo]<0, 0, 0>:

4 (opção) Escolha uma opção (consulte o comando *BMINSERT* para mais detalhes):
   - **Editar**: Edite os parâmetros da inserção selecionada.
   - **Rotacionar**: Defina o ângulo de rotação do componente.
   - **Ponto Base**: Especifique um ponto base diferente para inserir o componente.
   - **Nome**: Defina o nome de inserção do componente.
   - **Tipo de Inserção**: Especificar o tipo de inserção: Local ou Externa.
   - **Inverter**: Insira o recurso de forma na face oposta da flange.
   - **Múltiplo**: Insira várias inserções do mesmo componente, ou crie uma matriz.
5 Para inserir um recurso de forma proceda da seguinte maneira:
   - Especifique um ponto.
   - Digite uma distância nos campos de dimensão dinâmica. Aperte a tecla **TAB** para ir e voltar entre os campos de entrada.
6 Repita a etapa 5 para inserir outra cópia, ou pressione **Enter** para parar.

Para ter Recursos de Forma no painel de **Componentes**, você deve criar ou utilizar a área de trabalho **Projetos de Mecânica** no diálogo **Lançador BricsCAD**.

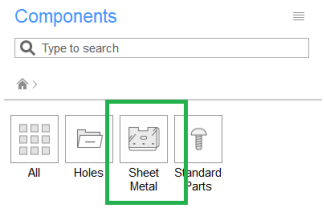


Figura 140: Componentes

21.5.19.3 Editar os Parâmetros de uma Biblioteca de Recursos de Forma

Recursos de forma na biblioteca de recursos de forma são componentes paramétricos. Vários parâmetros controlam suas dimensões. Parâmetros para controlar Comprimento (Length), Largura (Width), Altura (Height) e Raio do recurso de forma são denominados L, W, H e R. A espessura do recurso de forma é definida automaticamente igual à espessura da peça de sheet metal.

Os parâmetros de um único recurso de forma podem ser editados no **painel Propriedades** ou no **Navegador de Mecânica**. Parâmetros de múltiplos recursos de forma somente podem ser editados no **painel Propriedades**.

Métodos para selecionar recursos de forma

- Marque a opção **Selecionar Faces** de *SELECTIONMODES* , e então clique nos recursos do modelo.
- Selecione o(s) recurso no **Navegador de Mecânica**, e pressione a **barra de espaço** ou clique o botão-direito e escolha **Selecionar** no menu de contexto. Pressione e segure a tecla **CTRL** para selecionar múltiplos recursos de forma.

- Uma peça de Sheet Metal pode ter vários recursos de forma diferentes, o que pode complicar operações como exclusão ou substituição, especialmente se apenas uma seleção de recursos, que atendam a determinadas condições, necessita ser processada. Para selecionar mais facilmente recursos de forma, o comando **SMSELECT** permite selecionar recursos de forma que são idênticas a um recurso selecionado ou semelhante.

Editar parâmetros na barra das Propriedades

1 Selecione o(s) recurso de forma.

2 Selecione um parâmetro no nó **Parâmetros** no **painel Propriedades**.

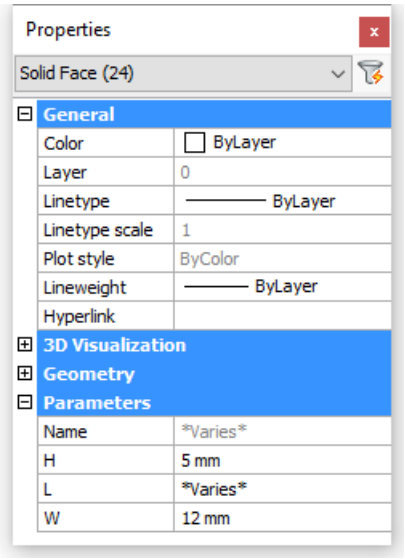


Figura 141: Propriedades

Editar parâmetros no Navegador de Mecânica

1 Selecione o recurso de forma no nó **Forma** da árvore da estrutura de componentes.

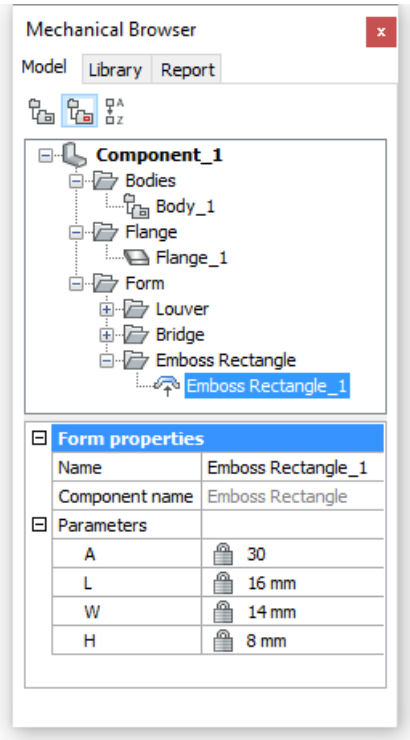
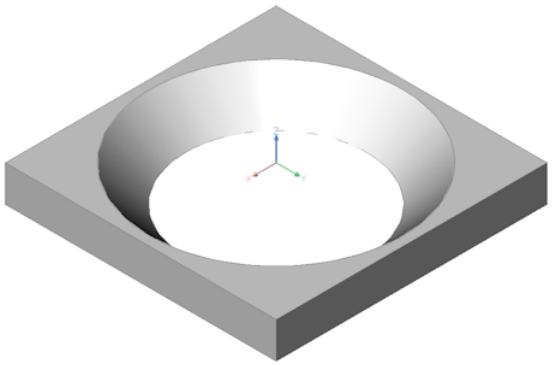


Figura 142: Navegador de Mecânica

2 Selecione um parâmetro no painel inferior do **Navegador de Mecânica** e digite um novo valor.

21.5.19.4 Criar um recurso de forma personalizado

1 Abra um novo desenho usando o arquivo **Mechanical-mm.dwt** como template.



2 Crie a geometria 3D do recurso de forma.

3 Execute o comando *SMCONVERT* ( ).

4 Execute o comando *BMMECH* ( ).

Você é solicitado: Nome do componente mecânico <Component_2>:

5 Digite um nome para o recurso de forma.

Você é solicitado: Converter blocos e referências externas para componentes mecânicos? [Sim/Não] <Sim>:

6 Selecione a opção **Não**.

7 No campo **Tipo de extensão** no **Navegador de Mecânica**, selecione **Recurso de Forma de Sheet Metal**.

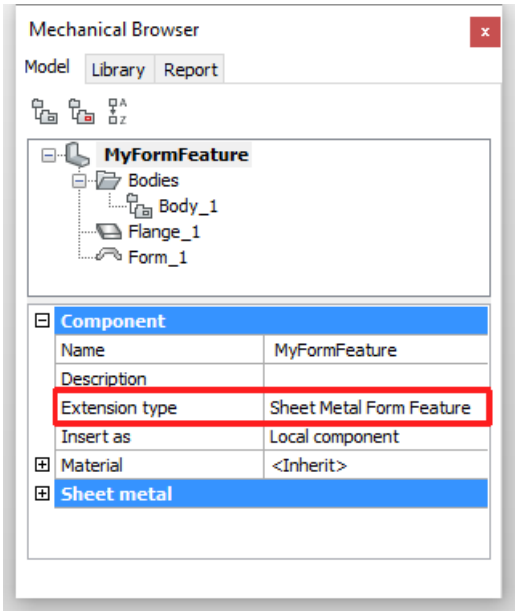


Figura 143: Navegador de Mecânica

8 Selecione os recursos Flange_1 e Form_1 no nó **Corpos**, então clique o botão-direito e selecione **Desabilitar** no menu de contexto.

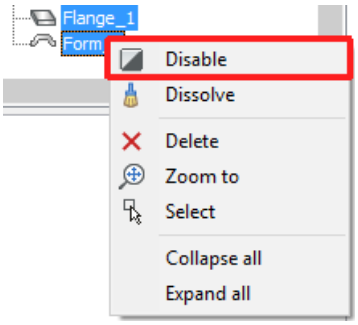
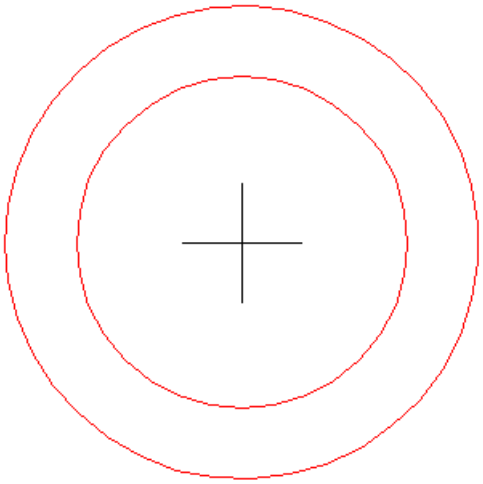


Figura 144: Diable

9 Crie 2 novas camadas: UNFOLD_SYMBOL_UP e UNFOLD_SYMBOL_DOWN.

10 Torne as camadas 0 e UNFOLD_SYMBOL_DOWN **Desl** e defina a camada UNFOLD_SYMBOL_UP como a **Atual**.

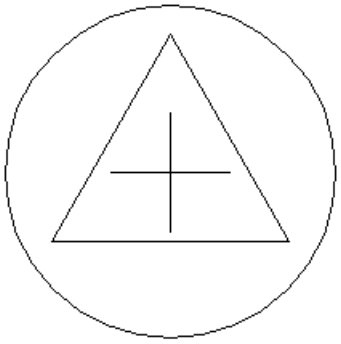
11 Crie o símbolo planificado, que será colocado na face na qual o recurso de forma é



colocado.

12 **Desligue** a camada UNFOLD_SYMBOL_UP e **Ligue** a camada UNFOLD_SYMBOL_DOWN. Defina a camada UNFOLD_SYMBOL_DOWN como **Atual**.

13 Crie o símbolo planificado, que será colocado na face oposta em que o recurso de forma é



colocado.

14 **Ligue** a camada 0 e **Desligue** as camadas UNFOLD_SYMBOL_UP e UNFOLD_SYMBOL_DOWN. Defina a camada 0 como a **Atual**.

15 Salve o desenho.

21.5.19.5 Criar um recurso de forma personalizado paramétrico

Nós iremos agora parametrizar o recurso de forma personalizado que criamos no procedimento anterior.

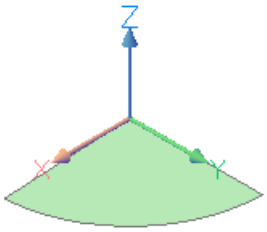
São necessários 2 parâmetros:

- **D** para um diâmetro da borda maior do furo cônico.
- **A** para um ângulo da face do furo cônico.

Em Sheet Metal, o parâmetro padrão **_SM_THICKNESS** controla a espessura da peça de chapa metálica.

1 Crie um corpo auxiliar para restringir uma nova camada CONSTRUCTION.

Torne esta camada atual, e desligue todas as outras camadas.



2 Crie uma Regiao no plano XY do WCS, na camada CONSTRUCTION.

3 Aplique uma restrição *DMFIX3D* ( ) à região.

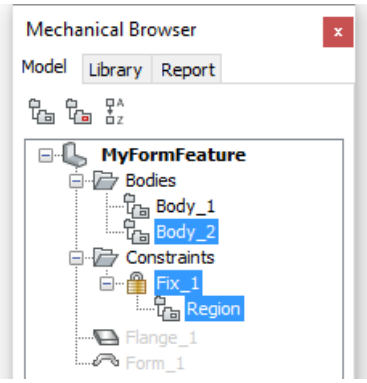
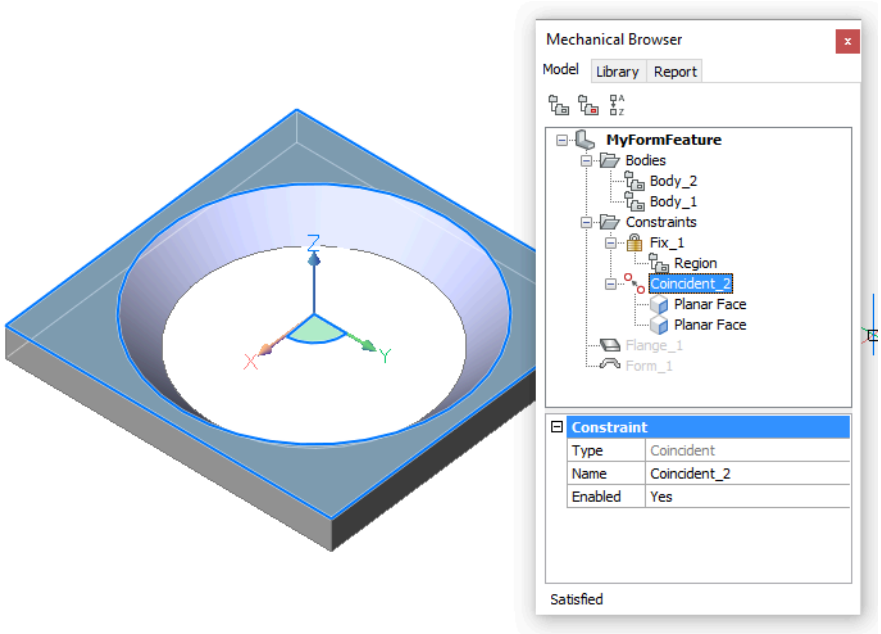


Figura 145: Navegador de Mecânica

4 **Ligue** a camada 0.

5 Aplique uma restrição *DMCOINCIDENT3D* ( ) entre a face da região e a face superior da flange.

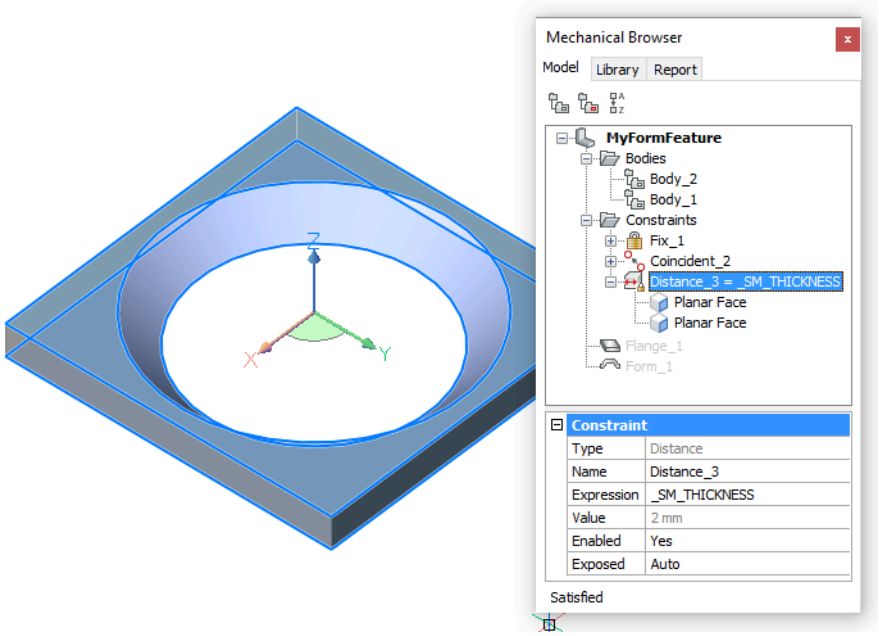


6 Aplicar uma restrição *DMDISTANCE3D* ( ) entre as faces superior e inferior da flange.

Primeiro, selecione a face superior e pressione a tecla **TAB** para selecionar a face inferior.

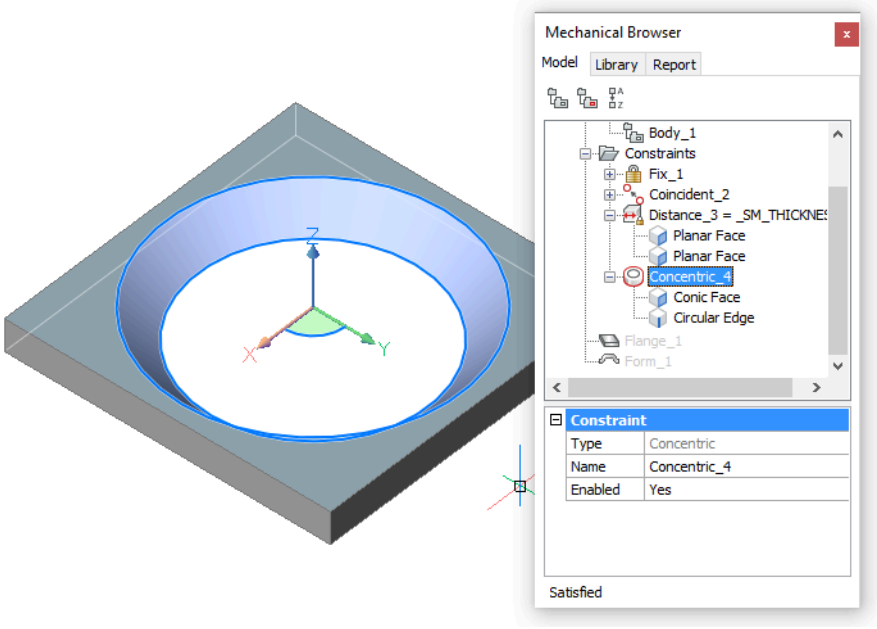
Aceite o valor padrão na dimensão dinâmica.

7 Digite _SM_THICKNESS no campo **Expressão** da restrição



Distance_3.

8 Desde que nosso Recurso de Forma é cônico podemos modificar a flange de um paralelepípedo para



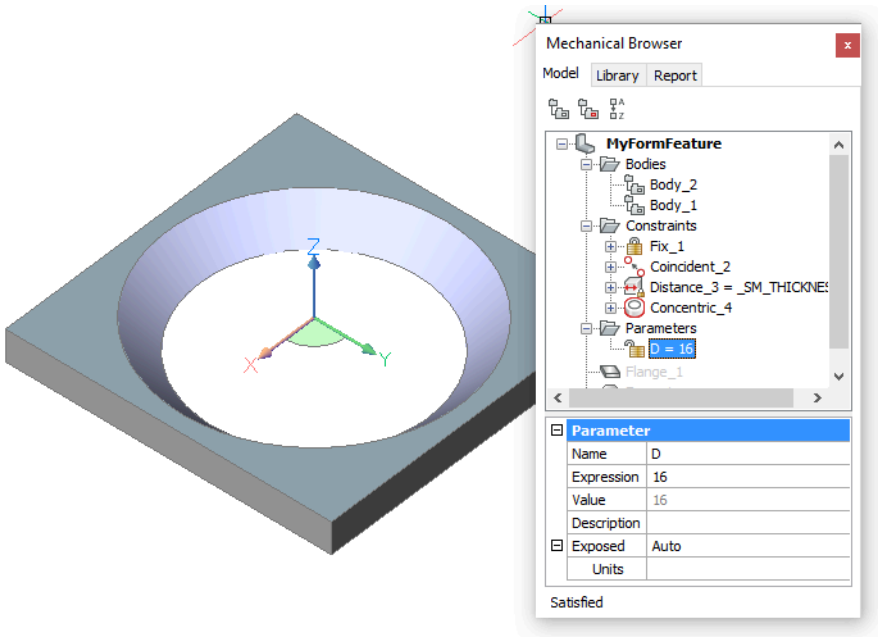
um cilindro.

9 No **Navegador de Mecânica**, clique o botão-direito do mouse no nome do recurso e selecione **Adicionar novo parâmetro** no menu de contexto.

Renomeie o novo parâmetro para **D.**

Selecione mm para o campo **Unidades**.

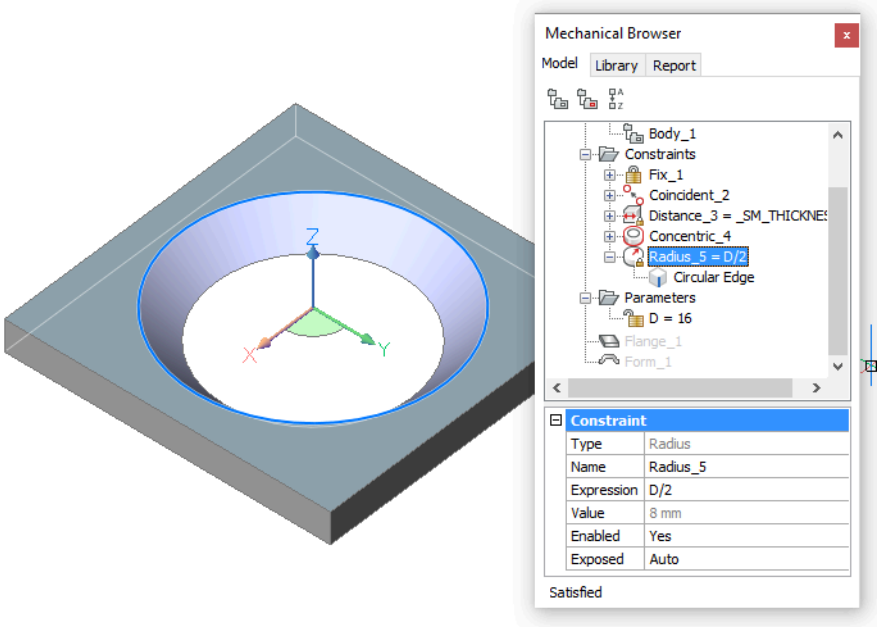
Digite 16 no campo **Expressão**.



10 Aplique a restrição *DMRADIUS3D* ( ) na borda superior da face cônica.

Aceite o valor padrão na dimensão dinâmica.

11 Digite D/2 no campo **Expression** da restrição



Radius_5.

12 No **Navegador de Mecânica** clique o botão-direito no nome do recurso e selecione **Adicionar novo parâmetro** no menu de contexto.

Renomeie novo parâmetro para **A**.
Deixe o campo **Unidades** vazio.

Digite 45 no campo **Expressão**.

13 Siga um destes procedimentos para criar uma Restrição de **Angulo do Cone** à face cônica:

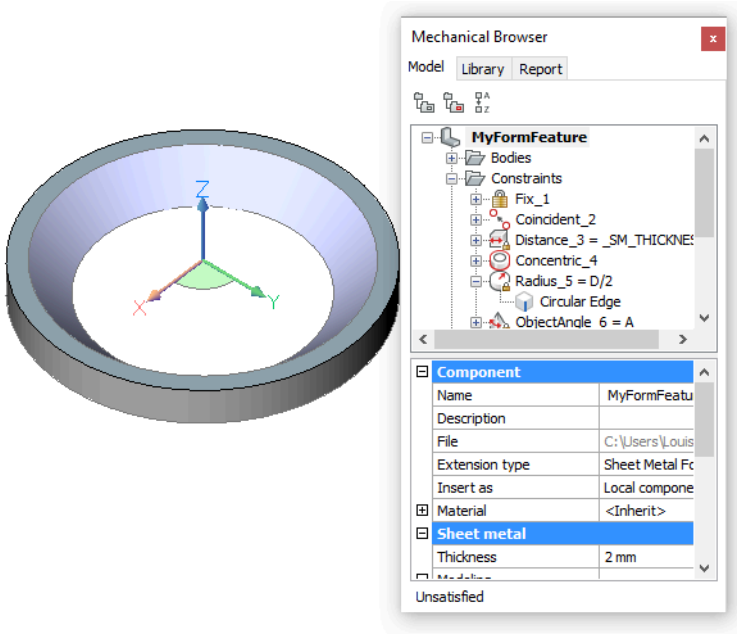
- Lance o comando *DMANGLE3D* (), e escolha a opção de **restrição do Angulo do Cone**, então selecione a face cônica.
- Passe o mouse sobre a face cônica, então escolha a **restrição Angulo do Cone** () no grupo de comandos **Restrições** no **Quad**.

Aceite o valor padrão.

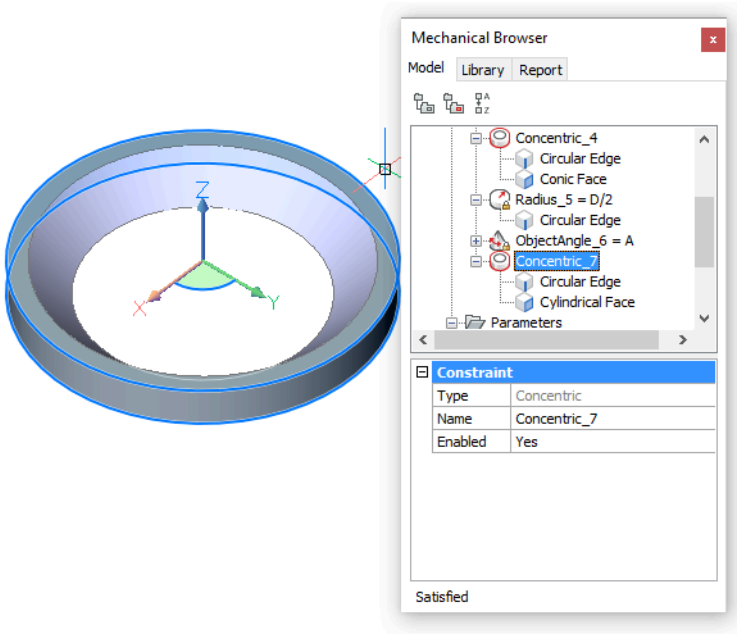
14 Digite A no campo **Expression** da restrição ObjectAngle_6.

15 Como nosso Recurso de Forma é cônico, modifique a flange de paralelepípedo para cilindro:

- Desenhe um círculo na parte superior do flange. O raio do círculo é de 9 mm.
- Selecione **Ativar Detecção de Limite** no painel **Configurações** na aba **Home** da Faixa de Opções.
- Coloque o cursor sobre a área entre o círculo e a aresta quadrada da flange, então escolha **Extrusao** () no **Quad** e mova o cursor para baixo.



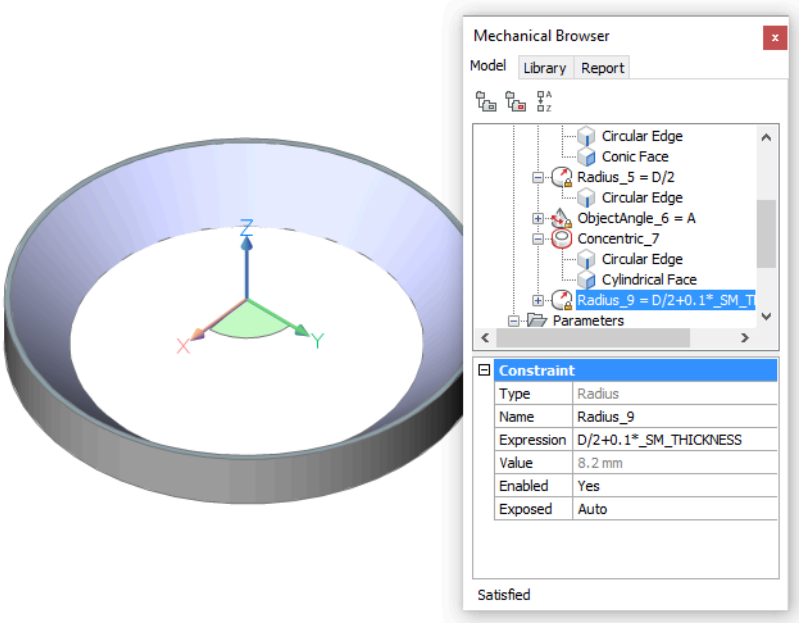
16 Aplique a restrição *DMCONCENTRIC3D* () entre a borda circular da região e a face cilíndrica.



17 Aplique a restrição *DMRADIUS3D* ( ) para a face cilíndrica.

Aceite o valor padrão na dimensão dinâmica.

18 Digite D/2+0.1*_SM_THICKNESS no campo **Expressão** da restrição



Radius_8

19 Defina o campo **Exposto** dos parâmetros A e D **Ligado**.

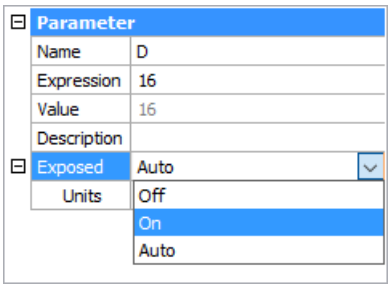


Figura 146: Parâmetro

Agora crie restrições 2D para os símbolos desdobrados.

20 Torne a camada UNFOLD_SYMBOL_UP a **Atual** e **Desligue** todas as outras camadas.

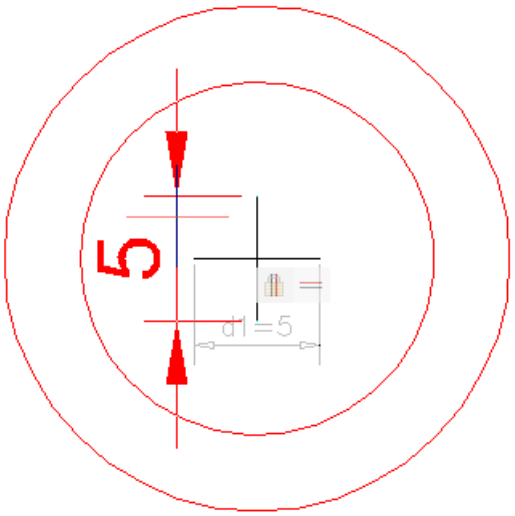
21 Aplique uma restrição *RGFIXA* ( ) para ambas as linhas cruzadas que indicam o centro dos círculos.

22 Aplique a restrição *RGHORIZONTAL* ( ) à linha central horizontal.

23 Aplique a restrição *RGVERTICAL* ( ) à linha central vertical.

24 Aplique a restrição *RGCONCENTRICA* ( ) entre os 2 círculos.

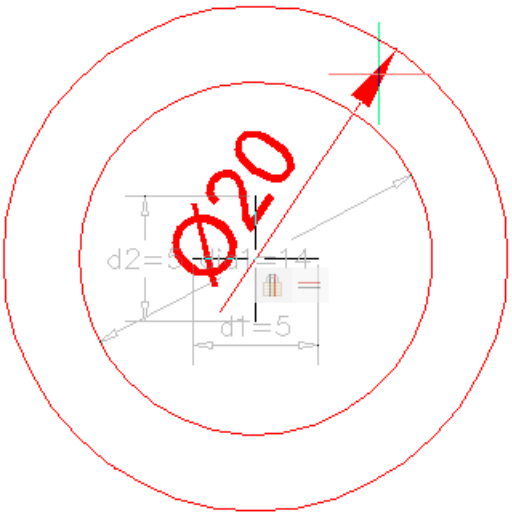
25 Aplique a restrição *RDLINEAR* ( ) entre os pontos finais das linhas centrais. Aceite o valor



padrão.

26 Aplique a restrição *RDDIAMETRO* ( ) para ambos os círculos.

Aceite o valor padrão para dia1 (=círculo interno) e dia2 (= círculo externo).



27 Ajustar os campos **Expressão** das restrições 2D como segue:

- d1 = 1
- d2 = d1
- dia 1 = D - 2*_SM_THICKNESS/tan(A)
- dia 2 = D

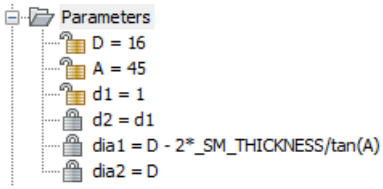
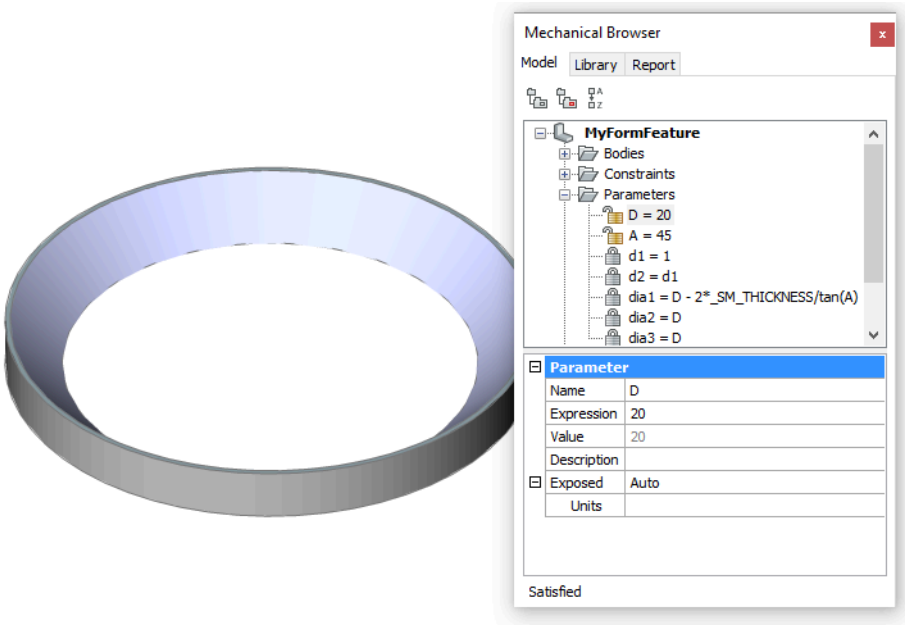


Figura 147: Parâmetros

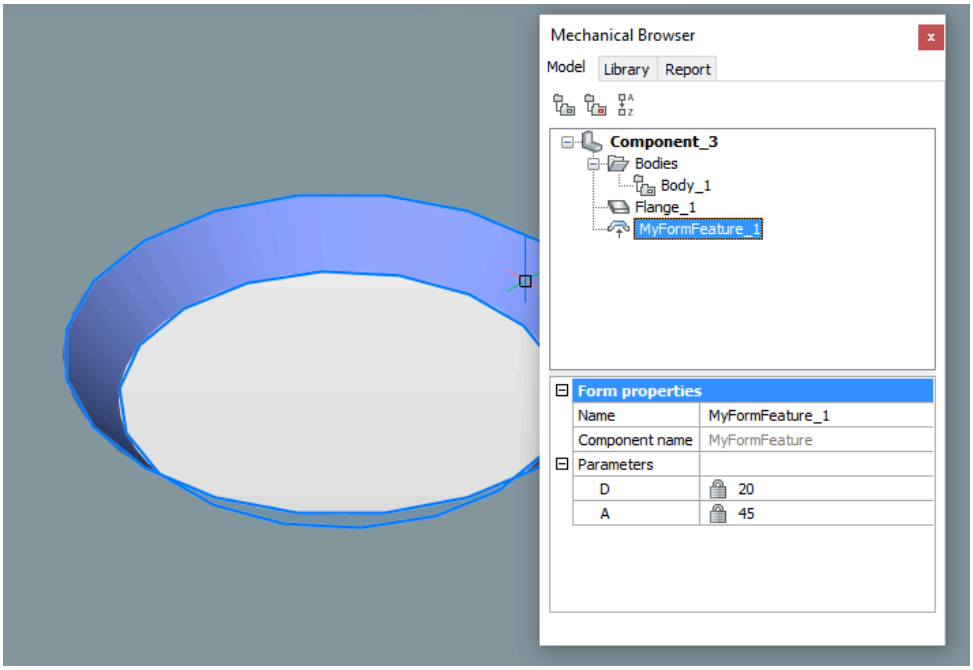
28 Defina **Desl** para exibição de todas as camadas, exceto a camada 0, e salve o



arquivo.

O recurso de forma agora pode ser usado para criar aberturas paramétricas cônicas em uma flange de sheet metal.

Use a ferramenta **Inserir Recurso de Forma** (), que ativa o comando *BMINSERT*.



Opcionalmente: Adicione o recurso de forma como um componente para uma paleta de ferramentas.

21.5.19.6 Para criar um recurso de nervura

Recursos de Nervura são criadas pelo comando *SMRIBCREATE* a partir de uma curva 2D na face da flange.

1 Siga um dos procedimentos:

- Clique o botão de ferramenta **Criar Recurso de Nervura** () na barra **Sheet Metal**.
- Clique no botão **Criar Recurso de Nervura** () no painel **Modifica**r da aba **Sheet Metal** na Faixa de opções.
- Digite *SMRIBCREATE* na barra de Comando, e pressione **Enter**.

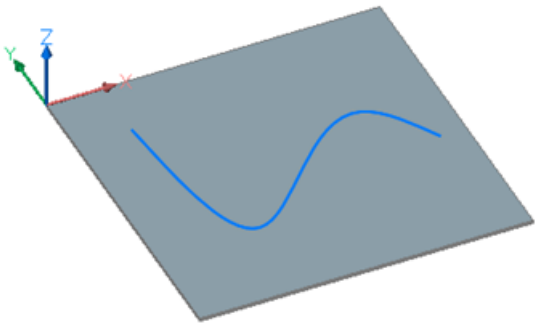
  Você é solicitado: Selecione a face da flange:

2 Selecione a face da flange.

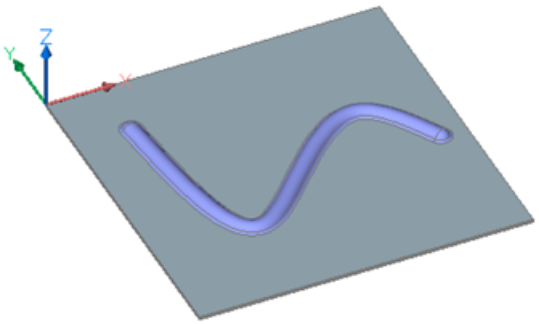
  Você é solicitado: Selecione a curva ou [Raio do perfil/Raio redondo]:

3 Selecione uma curva na face da flange selecionada.

  Linhas, polilinhas, círculos, arcos, elipses, arcos elípticos e splines são aceitos. Uma curva 2D pode ser aberta ou fechada, mas não pode se auto-cruzar ou outros recursos de forma.



Após a seleção da curva, um recurso de nervuras é criado:



21.5.19.7 Para modificar um recurso de nervura

Para modificar uma nervura:

- Modifique a 'curva geratriz' usando suas alças de manuseio.
- Edite o **Perfil** e o raio **Redondo** no **Navegador de Mecânica**.

Modificar a curva geratriz

1 Selecione a curva.

Como a curva está oculta pela nervura, pressione a tecla **TAB** para selecionar entidades sobrepostas na visualização, ou selecione a curva no painel **Estrutura**
.

2 Edite a curva pelas alças.

O recurso Nervura é atualizado de acordo com as alterações da curva.

3 Depois de selecionar a face do flange, você será solicitado:

Selecione a curva ou [Raio do perfil/Raio redondo]:

4 Selecione a curva no lado da flange selecionada.

Editar as propriedades no Navegador de Mecânica

1 Selecione o recurso de nervura no **Navegador de Mecânica**.

O recurso de nervura é destacado no modelo.

2 Selecione a propriedade na grade **Propriedades da Nervura**.

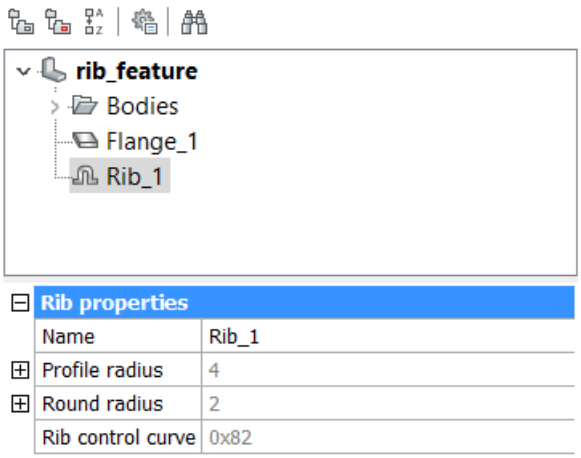


Figura 148: Propriedades nervura

3 Digite um valor.

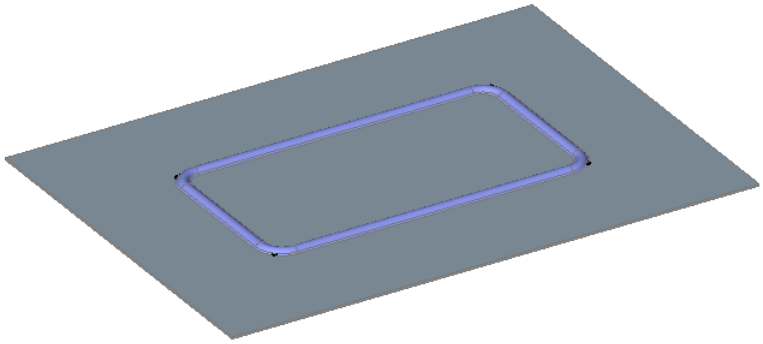
O recurso de nervura selecionado é atualizado.

21.5.19.8 Entender a diferença entre curvas de Controle e curvas de Trajetória

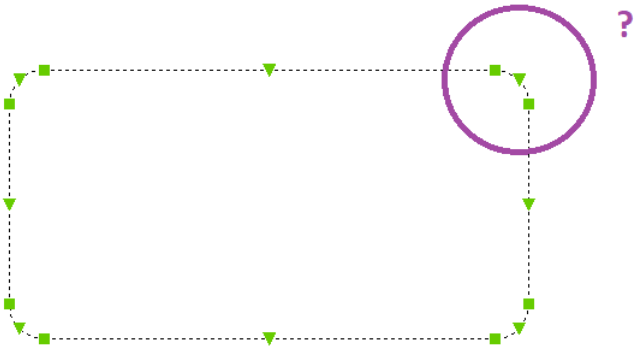
Ao criar um recurso de nervura retangular, primeiro desenhamos um retângulo que é então usado para criar um recurso de nervura pelo comando *SMRIBCREATE*. Quando esta curva de controle é editada por suas alças, o recurso de nervura é atualizado de acordo.



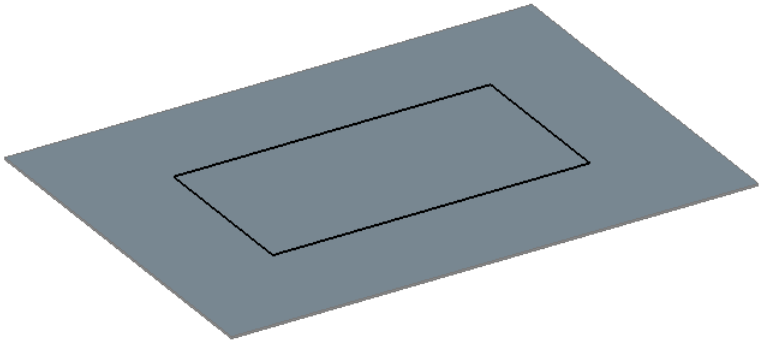
No lado da produção, a curva de controle retangular é substituída por uma curva de trajetória em concordância, de acordo com as limitações físicas da ferramenta de roda:



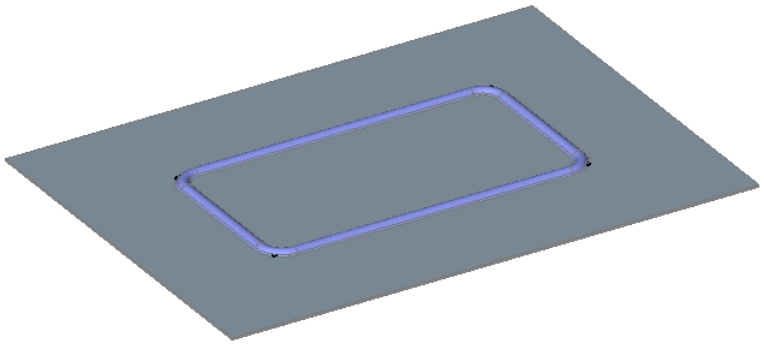
A curva de trajetória, mesmo correta, é difícil de manipular, por exemplo quando você quer mudar de lugar um canto.



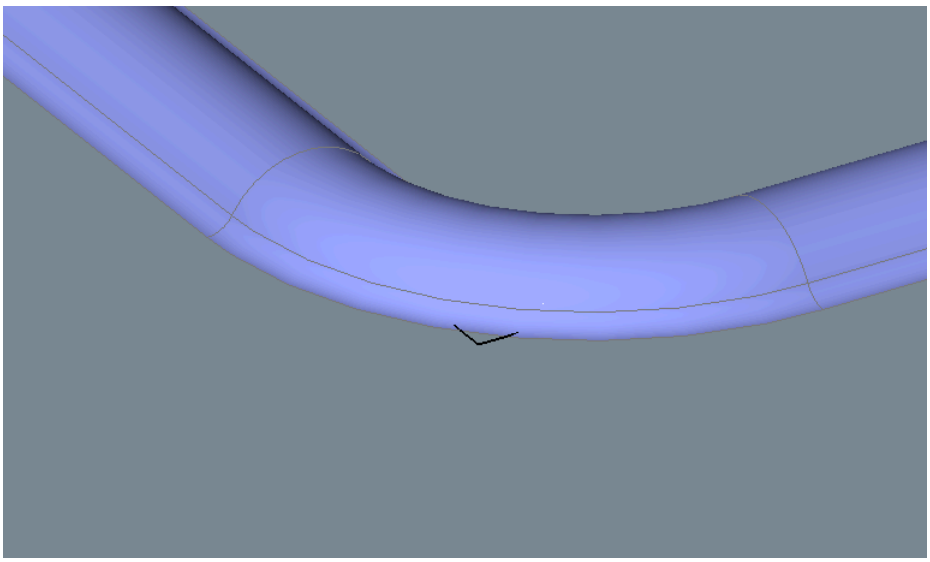
Na fase de projeto, usamos a **Curva de Controle**:



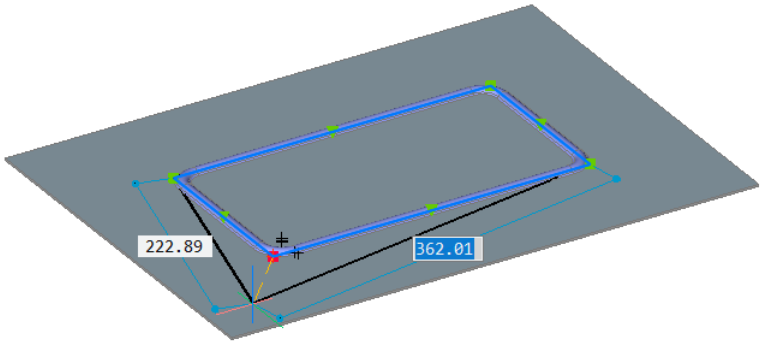
Ao criar o recurso de nervura, o comando *SMRIBCREATE* adiciona filetes de acordo com o valor do parâmetro **Raio de concordância da nervura** no **Navegador de Mecânica**. Este parâmetro não é controlado por recurso, uma vez que se relaciona com as propriedades físicas da ferramenta de roda:



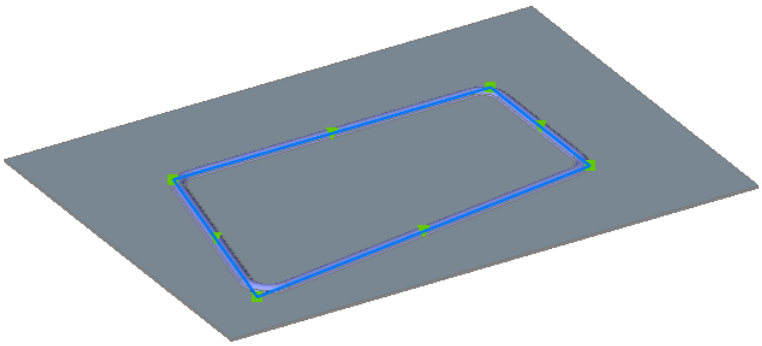
Ao aproximar em Zoom, você percebe a diferença entre a geometria 3D da nervura, e a curva de controle:



Quando a polilinha é editada por alça...

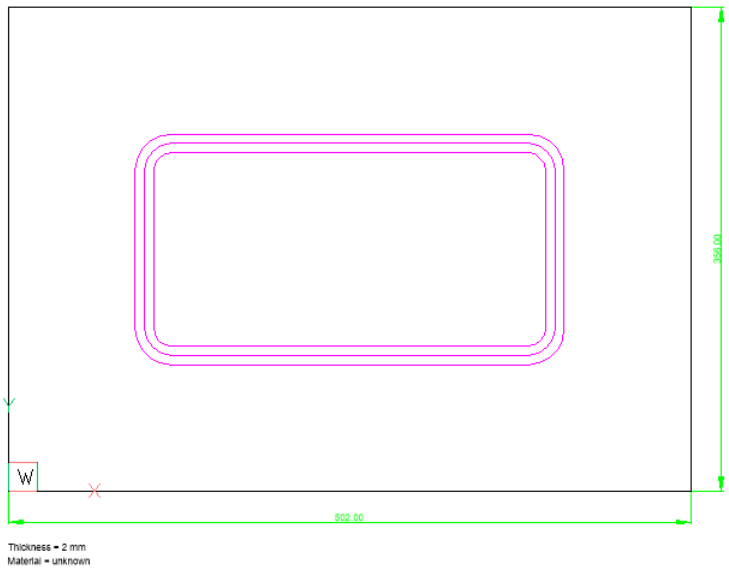


...o recurso de nervura é ajustado de acordo:



Isso ilustra a associatividade entre a curva 2D e o recurso de nervura 3D. O reconhecimento de recursos de costela no *SMCONVERT* é controlado pela *SmConvertRecognizeRibControlCurves* (preferência do usuário).

Quando a peça é desdobrada pelo comando *SMUNFOLD*, as curvas seguem o design 3D e têm as concordâncias. A curva do meio é chamada **Curva de Trajetória**:

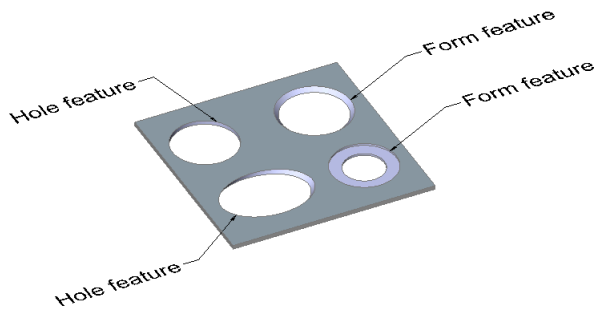


21.5.20 Trabalhar com Recursos de Furo

21.5.20.1 Visão Geral

Os recursos do Furo são uma versão simplificada dos Recursos de Forma. Na imagem abaixo você pode ver diferentes recursos, que têm somente faces de espessura (o comando *SMCONVERT* respeita estas regras):

- Furos ortogonais são recursos de Furo.
- Furos simples não-ortogonais são recursos de Furo.
- Furos ortogonais chanfrados são Recursos de Forma.
- Se a face da espessura do Furo consiste em 2 regiões diferentes, é um Recurso de Forma.



Recursos de Furo são reconhecidos pelo comando *SMCONVERT* se a preferência do usuário *SMCONVERTRECOGNIZEHOLES* estiver Ligada.

As seguintes operações básicas são permitidas para recursos de Furo:

- *SMDELETE*, *SMDISSOLVE*, desabilitar e *SMSELECT* estes, usando comandos ou através do **Navegador de Mecânica para Sheet Metal**.

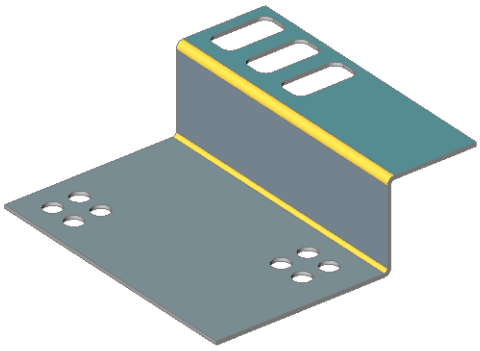
- **Modos de Recursos de Forma** no comando *SMUNFOLD*, incluindo modo **Símbolo**.
- Associe sua própria geometria 2D em um layout planificado.
- Os recursos do furo são preservados pelo comando *SMREPAIR*. Faces planas de furos se tornam ortogonais. Um recurso de Furo de mesma geometria não é afetado pelo comando *SMREPAIR*.

21.5.20.2 Reconhecimento de Matriz de Furos

Quando existem recursos de furo em um corpo, o comando *SMPARAMETRIZE* detecta matrizes retangulares de furos em flanges, de acordo com as seguintes regras:

- Furos que pertencem à mesma flange.
- Furos que estão em uma grade retangular.
- Não há lacunas (elementos ausentes) na matriz.
- Furos são igualmente orientados.
- O tamanho mínimo da matriz é 1x3 ou 2x2.

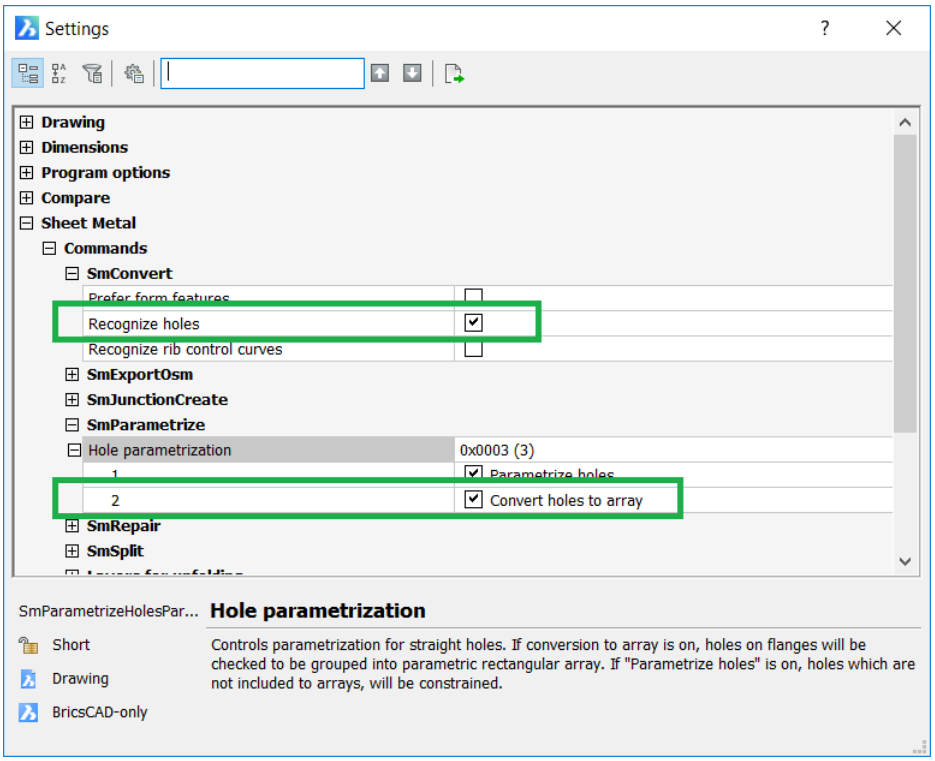
Vamos ilustrar o fluxo de trabalho em uma peça com recursos de Flange e Dobra.



1 Na caixa de diálogo *Configurações*, vá até **Sheet Metal > Comandos**.

2 Ative a opção **Reconhece furos** para o comando *SMCONVERT*.

3 Habilite as opções **Converter furos para matriz** e '**Parametrizar furos**', para o comando



SMPARAMETRIZE.

4 Execute o comando *SMCONVERT*.

5 Siga um dos procedimentos:

- Clique no botão **Converter para Sheet Metal** () na barra **Sheet Metal**;
- Clique no botão **Converter para Sheet Metal** () na aba **Sheet Metal** da Faixa de opções;
- Escolha **Converter para Sheet Metal** no menu **Sheet Metal**.

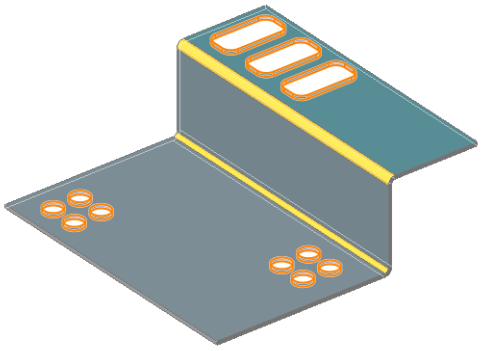
  Você é solicitado: Selecione Sólidos 3D ou [Entire model] <Entire model>:

6 Selecione o sólido 3D para converter.

Você é solicitado: Selecione sólidos 3D ou [Modelo inteiro] <Modelo inteiro>:

7 Pressione **Enter** para converter o sólido 3D em uma peça de Sheet Metal.

11 elementos de Furo são reconhecidos na peça.



8 Execute o comando *SMPARAMETRIZE*.

9 Siga um dos procedimentos:

- Clique no botão **Parametrizar** ( ) na barra de ferramentas **Sheet Metal**.
- Clique no botão **Parametrizar** ( ) na barra **Sheet Metal** da Faixa de opções.
- Escolha **Parametrizar** no menu **Sheet Metal**.

  Você é solicitado: Selecione sólidos 3D para parametrizar ou [Modelo inteiro] <Modelo inteiro>:

10 Pressione **Enter** para processar a peça de chapa metálica.

Os relatórios de comando devem se parecer com o abaixo:

Criadas 4 restrições de distância

Criadas 3 restrições de fixar

Criadas 0 restrições coincidentes

Criadas 0 restrições tangentes

Criadas 0 restrições de conjunto rígido

Criada matriz retangular 2x2 com base no recurso Hole_3

Criada matriz retangular 2x2 com base no recurso Hole_7

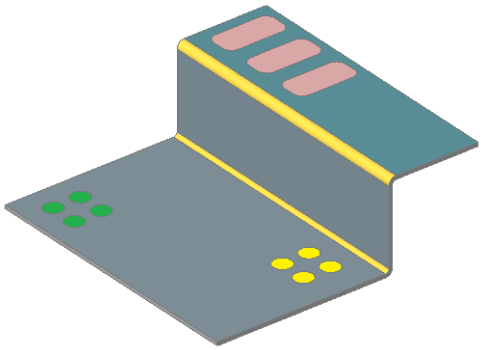
Criada matriz retangular 3x1 com base no recurso Hole_9

Total: 7 restrições e 3 matriz(es) criada(s)

Nota: Os nomes dos recursos e a distribuição das restrições podem variar dependendo da versão do BricsCAD.

Para examinar as matrizes, você pode descongelar temporariamente a camada **BC_SUBTRACT**.

Na imagem abaixo as matrizes são coloridas manualmente.

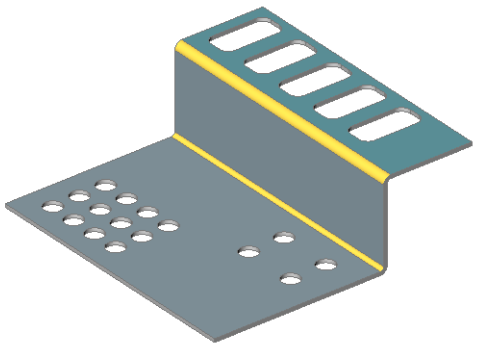


Nota: As matrizes verde e amarela não estão unificadas. Caso contrário, a matriz teria furos faltantes.

11 Edite as **Propriedades da Matriz** no **Navegador de Mecânica**.

Na imagem abaixo, o seguinte foi modificado:

- Matriz verde: número de linhas e colunas de 2x2 para 3x4
- Matriz amarela: espaçamento X e Y.
- Matriz vermelha: número de linhas e colunas de 3x1 para 5x1.



21.5.21 Converter para Sheet Metal e Reparar

Usar os comandos *SMCONVERT* e *SMREPAIR*.

21.5.21.1 Para converter geometria 3D em uma peça válida de Sheet Metal

Uma peça de chapa metálica pode ser criada a partir de qualquer geometria sólida 3D, seja criada no BricsCAD ou importada de outro software CAD. No entanto, a geometria pode não ser um modelo válido de chapa metálica: faltando dobras e alívios, ou que tenha faces de espessura não ortogonais às faces da flange. O BricsCAD Sheet Metal contém um conjunto de ferramentas dedicado para converter essas geometrias em peças válidas de chapa metálica, e adicionar todos os recursos faltantes.

1 Siga um dos procedimentos:

- Clique no botão **Converter para Sheet Metal** ( ) na barra **Sheet Metal**;
- Clique no botão **Converter para Sheet Metal** ( ) na aba **Sheet Metal** da Faixa de opções;
- Escolha **Converter para Sheet Metal** no menu **Sheet Metal**.

  Você é solicitado: Selecione Sólidos 3D ou [Entire model] <Entire model>:

2 Selecione o sólido 3D para converter.

  Você é solicitado: Selecione sólidos 3D ou [Modelo inteiro] <Modelo inteiro>:

3 Pressione **Enter** para converter o sólido 3D em uma peça de sheet metal.

As propriedades da peça de chapa metálica são exibidas no **Navegador de Mecânica para Sheet Metal**.

Nota: Os corpos convertidos podem ser desdobrados ou exportados para formatos de arquivo *.DXF ou *.OSM para processamento por um sistema CAM.

Em alguns casos, pode ser necessário corrigir corpos convertidos antes de desdobrar/exportar:

- Criar alívios faltantes: comando *SMRELIEFCREATE*.
- Converte bordas rígidas em dobras ou junções: comandos *SMBENDCREATE* e *SMJUNCTIONCREATE*.
- Flanges divididas e dobras em loft: comando *SMSPLIT*.
- Reparar corpos de sheet metal: comando *SMREPAIR*.

21.5.21.2 Para Reparar uma peça em Sheet Metal

Uma peça de sheet metal pode ter imprecisões de geometria e topologia ou em problemas do projeto de sheet metal. Este geralmente é o caso de peças importadas que são projetadas em núcleos geométricos diferentes do ACIS.

Exemplos de tais problemas são:

- Faces de espessura, que são medidas fora da perpendicular às faces de flange/dobra;
- Dobras em loft que são não-tangenciais às flanges (ou dobras) adjacentes;
- Dobras erradas;
- Problemas gerais em ACIS.

Tais problemas podem impedir que operações de edição sejam executados com êxito.

O fluxo de trabalho recomendado para corrigir tais problemas é:

- Execute o comando *DMAUDIT* para corrigir problemas gerais de modelagem.
- Execute o comando *SMREPAIR*.

21.5.21.3 Para Reparar um corpo de Sheet Metal

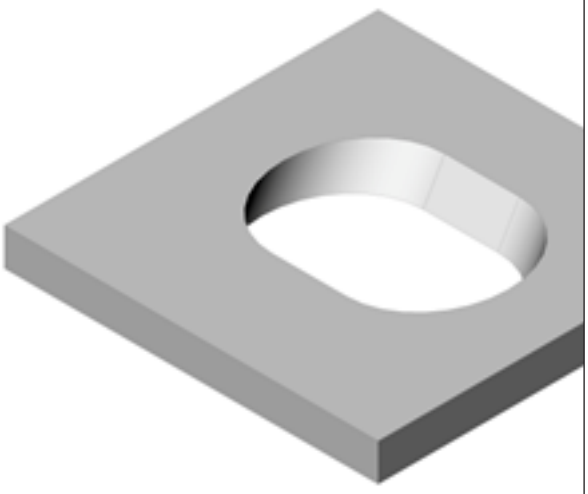
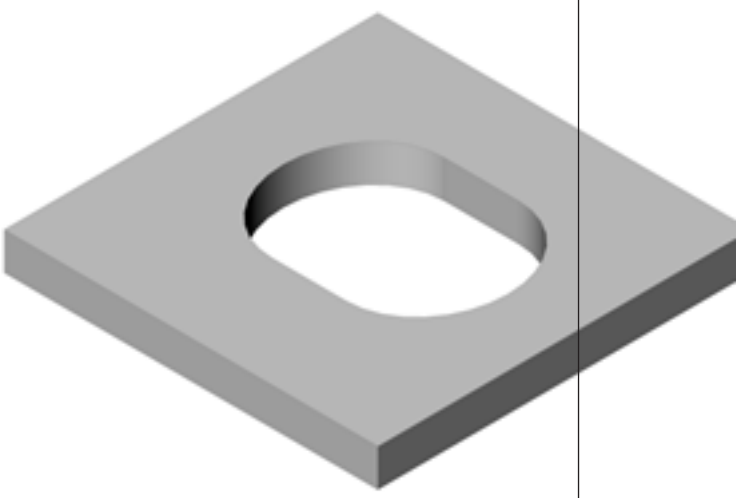
Os corpos de sheet metal com faces de espessura não-perpendiculares às faces de flange/dobra, não podem ser manufaturadas corretamente usando ferramentas tradicionais.

1 Siga um dos procedimentos:

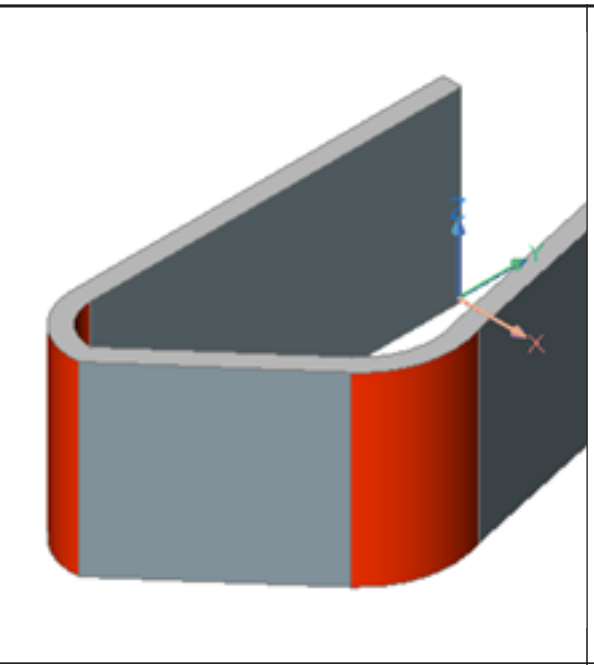
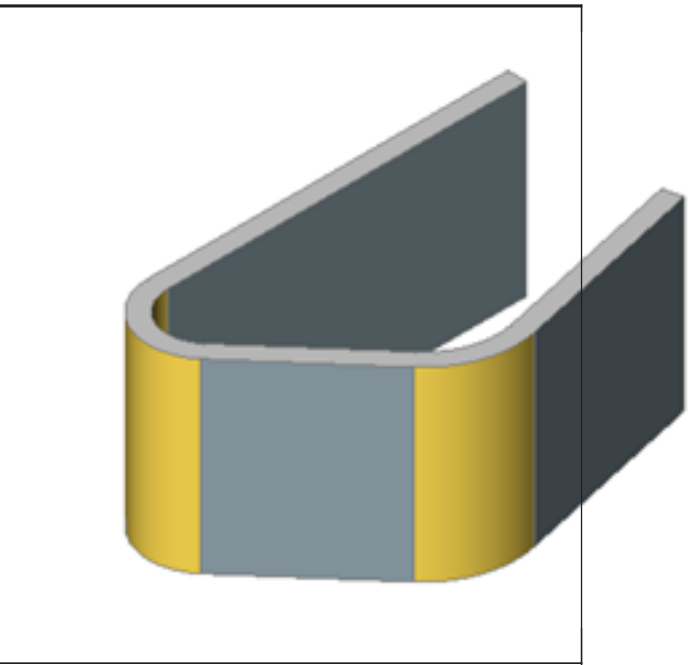
- Clique a ferramenta **Reparar Sheet Metal** () no painel Reparar da aba Sheet Metal na Faixa de opções;
- Clique no botão **Reparar Sheet Metal** () na barra de ferramentas **Sheet Metal**;
- Escolha **Reparar Sheet Metal** no menu **Sheet Metal**;
- Digite *SMREPAIR* na barra de Comando.

  Você é solicitado: Selecione a face da flange ou a face de dobra em loft a reparar ou [Trocar para reparo Local/Pesquisar por detalhes perdidos]:

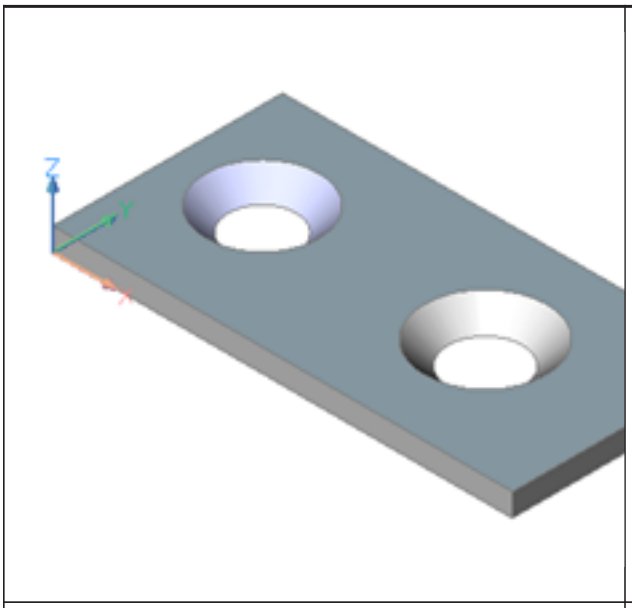
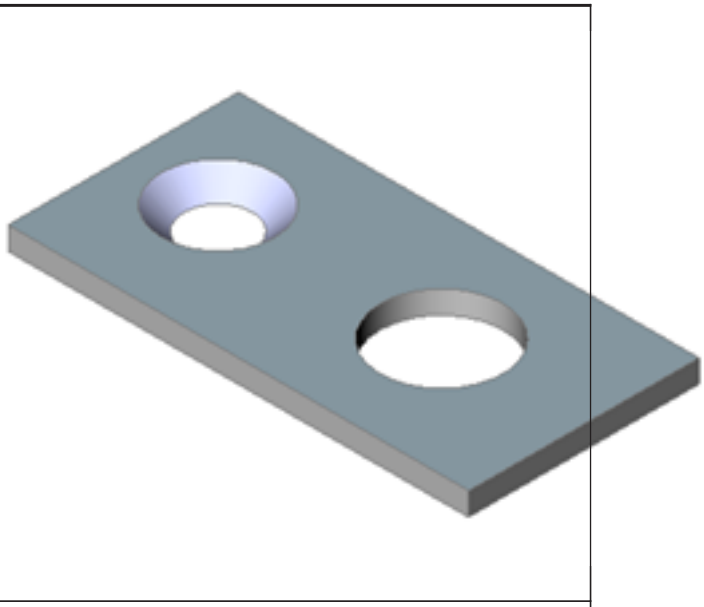
2 Selecione a face a ser reparada.

	
O mesmo modelo antes...	... e depois da operação de reparo:

Quando o modelo tem dobras erradas, o *SMREPAIR* os transforma em dobras regulares:

	
Dobras erradas, com conexão não-tangencial com flanges.	Dobras corretas após reparação.

Recursos de forma são convertidos para o novo modelo reparado:

	
Modelo antes de reparar.	O recurso de forma não mudou, enquanto um furo, que não é recurso de forma, é ortogonalizado.

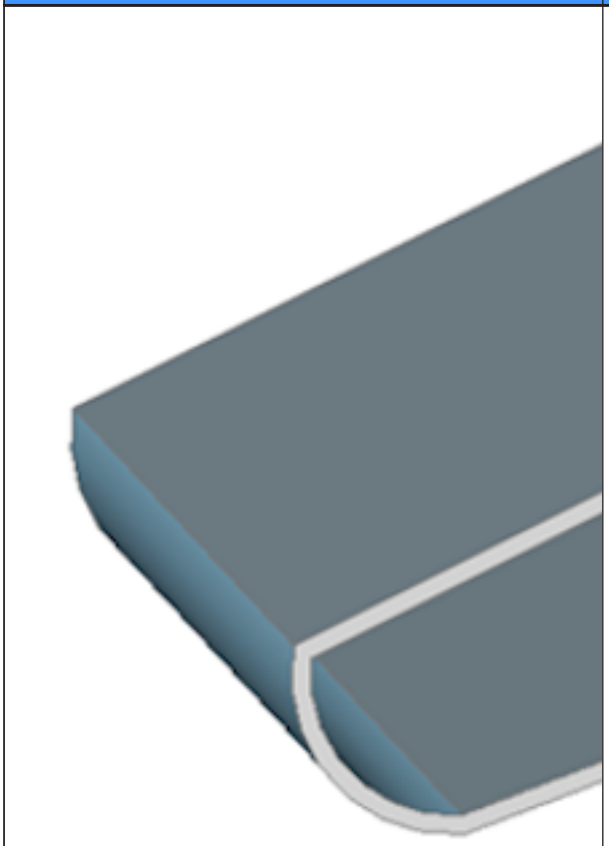
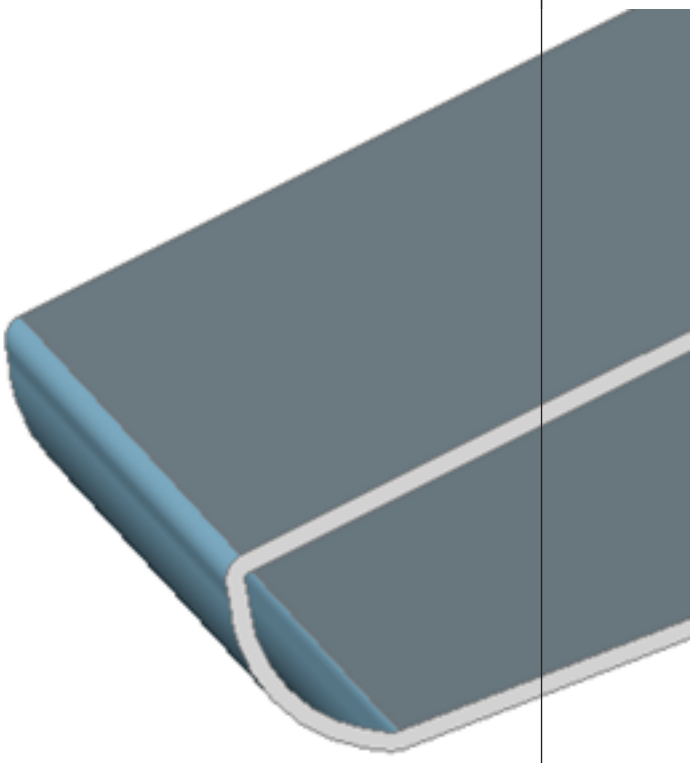
21.5.21.4 Para Reparar um corpo de Sheet Metal com dobras em loft

1 Siga um dos procedimentos:

- Clique a ferramenta **Reparar Sheet Metal** ( ) no painel **Reparar** da aba **Sheet Metal** na Faixa de opções;
- Clique no botão **Reparar Sheet Metal** ( ) na barra de ferramentas **Sheet Metal**.
- Escolha **Reparar Sheet Metal** no menu **Sheet Metal**.
- Digite *SMREPAIR* na barra de Comando.

  Você é solicitado: Selecione a face da flange ou a face de dobra em loft a reparar ou [Trocar para reparo Local/Pesquisar por detalhes perdidos]:

2 Selecione uma face de um flange ou de uma dobra em loft. A face selecionada fornece a superfície a ser engrossada.

Antes	Após
	
A dobra em loft não está tangente à flange. Desdobrar é impossível.	Observe a diferença na conexão da dobra em loft azul com a flange superior, que é suave agora. Desdobrar é possível.

Opção: Se a preferência de usuário **SmRepairLoftedBendMerge** estiver Ligada, as dobras em loft adjacentes com uma conexão tangencial são mescladas para uma única dobra em loft.

21.5.22 Desdobrar Peças de Sheet Metal e Exportar para CAM

21.5.22.1 Visão Geral

Usar o comando *SMUNFOLD*.

Peças de Sheet Metal podem ser projetadas de muitas maneiras diferentes. Para fabricar a peça de chapa metálica é necessário um padrão planificado, com anotações de dobra, que é então usado em um sistema CAM para programar máquinas CNC, como cortadores a laser e prensas dobradeiras. Esta operação de planificação é chamada de desdobramento.

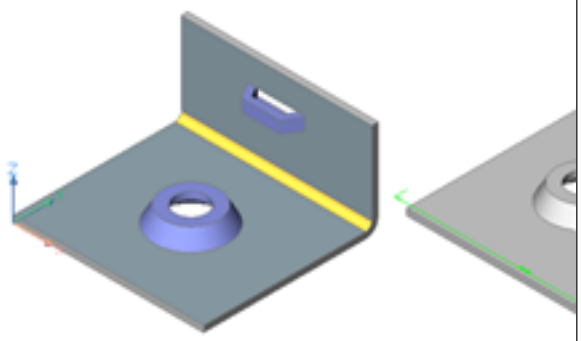
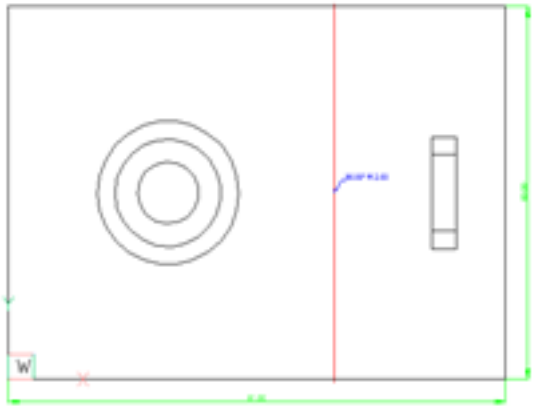
21.5.22.2 Desdobrar um corpo de Sheet Metal

O comando *SMUNFOLD* desdobra peças de Sheet Metal. O BricsCAD cria automaticamente um corpo sólido correspondente à folha de metal planificada necessária para fabricar a peça de chapa metálica, usando técnicas de dobra. Esta folha é colocada no plano-XY e orientada ao longo dos eixos de coordenadas da mesma forma como é a orientação do corpo inicial no espaço 3D. Para alterar a orientação do desdobramento no eixo-Z, execute o comando *SMUNFOLD* novamente e selecione a face oposta do corpo inicial.

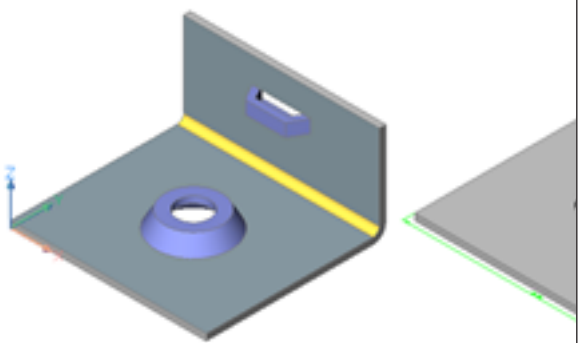
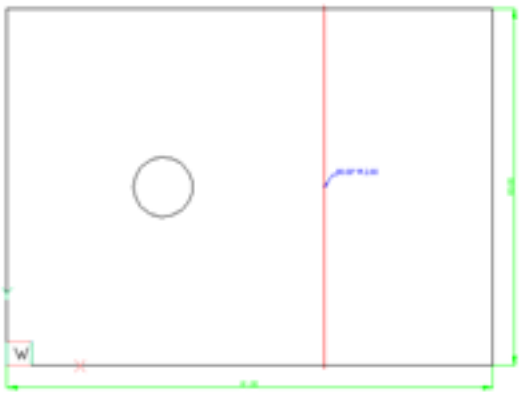
Os recursos do forma são desdobrados de acordo com a configuração **Modo de Desdobramento** no **Navegador de Mecânica**.

As opções são:

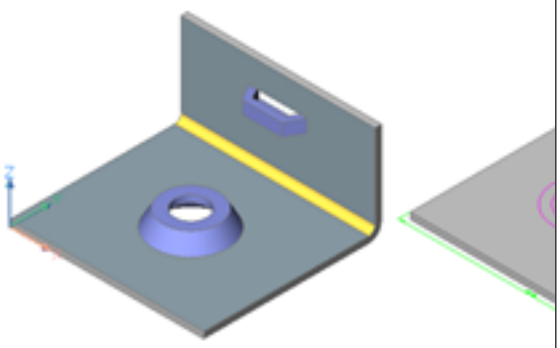
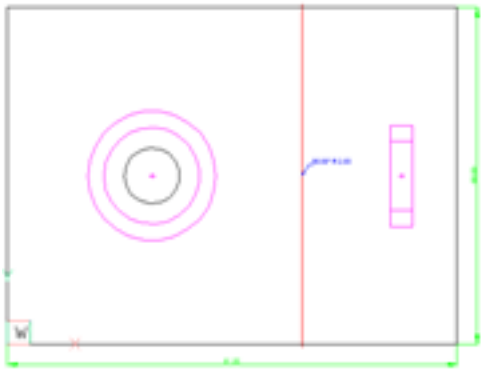
- **Manter** (0): permite manter a geometria dos recursos de forma na representação desdobrada.

Modelo 3D e corpo desdobrado	2D
	

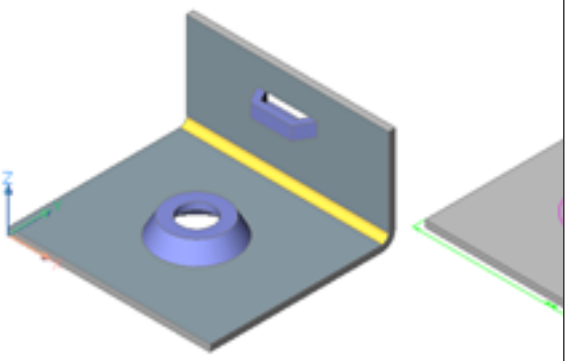
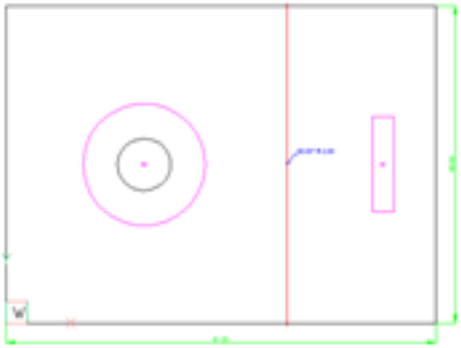
- **Remover** (1): remove a geometria do recurso de forma na representação desdobrada.

Modelo 3D e corpo desdobrado	2D
	

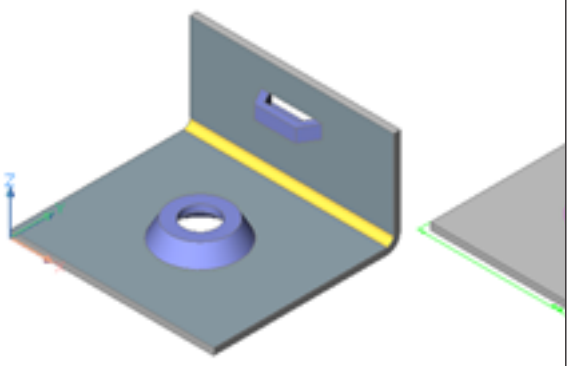
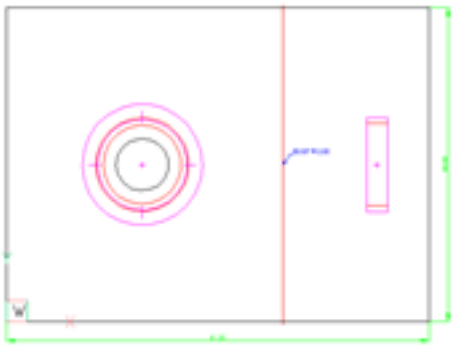
- **Projetar** (2): Gera um símbolo de recurso de forma, projetando as bordas do recurso de forma no plano da flange.

Modelo 3D e corpo desdobrado	2D
	

- **Contorno** (3): Gera um símbolo do recurso de forma ao projetar o contorno externo e furos do recurso de forma no plano da flange.

Modelo 3D e corpo desdobrado	2D
	

- **Símbolo** (4): Cria o símbolo desdobrado na camada **UNFOLD_SYMBOL_UP** ou **UNFOLD_SYMBOL_DOWN**, dependendo da direção do recurso de forma.

Modelo 3D e corpo desdobrado	2D
	

O modo de desdobramento para a atual peça de chapa metálica pode ser definido no **Navegador de Mecânica**.

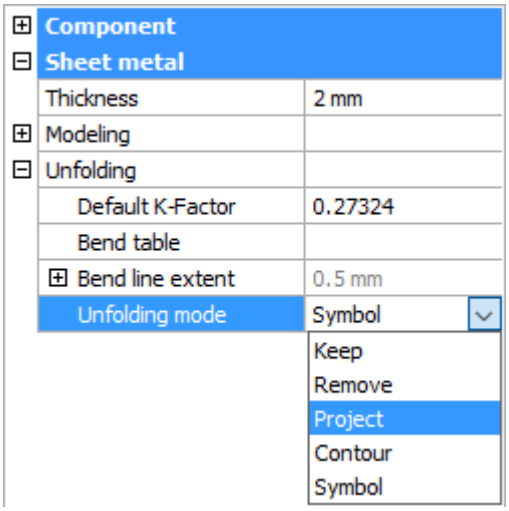


Figura 149: Modo de desdobramento

As dobras em loft que se encontram em superfícies cilíndricas e cônicas podem ser desdobradas analiticamente: as dimensões e a forma da dobra em loft desdobrada serão como se sua superfície média fosse desenrolada matematicamente no plano. Esta forma também pode ser alcançada por um número grandemente aumentado de amostras para a curva em loft. Você deve definir o **modo de desdobramento** individualmente para cada recurso de dobra em loft. Se o desdobramento analítico for permitido, a opção estará presente na lista suspensa:

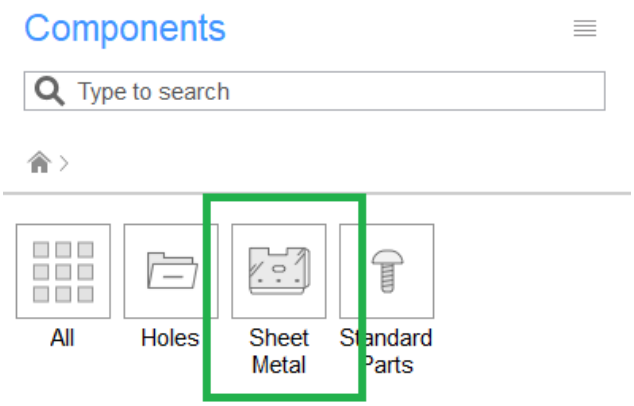


Figura 150: Analítico

21.5.22.3 Para desdobrar um corpo de Sheet Metal

1 Lance o comando *SMUNFOLD*.

Você será solicitado: Selecione uma flange ou face de dobra em loft, para começar a desdobrar [Associativa]:

2 Selecione uma face na peça de chapa metálica.

Você é solicitado: Selecione a posição do corpo desdobrado:

3 Especificar um ponto.

Você é solicitado: Valide o corpo desdobrado e selecione uma opção [Salvar geometria 2D/Salvar geometria 3D/Otimizar anotações de dobra/Manter] <Manter>

4 Siga um dos procedimentos:

- Salve o sólido 3D desdobrado em um arquivo de desenho separado.
- Converta o sólido 3D desdobrado em um arquivo de desenho 2D (*.DWG ou *.DXF).
- Reorganize as anotações de dobra.
- Mantenha o bloco com a peça desdobrada no modelo de peça em Sheet Metal.

O comando *SMUNFOLD* considera a deformação do material de chapa metálica a ocorrer durante a dobra. Quando uma chapa metálica plana é dobrada em uma peça 3D com uma ferramenta de dobra (ex., uma prensa dobradeira), o material é plasticamente deformado, o que significa que é comprimido no interior da dobra e esticado no exterior dessa mesma dobra. Como resultado, o comprimento da peça medida ao longo de sua superfície é diferente nos estados plano e dobrado. BricsCAD é capaz de calcular automaticamente o comprimento desdobrado adequado de sua peça, tomando como base as propriedades de deformação do material. Essas propriedades podem ser definidas ajustando o valor do parâmetro **Fator-K**, ou anexando uma Tabela de Dobras.

Tutorial: Retrabalhar um modelo importado no BricsCAD Sheet Metal

21.5.22.4 Gerenciar propriedades de camada da geometria 2D

A geometria 2D gerada pelos comandos *SMUNFOLD*, *SMEXPORT2D* e *SMASSEMBLYEXPORT* tem cores diferentes para contornos, linhas de dobra para cima, linhas de dobra para baixo, dimensões gerais, recursos de forma para cima, recursos de forma para baixo, anotações de dobra, e atributos.

Você pode definir cores de forma diferente no grupo de definições **Sheet Metal/Comandos/Camadas para desdobramento** na caixa de diálogo *Configurações*. Cada tipo de entidade é colocada em uma camada dedicada, na qual a cor é controlada através de uma série de preferências do usuário:

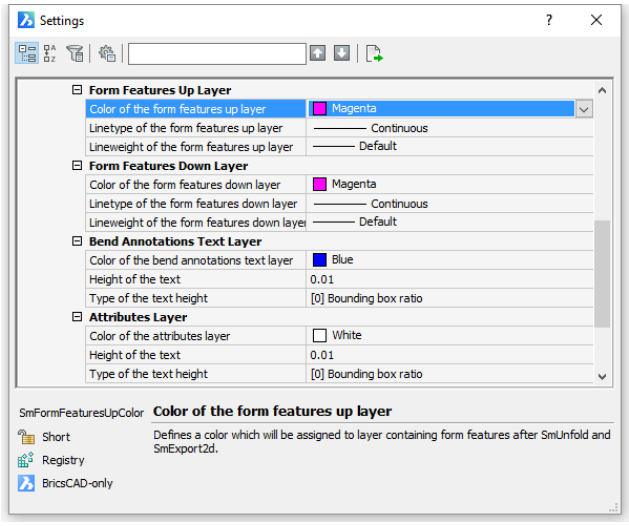


Figura 151: conFigurações

Se a geometria 2D de um recurso de forma foi retirada de um componente de biblioteca, suas propriedades são controladas pelas propriedades deste componente, no momento da inserção. Existem outros dois modos do adotar o **Modo de Desdobramento** que levam à geração de geometria 2D para recursos de forma: **Projeto** ou **Contorno**. Nestes modos, a geometria 2D é gerada automaticamente a partir da geometria 3D e sua aparência é controlada pelos grupos de configurações **Camada Recursos de Forma Acima** e **Camada Recursos de Forma Abaixo**.

21.5.22.5 Para exportar um corpo de Sheet Metal

Peças de Sheet Metal criadas ou editadas no BricsCAD podem ser processadas por diferentes sistemas CAM (como JETCAM ou CADMAN- B) para gerar código NC para máquinas de corte e dobra.

A preferência do usuário controla a opção **Salvar geometria 2D** do comando *SMUNFOLD* para personalizar o arquivo *.dxf, para este ser compatível com a lista de populares sistemas CAM.

O comando *SMEXPORT2D* exporta a representação desdobrada de um corpo de Sheet Metal como um perfil 2D, em formato de arquivo *.dxf ou *.dwg.

O comando *SMEXPORTOSM* exporta um sólido de chapa metálica para o formato de arquivo *.osm (Open Sheet Metal) - nativo para o sistema de CAM CADMAN-B.

21.5.22.6 Para exportar uma montagem com peças de Sheet Metal para .dxf

O comando *SMASSEMBLYEXPORT* pesquisa na árvore de montagem para o projeto de Sheet Metal e o exporta para um arquivo *.DXF. Como resultado, um conjunto de arquivos *.DXF, ordenados por espessura em diferentes subpastas, é criado na pasta de saída. O comando gera um relatório HTML que permite controlar a saída. A maioria das peças de chapa metálica será processada automaticamente. As peças de chapa metálica com design pobre serão listadas pelo comando para ser retrabalhadas manualmente.

21.5.23 Representações Desdobradas Associativas de Sólidos 3D

21.5.23.1 Visão Geral

O BricsCAD oferece associatividade bidirecional entre a peça 3D Sheet Metal e sua representação planificada. Neste modo, ambas representações existem em documentos separados: a Peça original é armazenada em seu documento nativo e a Peça planificada é armazenada em um documento temporário. Atenção: A associatividade é temporária, e persiste enquanto ambos os documentos estiverem abertos. Para voltar a trabalhar no 'modo de associatividade' numa próxima sessão do BricsCAD ou quando o documento estiver fechado, deve-se repetir novamente os passos.

21.5.23.2 O que significa associatividade?

O Desdobramento Associativo permite que você faça cortes na geometria planificada, e propague as alterações de volta ao modelo 3D original. Isso ajuda a simplificar 2 cenários:

- Criar Dobras complexas, não-padronizadas, e cortes de Alívio de Canto.
- Evitar auto-interseções (colisões) na chapa desdobrada.

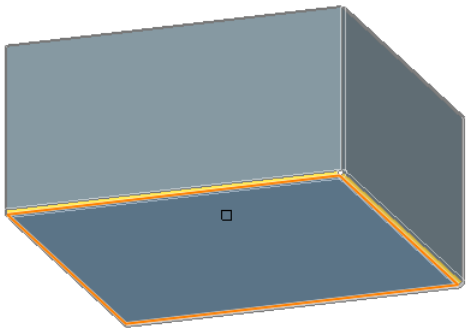
Nestes dois cenários, ambos são mais fáceis de fazer em 2D do que em 3D. No modo planificado, a peça de Sheet Metal é representada como um conjunto de sólidos planos separados (por-recurso). Use o comando *SMDELETE* para excluir flanges de sólido e dobras em padrões planificados.

A associatividade é bidirecional; alterações podem ser propagadas a partir das peças planificadas, de volta aos modelos 3D, e também no sentido contrário. Alterações feitas em uma das representações são propagadas para o documento com a outra representação, assim que esta se torna ativa..

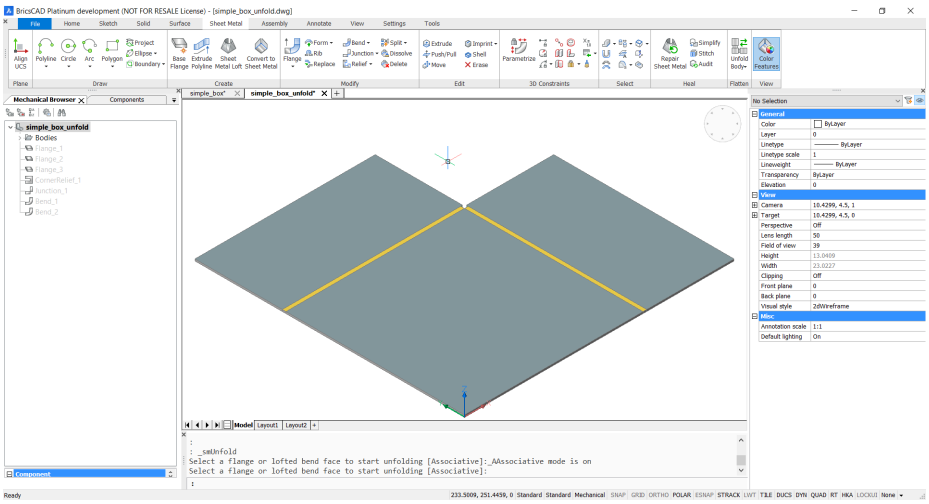
21.5.23.3 Trocar para o Modo de Associatividade

O modo de associatividade é ativado usando o comando *SMUNFOLD*. Certifique-se de que não há faces selecionadas.

Na seção **Sheet Metal** da Faixa de opções, selecione o ícone Flatten > **Desdobrar Associativamente Corpo** () e selecione a face da **Flange**.



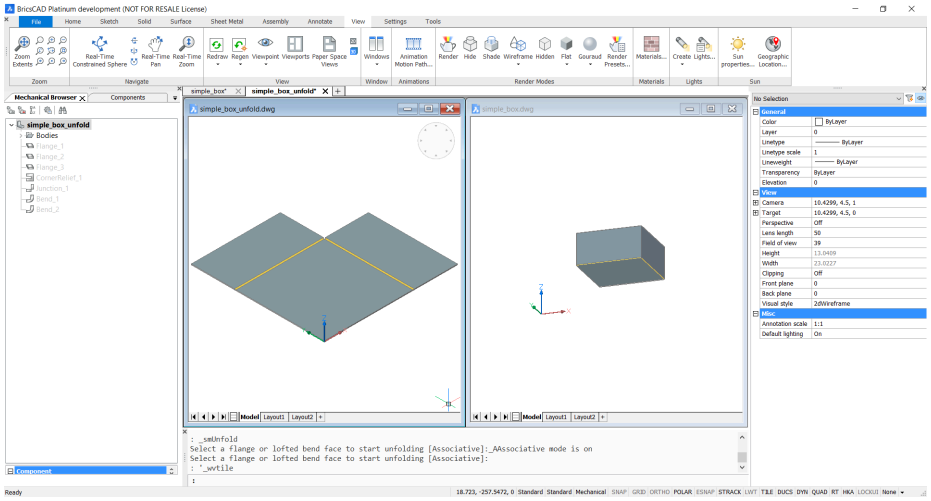
A aplicação vai calcular a planificação daquela peça, e mudará para o modo associativo. A condição de sucesso de uma determinada operação é a mesma que para o desdobramento regular. Continue trabalhando no modo de associatividade somente se nenhum erro ou avisos aparecerem durante a operação. Um novo documento temporário contendo a peça planificada será criado.



Nota: Não há necessidade de salvar este documento. Ele é usado para propagar operações de edição do sólido planificado de volta para o modelo 3D.

Ajuste as janelas **Lado a Lado Vertical** para visualizar tanto o modelo 3D como os documentos associativos desdobrados.

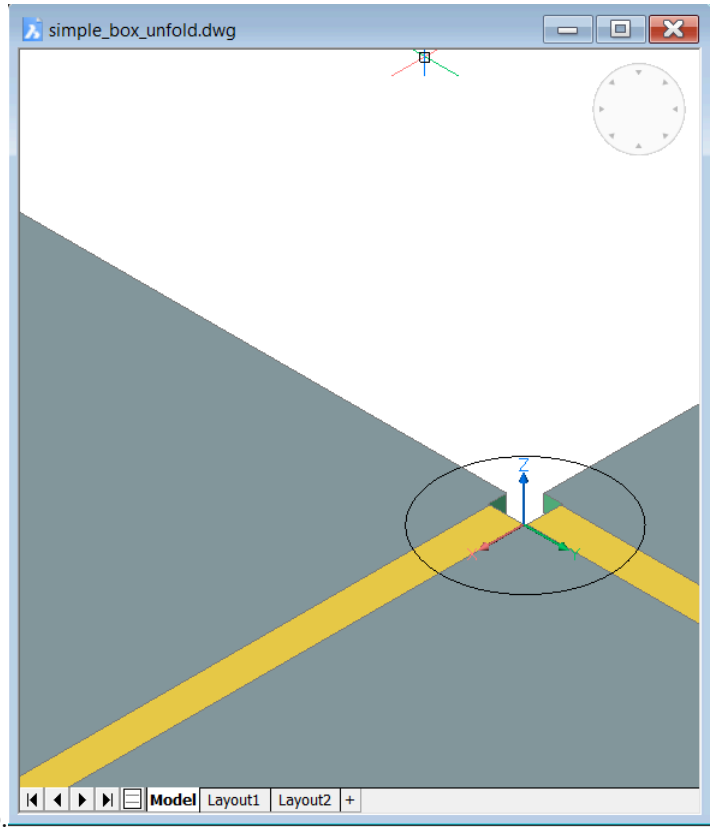
Clique no botão **Lado a Lado Vertical** na Barra de ferramentas ou Faixa de opções.



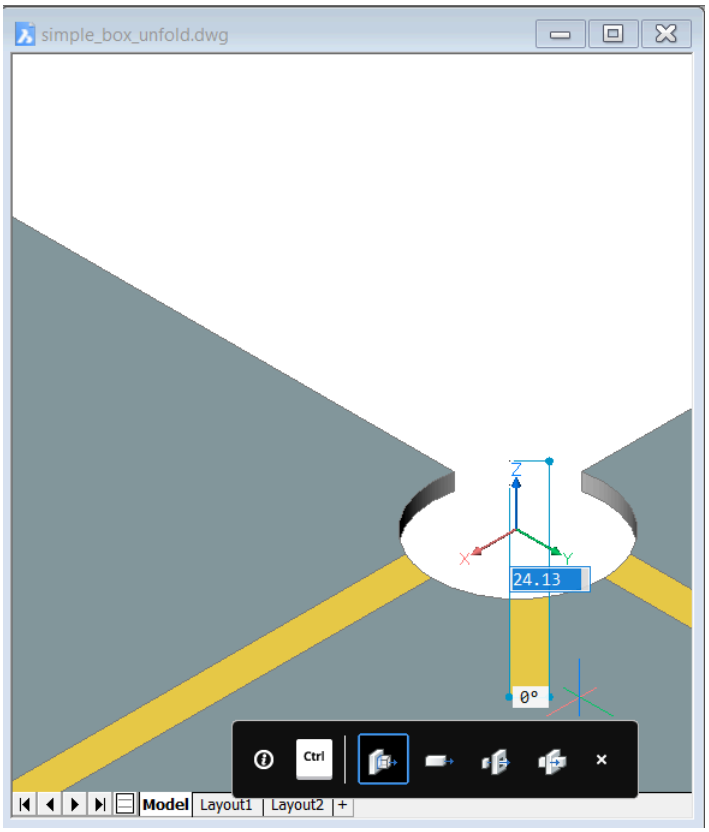
Nota: No **Navegador de Mecânica**, quando a **geometria planificada** é selecionada, esta lista os recursos que estão desativados.

21.5.23.4 Editar a Geometria

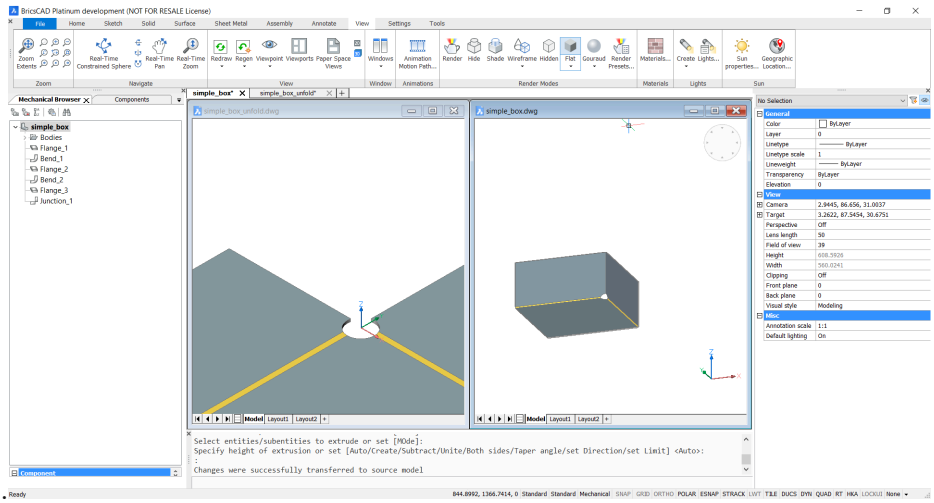
1 Trabalhando com documento de projeto planificados, desenhe um círculo no canto.



2 Aplique o comando *DMEXTRUDE* ao círculo, cortando o volume.



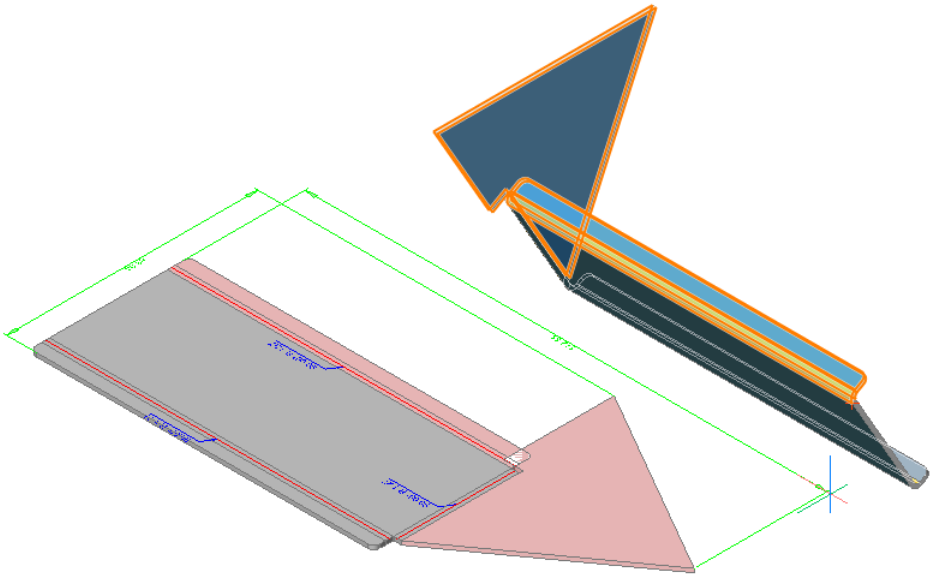
3 Clique no documento original novamente, para ativá-lo.



4 O modelo será atualizado e o alívio de canto personalizado será exibido. A aplicação escreve na linha de Comando: Alterações foram transferidas com sucesso para o modelo de origem.

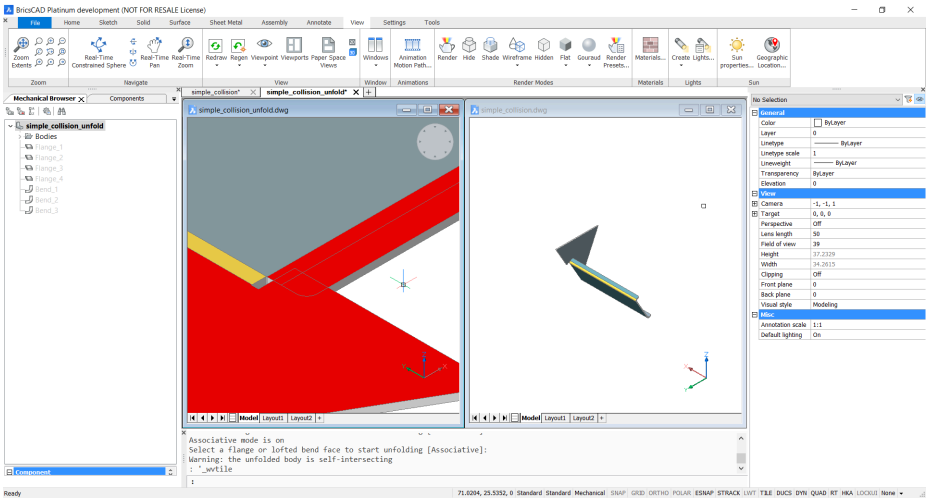
21.5.23.5 Evitar Colisões no Modelo Planificado

Outra aplicação útil para o desdobramento associativo é corrigir as colisões de materiais, que não são tão óbvias.

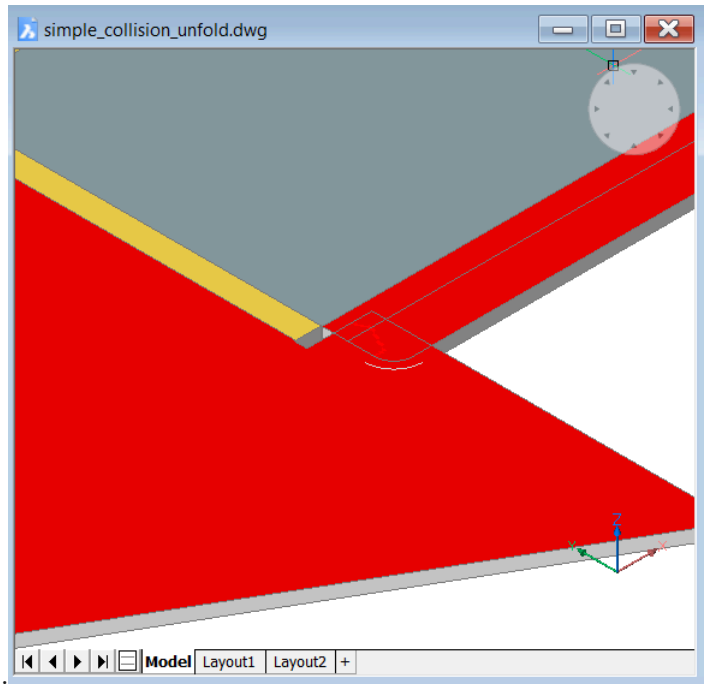


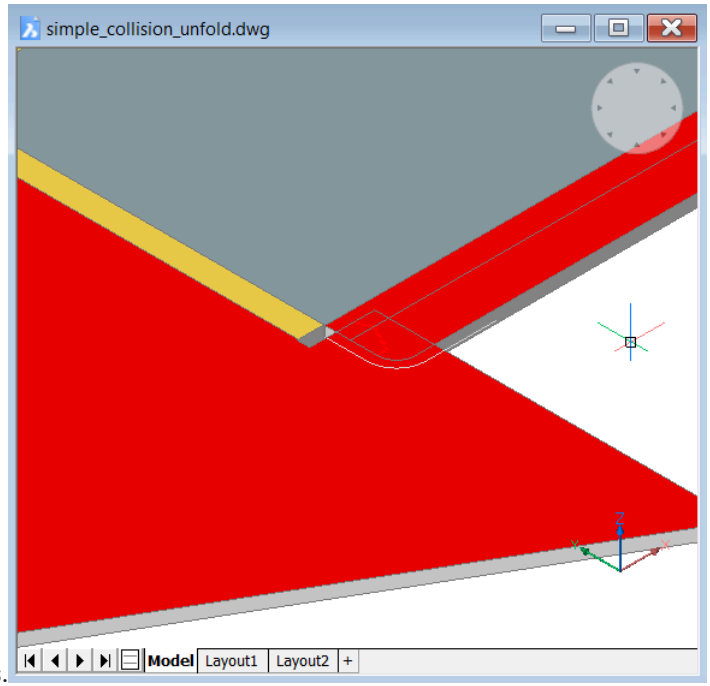
Neste exemplo, não sabemos quais alterações devem ser aplicadas ao modelo 3D, para evitar a colisão, quando planificadas. Vamos retrabalhar o modelo com o Desdobramento Associativo.

1 Na seção **Sheet Metal** da Faixa, no painel Flatten > selecione o ícone **Desdobrar Associativamente Corpo** (), selecione a face da **Flange**, desdobre e clique no botão **Lado a Lado Vertical** () na Barra de ferramentas ou Faixa de opções, para posicionar lado a lado os documentos abertos e amplie o Zoom na área de colisão.

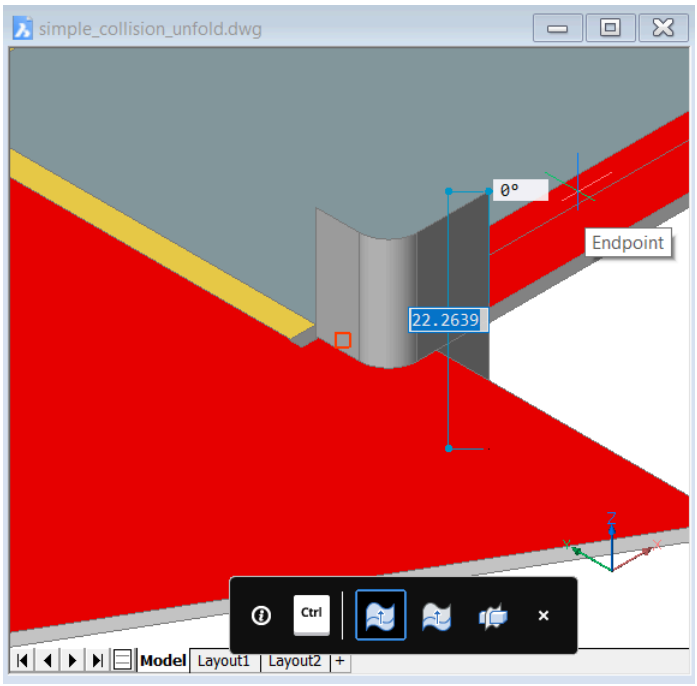


2 Selecione a borda do limite circular e desloque este em

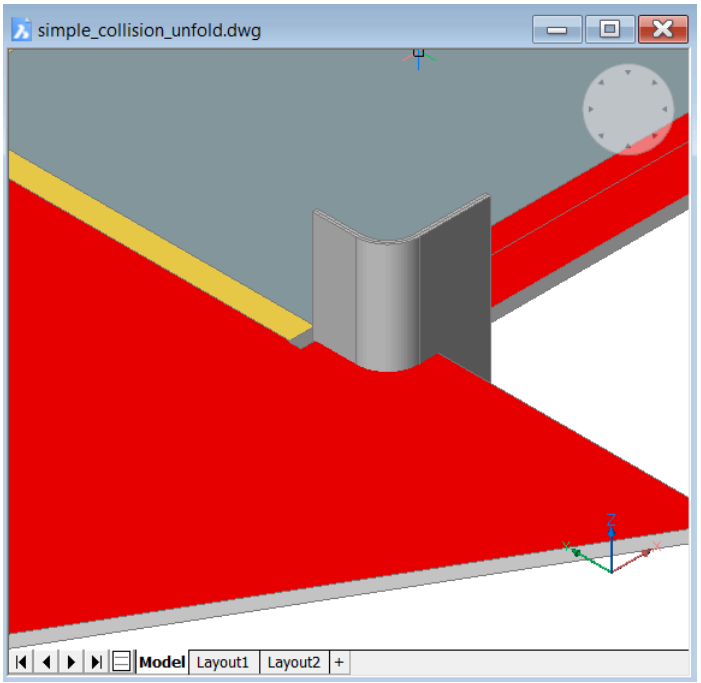
1. 

3 Desenhe linhas retas extras adjacentes à borda, e junte-as. 

4 *DMEXTRUDE* a polilinha em ambos os lados.

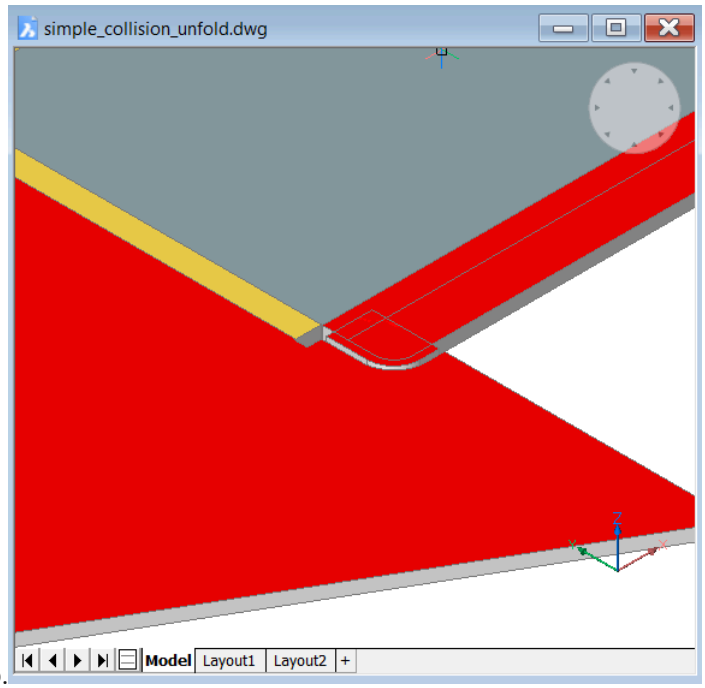


5 *DMTHICKEN* a superfície por um valor maior.

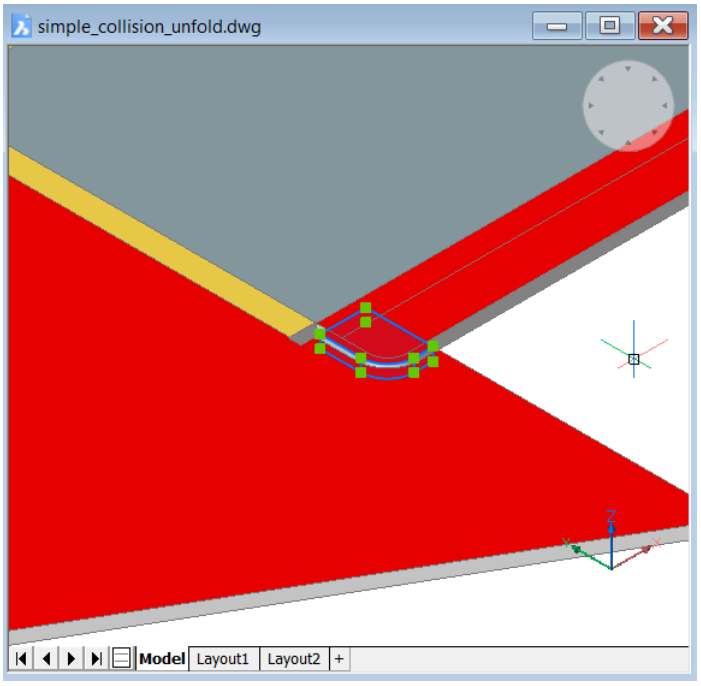


6 *SUBTRAIR* o corpo espessado a partir do padrão

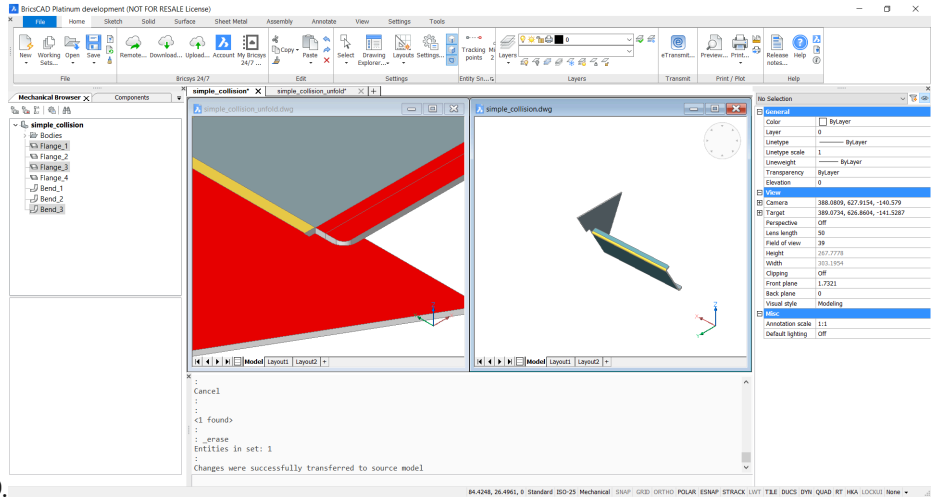
desdobrado.



7 *APAGAR* as peças restantes.



8 Volte para o documento que contém o modelo

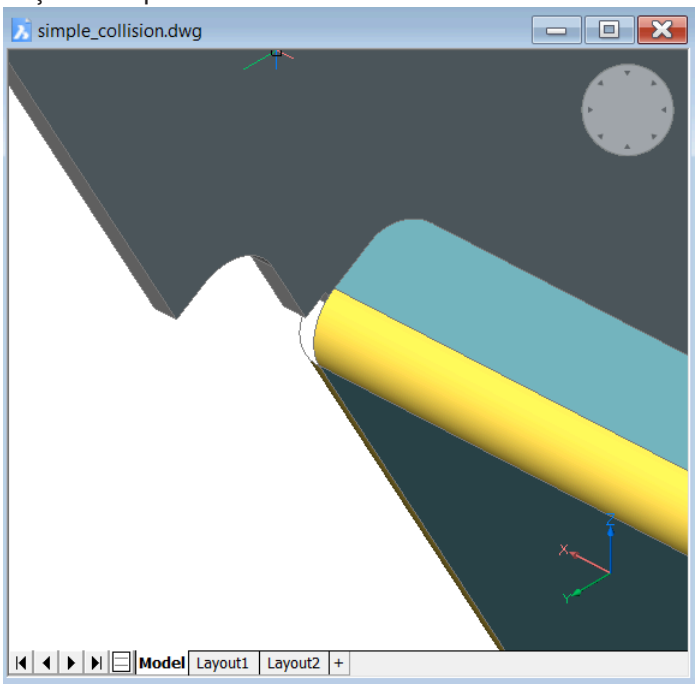


3D.

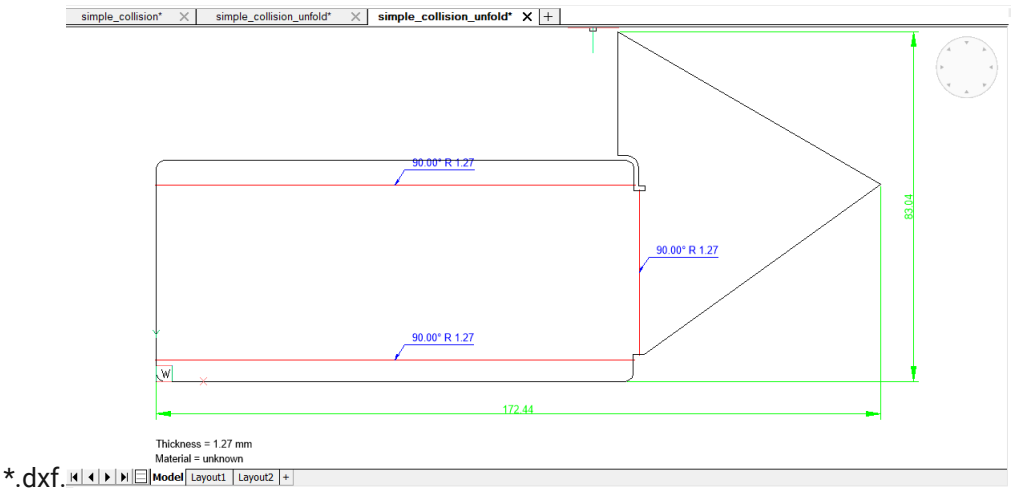
9 Certifique-se de que as alterações foram propagadas.

Alterações foram transferidas com sucesso para o modelo de origem.

Faça Zoom para o canto atualizado.



10 Para testar o desdobramento 2D, exporte o arquivo no formato



*.dxf.

21.5.24 Recursos inteligentes

21.5.24.1 Visão Geral

A tecnologia Smart Features (Recursos inteligentes) gerencia a atualização inteligente local da geometria após executar comandos de Sheet Metal. O comportamento é controlado através da preferência de usuário **SMSmartFeatures** .



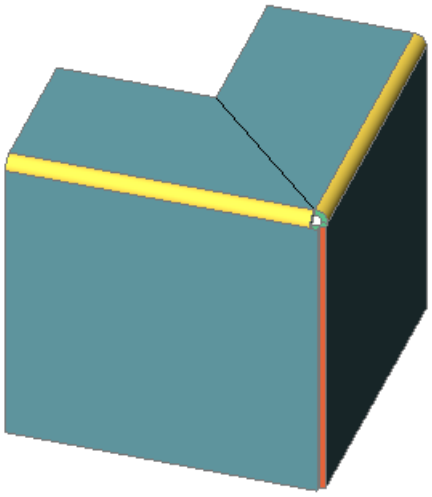
Há 3 opções disponíveis:

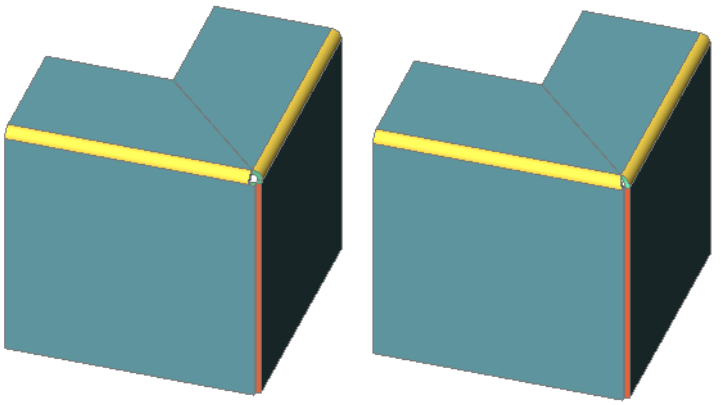
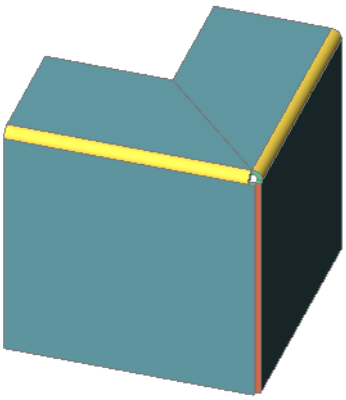
- Permitir a reconstrução de recursos de chapa metálica:
  - Alívio de canto é transformado em 2 alívio de dobra após a execução de *SMSPLIT* em uma flange.
  - Alívios desnecessários são removidos.
- Permitir estamparia automática de bordas, após a reconstrução: estamparia automática de bordas para faces de espessura.
- Permitir criação automática de junções após criar dobras.

Nota: Sempre mantenha Ligadas as opções de Recursos inteligentes (Smart Features), a menos que ocorra um comportamento inconsistente.

21.5.24.2 Transformação de um alívio de canto em alívios de dobra

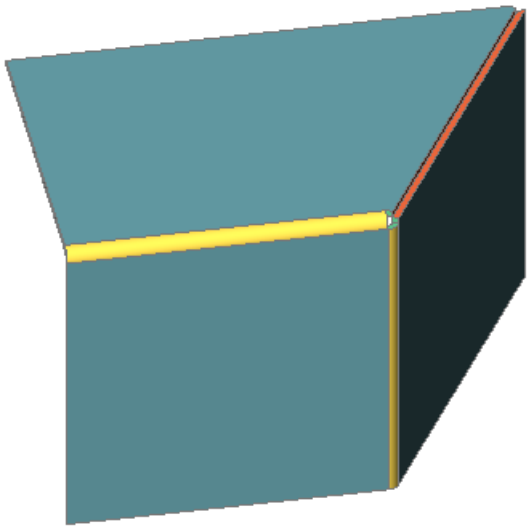
Ao dividir uma flange superior por uma linha, em 2 partes separadas, o alívio de canto é transformado em 2 alívios de dobra.



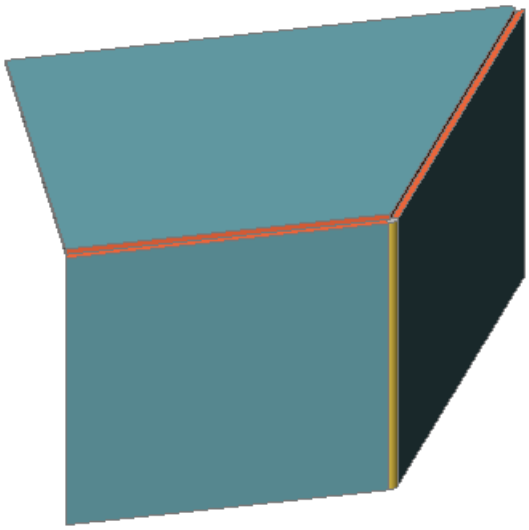
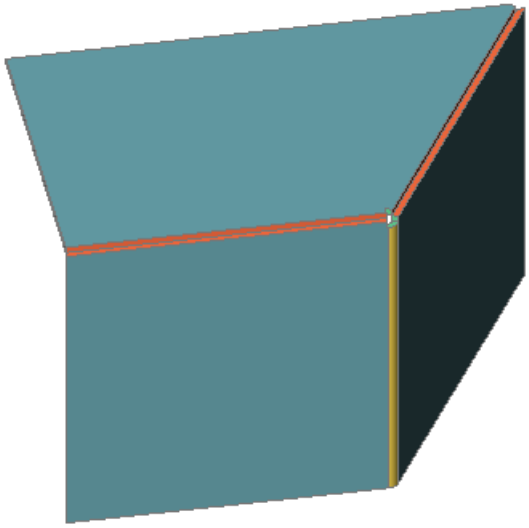
	
Recursos Inteligentes criam alívios de dobra automaticamente, que podem ser alternados para suave.	Sem Recursos Inteligentes, o alívio de canto é mantido, e não pode ser trocado.

21.5.24.3 Remover cortes desnecessários

No modelo abaixo, existe um corte em alívio de canto, o que é necessário para evitar que o material seja esticado.



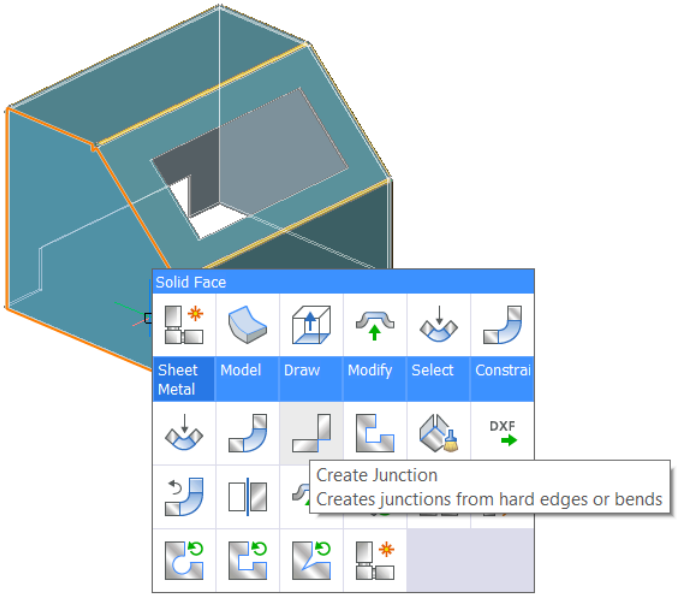
Quando a dobra superior é convertida em uma junção, chegamos a peças separadas. Os recursos inteligentes removem automaticamente o alívio agora desnecessário.

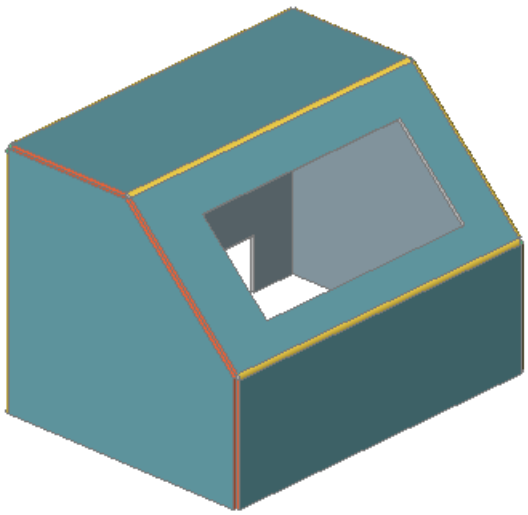
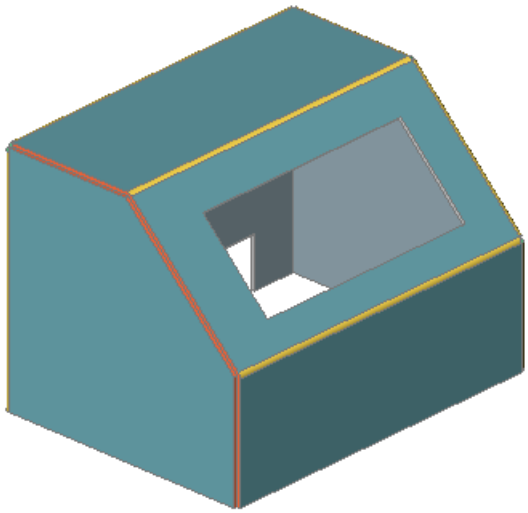
	
Recursos Inteligentes preenchem automaticamente cortes desnecessários, simplificando assim a geometria.	Sem recursos Inteligentes, o alívio de canto ainda existe.

21.5.24.4 Estamparia automática de bordas

Bordas estampadas nas faces de espessura as separam em áreas que podem ser retrabalhadas automaticamente. Isso significa que junções localizadas podem ser convertidas em dobras. Sem bordas automaticamente estampadas, o comando *SMIMPRINT* é necessário aqui.

No exemplo abaixo, vamos separar uma flange usando junções. Ligamos a **reconstrução de recursos de chapa metálica**, e depois usamos a **Estamparia automática de bordas após a reconstrução**.

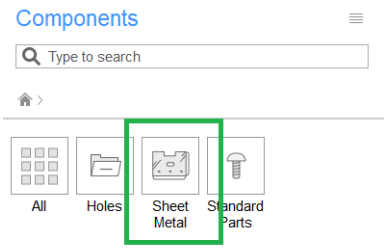


	
Permitir estamparia automática de bordas após reconstrução = Liga. As faces de junção são divididas corretamente.	Permitir estamparia automática de bordas após reconstrução = Desl. Os recursos de junção se sobrepõem.

21.5.24.5 Criação automática de junções

Em certas situações, quando as bordas rígidas se encontram no canto e nós criamos uma dobra em tudo menos um, que tem que se tornar uma junção para tornar a peça desdobrável.

Suponha que criamos dobras na face superior de uma caixa de chapa metálica:



Então as bordas rígidas restantes, adjacentes à face, têm que se tornar junções. Quando **SMSmartFeatures** = Liga, isto é executado automaticamente:



Para ser preciso, a área de criação de junção automática está localizada no vértice comum das bordas rígidas (ou alívio de canto quando as dobras são criadas). A opção não otimiza o layout da peça desdobrada.

21.6 Desenhos Gerados

21.6.1 Vistas do Desenho

21.6.1.1 Visão Geral

BricsCAD Mechanical oferece um conjunto de ferramentas para gerar vistas de desenho a partir de modelos em 3D de peças e de montagens. As vistas de desenho são criadas no Paper Space do mesmo arquivo DWG. O modelo 3D é criado no Model Space. Vistas de desenho podem ser anotadas usando dimensões, chamadas, texto e tabelas. Vistas e anotações são atualizadas quando o modelo 3D associativo é alterado.

As ferramentas para gerar, atualizar e anotar vistas de desenho estão localizadas na aba **Anotar** da **Área de Trabalho Mecânica**:



Figura 152: Anotar

Tabela 45: Anotar

Vista do Desenho	Descrição
Vistas Base	Gera vistas associativas, isométricas e ortográficas padrão, de um modelo sólido 3D, em um layout do Paper Space.
Vistas Projetadas	Gera vistas projetadas adicionais a partir de uma vista de desenho existente.
Vista Corte	Cria uma vista de corte transversal com base em uma vista de desenho padrão, gerada pelo comando *VISTABASE* em um layout de Paper Space.
Vista Detalhe	Cria uma vista de detalhe de uma parte do desenho gerado por padrão em uma escala maior.
Editar Vista	Altera a escala e a visibilidade da linha oculta das vistas do desenho.
Atualizar Vistas	Atualiza uma seleção de vistas do desenho obtidas por *VISTABASE* e *VISTACORTE* quando *VIEWUPDATEAUTO = 0*.
Estilo Corte	Estilos predefinidos e definidos pelo usuário.
Estilo de Detalhe	
Exportar Vistas	Exporta o conteúdo das vistas do desenho obtidas por *VISTABASE* e *VISTACORTE* para o Model Space do desenho, ou para um novo desenho.

21.6.1.2 Vistas do desenho em qualidade-rascunho

Por padrão, o BricsCAD gera visualizações de desenho precisas (de alta qualidade) para qualquer modelo 3D. As bordas de tais vistas (incluindo bordas de silhueta) são representadas por entidades 2D; linhas, círculos, arcos e splines. A geração de vistas precisas para um grande número de entidades 3D pode levar uma quantidade significativa de tempo. Isso ocorre porque cada borda da silhueta deve ser aproximada com uma curva analítica 2D.

Você pode reduzir o tempo necessário para gerar vistas de desenho, alterando a configuração *DRAWINGVIEWQUALITY* para *0*. Isso permite vistas de desenho com qualidade de rascunho. Tais vistas de desenho são representadas internamente como sólidos 3D, o que não permite colocar anotações em suas bordas de silhueta. No entanto, elas parecem muito semelhantes a uma vista de desenho precisa (de alta qualidade) e você pode usá-las para criar rapidamente os layouts, economizando tempo.

Uma vista de desenho com qualidade de rascunho pode ser mudada para uma vista de desenho de alta qualidade usando sua propriedade *Geometry* (altere de *Rascunho* para *2D*).

21.6.2 Anotações do Desenho

Criar e editar anotações, revise o artigo Visão Geral das Ferramentas de Dimensionamento.

As dimensões e o Multi-chamadas que estiverem na vista do desenho são atualizados após a alteração do modelo 3D subjacente.

Os desenhos de Montagem podem ser anotados com balões BOM. Cada balão se refere a uma linha numa tabela BOM gerada a partir da montagem.

A tabela BOM pode ser colocada em Paper Space (layouts) com a vista de desenho de uma montagem.

Representações explodidas de montagens podem ser usadas para gerar vistas do desenho e anotá-las com balões.

22. Communicator for BricsCAD

22.1 BricsCAD Communicator

BricsCAD Communicator is an optional plug-in for Pro and higher license levels of BricsCAD that imports and exports 3D data between major CAD formats. Its high-quality conversions facilitate the re-use of 3D CAD data to improve productivity and reduce project costs.

22.1.1 Supported export file formats

Windows

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **CATIA v5:** .CATPart, .CATProduct
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vd
- **XCGM:** .xcgm

Linux

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

Mac

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

Nota: When the EXPORTPRODUCTSTRUCTURE system variable is set, you can export a BricsCAD assembly with mechanical components as a file with product structure.

22.1.2 Supported import file formats

Windows

- **ACIS:** .asab, .asat, .sab, .sat
- **Autodesk Inventor:** .iam, .ipt
- **CATIA v4:**.model
- **CATIA v5/v6:**.CATPart, .CATProduct, .CGR
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **NX:** .prt
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Linux

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Mac

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda

Nota:

Depending on the Communicator settings:

- You can import an assembly file, 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 (Communicator license is required): ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM. 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'. This feature is currently supported on CATIA v5 only.
- You can set import options to automatically repair, stitch and simplify imported geometry.
- The IGES format has its own options to stitch and simplify. 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 an alternate search path. This will be used to look for parts absent in the imported assembly folder.

22.2 BricsCAD Communicator settings

The **Communicator** section in the **Settings** dialog box allows you to customize the behavior of the related commands:

- setting environment options.
- control the import/export of data.
- toggling post-processing operations.

Settings are applicable when the EXPORT and IMPORT commands engage one of Communicator file formats.

22.2.1 Import settings

22.2.2 Product structure

Name: ImportProductStructure

Description: Determines how the imported entities will be structured and whether it will form a soup of entities, a block hierarchy or an assembly structure.

Values:

- **[0] None**: Creates a flat structure without blocks in the modelspace of the target database, regardless of 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 the target database.

- **[2] As mechanical components**: In this mode, data will be translated into BricsCAD assembly data, the structure and its properties (physical materials – BLMATERIALS command). If the imported file has no assembly data, a mechanical component will be created in the BricsCAD target document root anyway.

22.2.3 Translate colors

Name: ImportColors

Description: Determines how the colors of the imported entities will be assigned: either as true colors or as matching the current color palette.

Values:

- **[0] To RGB**: All entity colors will be converted to RGB, regardless of the 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. Otherwise, it is given 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 index color is assigned to the entity.

22.2.4 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.

22.2.5 Perform stitching

Name: ImportStitch

Description: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported.

Nota:

- Stitch operations are time-consuming when importing large files.
- Check the IMPORTIGESSTITCH setting, which can set an override for the IGES file format.

Values: **on**/**off** checkbox.

22.2.6 Repair model on import

Name: ImportRepair

Description: Applies operations, similar to the DMAUDIT 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.

22.2.7 Perform simplification

Name: ImportSimplify

Description: Applies operations similar to the DMSIMPLIFY command:

- Convert imported splines into canonic surfaces.
- Simplify topology (remove imprinted edges) if possible.

Nota: Also check the ImportIgesSimplify setting, which can set an override for the IGES file format.

Values: **on**/**off** checkbox.

22.2.8 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.

22.2.9 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.

22.2.10 IGES options

1 **Perform stitching**

Name: ImportIgesStitch

Description: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported. This setting determines whether a stitch operation has to be applied to imported entities when the format of the source file is IGES.

Nota:

- Stitch operations are time-consuming when importing large files.
- For file formats other than IGES, the ImportStitch setting applies.

Values: **on**/**off** checkbox.

2 **Perform simplification**

Description: Determines whether a DMSIMPLIFY operation has to be applied to the imported geometry when the source file format is IGES

Nota: For other formats, see the ImportSimplify setting.

Values: **on/off** checkbox.

3 **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.

4 **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, the search is performed in the folder of the imported file only.

5 **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.

6 **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.

7 **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.

22.2.11 Export settings

22.2.12 Product structure

Name: ExportProductStructure

Description: Determines whether the assembly 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.

22.2.13 Hidden parts

Name: ExportHiddenParts

Description: Determines if invisible entities are exported or not. Entities can be invisible because of:

- the result of the HIDEOBJECTS command.
- sitting on a hidden layer.
- owned by an invisible 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.

22.2.14 STEP options

STEP export format version

Name: ExportStepFormatVersion

Description: Allows you to select the available specifications of the STEP format.

Values:

- **[0] AP203**: Selects AP203 protocol.
- **[1] AP214**: Selects AP214 protocol.
- **[2] AP242**: Selects AP242 protocol.

23. BricsCAD Shape

23.1 Introduction to BricsCAD Shape

23.1.1 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.

23.1.2 BricsCAD Shape lets you create without limits

Shape's smart modeling tools make it easy to capture in 3D. The multifunctional "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.

23.1.3 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.

23.1.4 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.

23.1.5 Stop sketching. Start shaping!

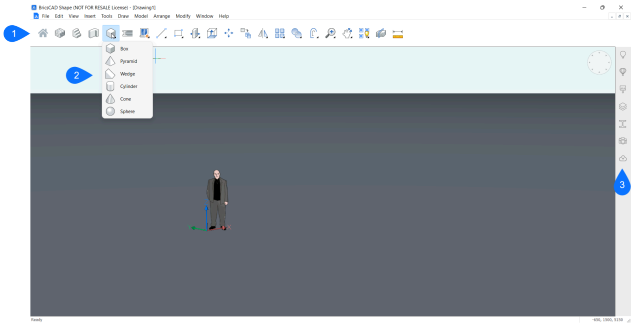
Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "10 Minutes to BricsCAD Shape" on YouTube, Bricsys channel.

23.2 BricsCAD Shape user interface

The toolbar at the top of your screen (1) 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 (2). 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 (3) 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.



Nota: Many of the modeling tools that are used in BricsCAD and BricsCAD BIM are also available in Shape. You can get an overview of all available commands in Shape by typing in COMMANDS in the Command line.

23.3 Basic concepts of modeling in BricsCAD Shape

23.3.1 Selection methods

Highlighting and selecting objects

Entities in BricsCAD Shape are solids, which are made up of edges and faces.

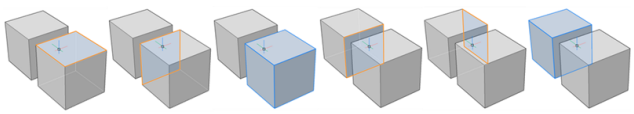
When hovering over an entity, a part of the entity is highlighted. To select the highlighted part of the entity, left-click the edge or face.

Nota: To select or highlight the whole entity, hold down the **CRTL** key and left-click.

For some operations, it's not necessary to select entities. When a surface or edge is highlighted, the **Quad** opens and several commands can be accessed.

Nota: You can only highlight one entity at a time, while it is possible to select multiple entities at once by left-clicking them one-by-one, or using a window selection method.

Sometimes, the entity you want to select is hidden behind other entities. Place the cursor over the obscured entity, then repeatedly press the **TAB** key. All (sub)entities under the cursor highlight one by one. Click to select the currently highlighted (sub)entity.



To clear a selection of objects, press the **ESC** button.

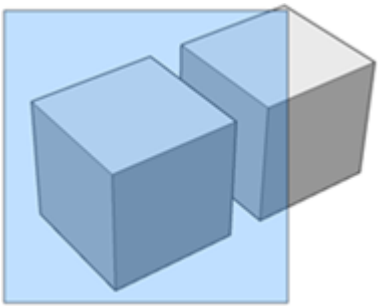
Selection windows

To select one or multiple elements at once, a selection window is used. Left click on an empty space in the model area. When you move the cursor a selection window displays.

Nota: The selection window selects entire solids by default. Press the **Ctrl** key to select faces, press the **Ctrl** key again to select edges and again to select solids. The current selection method is indicated in the **Hot Key Assistant** (HKA) at the bottom of the screen.

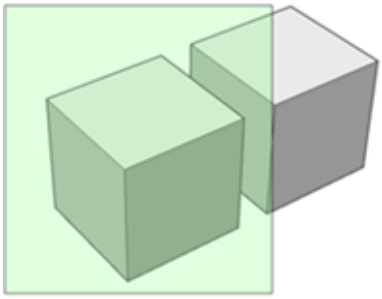
Blue selection window

Moving the cursor from left to right, a blue selection window displays: all entities that are completely inside the rectangle are selected. This selection method is called 'Window Inside'.



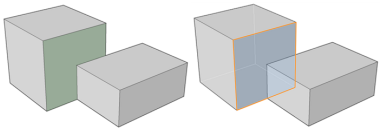
Green selection window

Moving the cursor from right to left, a green selection window displays: all entities that are completely inside or overlapping the selection rectangle are selected. This selection method is called 'Window Overlap'.



Boundaries vs Faces

Boundaries can be defined by a set of 2D entities in the XY field of the current coordinate system or on a 3D solid face or by partially overlapping faces of 3D solids. 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. Boundaries can be used to create new solids or making holes or offsets and recesses.



Nota: To select the entire solid face instead of a boundary, press the **TAB** key.

23.3.2 Quad

The Quad cursor menu, or Quad for short, is an alternative to the Command line, toolbars or ribbon, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. It includes rollover tips, which display a limited set of properties when selecting or hovering over an entity.

The Quad displays when:

- You hover over an entity.
- A selection set exists.
- You right click on an empty space on the screen. This will open the No Selection Quad.

The initial content of the Quad when hovering over an entity is:

- Entity type.
- The icon of the most recently used command on this entity type.
- Relevant properties.

When you move the cursor to the Quad, it expands. On top you can find the most relevant or most recently used commands for the entity. Below you see a series of command tabs:

- Move the cursor over a tab to expand it.
- Place the cursor over an item to see a tool tip.
- Click an icon to launch a command.
- Click the Quad title bar to collapse, click again to expand.

23.3.3 Manipulator

The manipulator allows you to move, rotate scale and mirror selected entities and optionally create copies.

To display the Manipulator do one of the following:

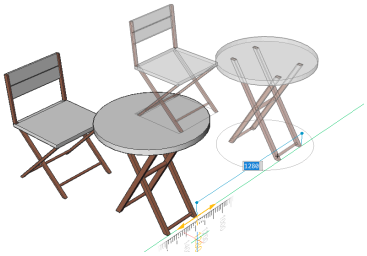
- Select the entity or subentity and choose **Manipulator** in the **Modify** tab on the **Quad**.
- Hold down the left mouse button a little bit longer when selecting.

Nota: The Manipulator aligns along the edge that you enter the entity by. Its origin sits at the closest endpoint of that edge.

Select an axis

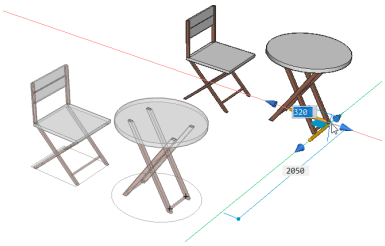
By selecting an axis of the manipulator you are able to move the entity. Specify the value of displacement by entering a value or picking a point. The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

Nota: To copy the entity, hold down the **CTRL** key before clicking the axis.



Select a plane

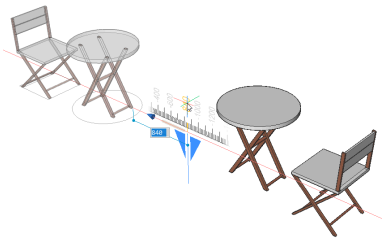
By selecting a plane of the manipulator you are able to move the entity in the selected plane after specifying the value of displacement by entering a value or picking a point.

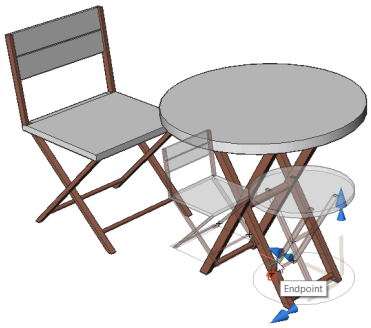


Select an arrowhead

By selecting an arrowhead of the manipulator you are able to scale or rotate the selected entity. By right clicking when hovering over the entity, a contexts menu with other options is displayed.There the function of an arrowhead can be changed between 3D mirror and 3D scale.

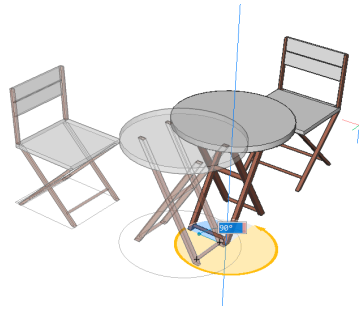
Select one of the arrowheads and specify the mirror axis or scale.





Select a rotation arc

By selecting an arc of the manipulator you are able to rotate the entity after entering the rotation angle value or specifying by clicking on a point in the drawing.



Select the anchor handle

By selecting the anchor of the manipulator you are able to relocate the manipulator.

23.4 Navigation in BricsCAD Shape

23.4.1 Panning

You can move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space.

There are several ways to pan:

- Hold down the middle mouse button.
- Press and hold the **CRTL+SHIFT** and right mouse button.
- Type the *PAN* in the Command line.
- Type the *RTPAN* in the Command line.
- Click the **Real-Time Pan** icon in the toolbar.

To abort the *PAN* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

23.4.2 Rotating a view

You can rotate a view freely, in a constrained manner.

To rotate in real-time:

- Hold down the **SHIFT** and middle mouse button.
- Type *RTROT* in the Command line.
- Click the **Real-Time Constrained Sphere** icon in the toolbar.

To abort the *RTROT* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

23.4.3 Zoom in/out

You can change the magnification of your drawing at any time by zooming.

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.

To zoom in/out:

- Scroll the mouse wheel.
- Press and hold the **CRTL+SHIFT** and left mouse button to zoom in or out.
- Type **ZOOM** in the Command line.
- Click the **Zoom In** or **Zoom Out** icons in the toolbar.

Nota: **Zoom Extents** displays the drawing to the extents of entities

23.4.4 Look From control

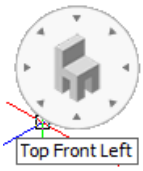
The **Look From** control allows you to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen.

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.



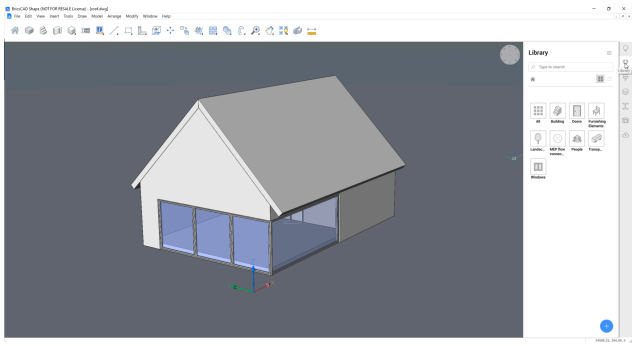
3 Click to confirm.

4 The view is updated.

23.5 Components in BricsCAD Shape

BricsCAD Shape includes a collection of components in the **Library** panel.

Open the library by clicking the airballoon icon on the right hand edge of the screen. Several categories are available.



Click the menu button at the top right corner of the panel to select or hide the available libraries:

- **Bricsys BIM library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Bim\Components*.
- **Bricsys 2D library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Blocks2D*.
- **User library**: content of *C:\ProgramData\Bricsys\Components*

Nota: Choose **Manage Libraries** in the menu to add folders to the COMPONENTSPATH system variable in the **Settings** dialog box.

23.5.1 To insert a component

1 Click the component in the library.

2 Move the cursor into the model.

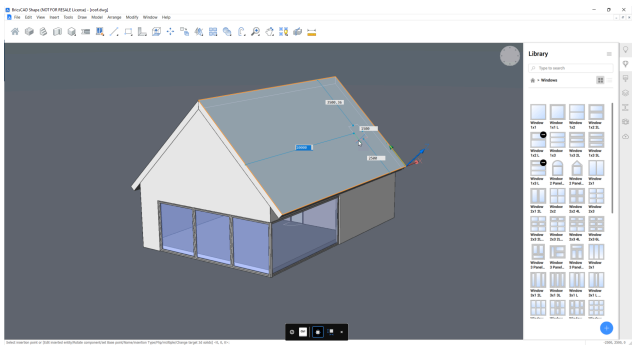
Nota: The selected component is attached to the cursor. The X and Y distances from the origin of the current coordinate system display dynamically.

3 Do one of the following:

- Click to place the component.
- Type a value in the X and Y distance fields.

Nota: Press the **TAB** key to jump to the other field. Press Enter to place the component.

23.5.2 To insert a window in the face of a solid



1 Click the window in the library.

2 Move the cursor into the model over the face of 3D solid.

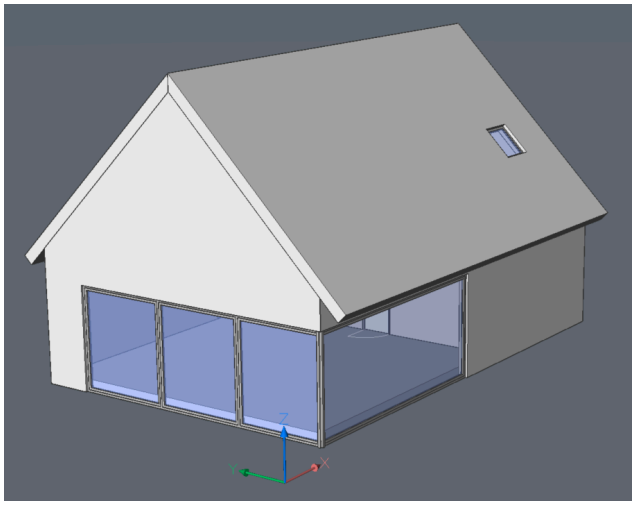
Nota: The window aligns to the face under the cursor. Distance fields from the sides of the face display dynamically.

3 Do one of the following:

- Click to insert the window.
- Type a value in distance fields.

  Nota: Press the **TAB** key to jump to the next field. Press Enter to place the component.

  The window automatically makes an opening in the solid.



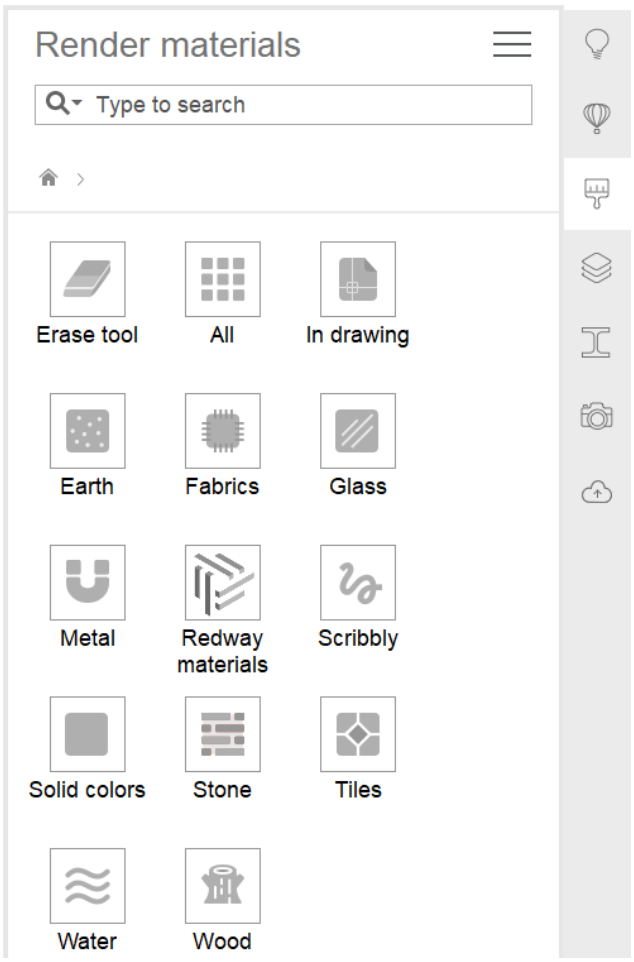
Nota: When you delete a window, the opening is removed.

23.6 Materials in BricsCAD Shape

BricsCAD Shape includes a collection of materials in the **Render materials** panel.

Render materials are stored in: *C:\ProgramData\Bricsys\RenderMaterials\UserMaterials*.

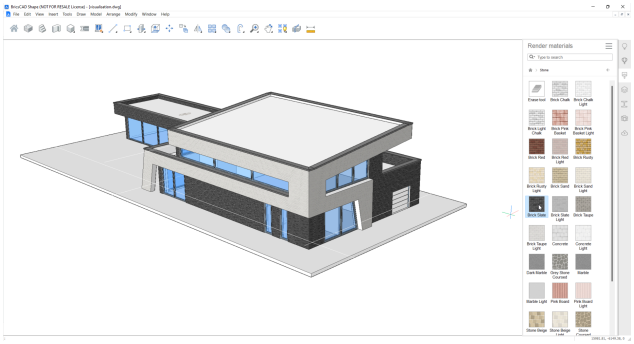
Open the **Render materials** panel by clicking the **paintbrush** icon on the right-hand edge of the screen. Several categories are available. To view all materials that are used in the drawing, select the **"In drawing"** category.



To assign a material to an entity in your model:

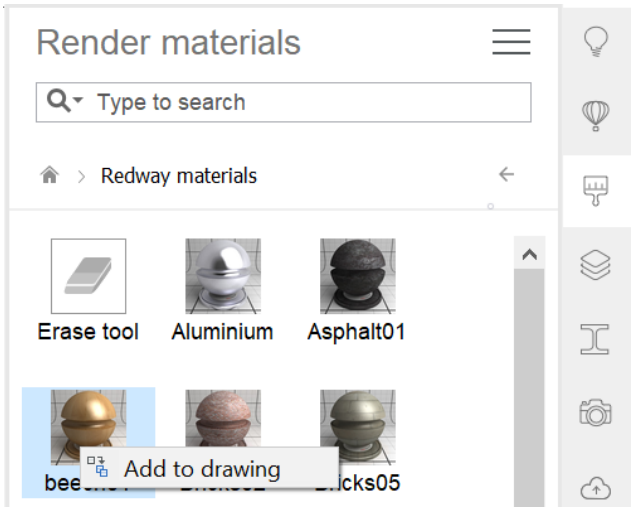
- Select the material in the **Render materials** panel.
- Select the entity you want to assign the material.

Nota: You can apply a material to multiple entities at once by preselecting them.



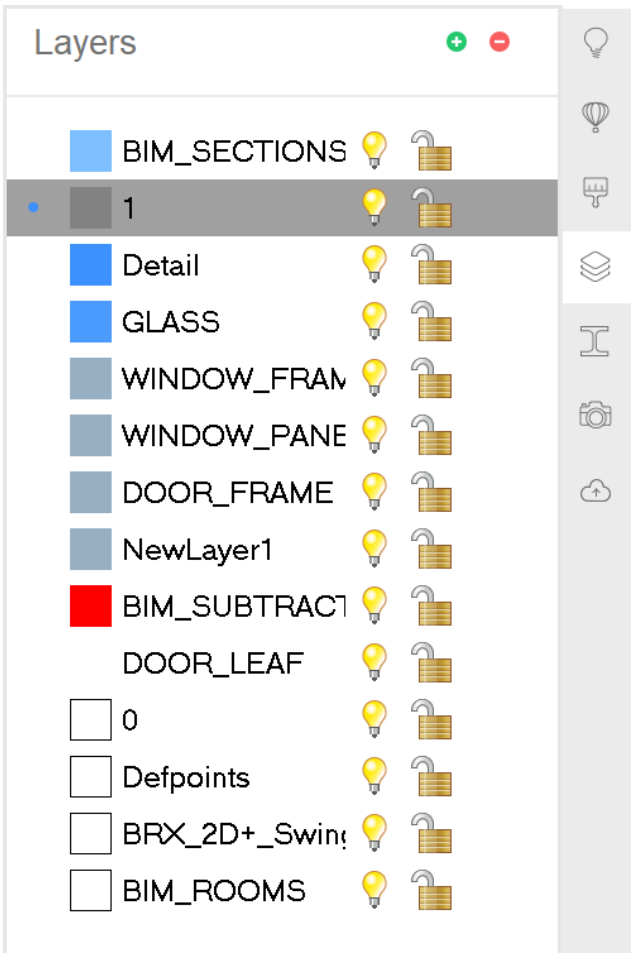
Nota: To remove a material from an entity, use the erase tool and click any entity to remove the material from.

It you want to add materials to the drawing to use later, right-click the material icon and select the "**Add to drawing**" option.



23.7 Layers in BricsCAD Shape

Layers can be used to group entities to organize your 3D model. The **Layer** panel can be accessed through the stacked sheets icon on the right side of the screen.



Nota: The current layer is indicated with the small blue dot. To change the current layer, click in the same column next to the layer you want to set as current layer.

23.7.1 Turn on/off

To turn on and of specific layers, toggle the lightbulb of the layer on and off. This way elements can be hidden or isolated. This can be of huge help with navigation.

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

23.7.2 Lock/unlock

To lock or unlock layers, toggle the lock icon of the layer.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

23.7.3 Create/delete

To create a new layer, click the **green plus** sign. A new layer with a generic name is created. To delete a layer, make sure the layer to delete is selected and click the **red minus** sign.

23.7.4 Edit

Properties such as name and color of layers can be edited:

- To rename a layer, double-click the layer name or select **Rename** from the right-click menu.
- To change the color, select one or more layers and then click the current color for one of the selected layers. The **Select Color** dialog box is displayed. The color you select is applied to all selected layers.

23.8 Working with steel profiles

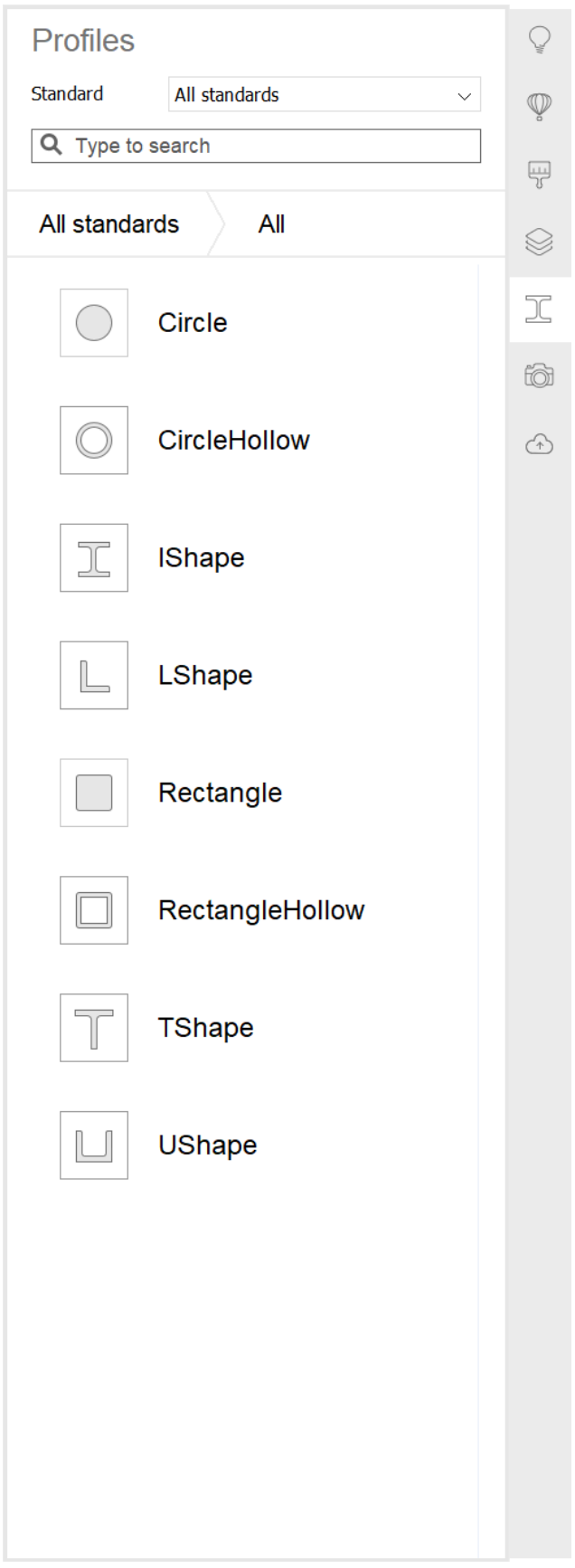
BricsCAD Shape offers standard structural steel profiles which enable you to easily create structural elements. Multiple standards are available, including American, British and European standards.

Steel profiles are created as linear solids.

23.8.1 Creating a steel profile

1 Click the **Profile** icon the vertical toolbar, on the right-hand edge of the screen.

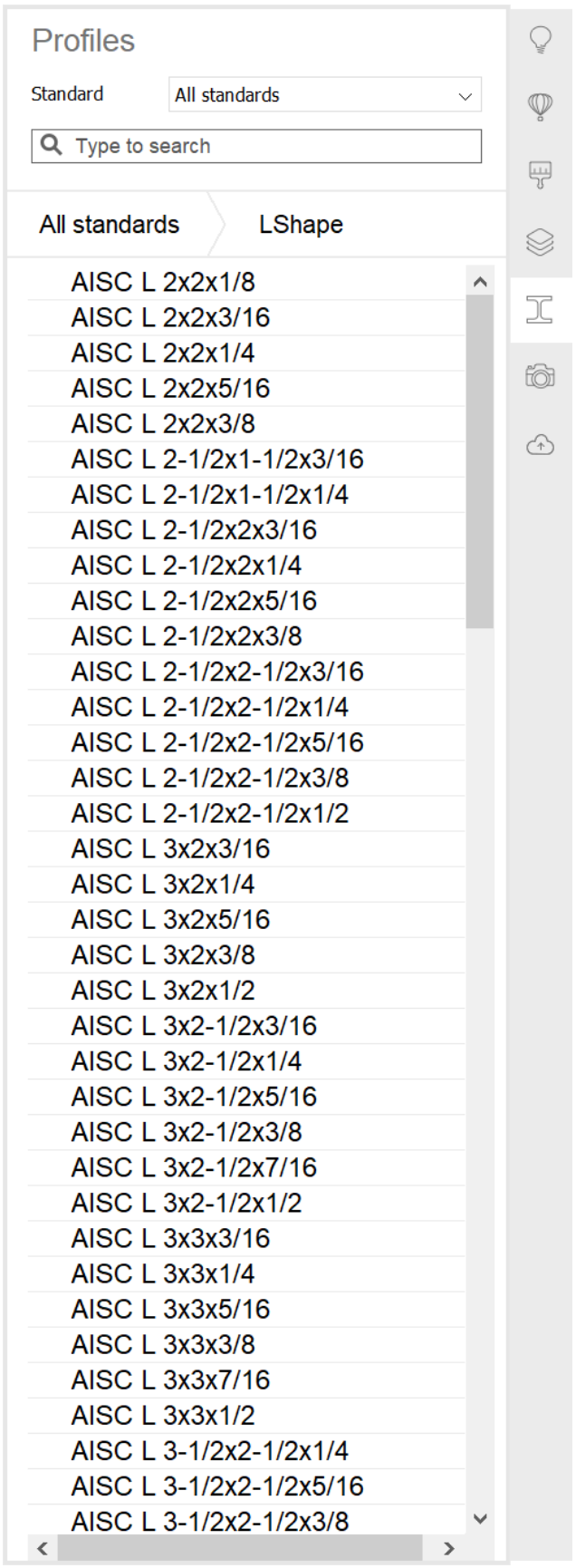
The **Profiles** panel displays.


Profiles
Standard
All standards
Type to search
All standards
All
Circle
CircleHollow
IShape
LShape
Rectangle
RectangleHollow
TShape
UShape

2 Optionally select a standard.

3 Select a profile shape.

4 A list of all profiles of the selected shape displays.


Profiles
Standard
All standards
Type to search
All standards
LShape
AISC L 2x2x1/8
AISC L 2x2x3/16
AISC L 2x2x1/4
AISC L 2x2x5/16
AISC L 2x2x3/8
AISC L 2-1/2x1-1/2x3/16
AISC L 2-1/2x1-1/2x1/4
AISC L 2-1/2x2x3/16
AISC L 2-1/2x2x1/4
AISC L 2-1/2x2x5/16
AISC L 2-1/2x2x3/8
AISC L 2-1/2x2-1/2x3/16
AISC L 2-1/2x2-1/2x1/4
AISC L 2-1/2x2-1/2x5/16
AISC L 2-1/2x2-1/2x3/8
AISC L 2-1/2x2-1/2x1/2
AISC L 3x2x3/16
AISC L 3x2x1/4
AISC L 3x2x5/16
AISC L 3x2x3/8
AISC L 3x2x1/2
AISC L 3x2-1/2x3/16
AISC L 3x2-1/2x1/4
AISC L 3x2-1/2x5/16
AISC L 3x2-1/2x3/8
AISC L 3x2-1/2x7/16
AISC L 3x2-1/2x1/2
AISC L 3x3x3/16
AISC L 3x3x1/4
AISC L 3x3x5/16
AISC L 3x3x3/8
AISC L 3x3x7/16
AISC L 3x3x1/2
AISC L 3-1/2x2-1/2x1/4
AISC L 3-1/2x2-1/2x5/16
AISC L 3-1/2x2-1/2x3/8

5 Click a profile in the list, then move the cursor to the model.

6 The outline of the selected profile is attached to the cursor.

7 Specify a point.

8 A linear solid displays dynamically.

A length and angle field display the current length an angle of the profile.

When the current direction is close to the X, Y or Z direction of the current coordinate system a colored tracking line displays. Press the **Shift** key to lock the tracking line. Press again to unlock.

9 Do one of the following to create the profile:

- Specify a point.
- Type a value in the length field, then press **Enter** or press the **TAB** key to activate the angle field.

The next profile displays dynamically.

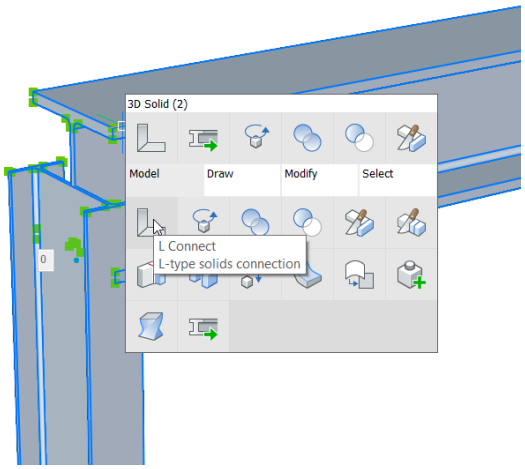
10 Repeat the previous steps to create more profiles or right click to stop.

23.8.2 Connecting Profiles

The L-Connect tool creates various types of L-connections between two selected profiles.

1 Select both profiles.

2 Click the L-Connect tool in the Model tab on the Quad.



- A bisector L-connection is created.
- The **Hotkey Assistant** (HKA) displays at the bottom of the screen.

3 Press the **Ctrl** key to modify the connection type.

The current connection type highlights in the **HKA**.

4 Right click to accept the connection type.

23.8.3 Assigning a different profile

1 Place cursor over the new profile in the **Profiles** panel. Do not click.

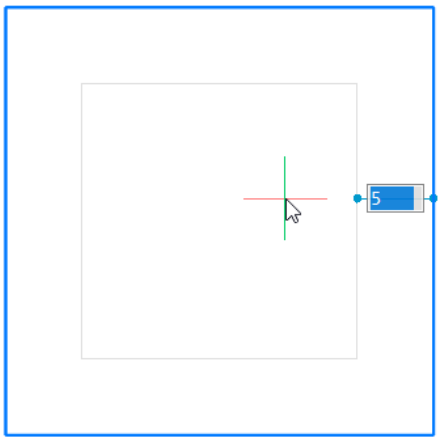
2 Hold down the left mouse button, then drag the profile onto the linear solid.

The selected profile is assigned to the linear solid.

23.8.4 Assigning a custom profile

1 Draw a custom profile shape using the tools in the **Draw** tab on the **Quad**:
   - A closed polyline.
   - A square or rectangle using the **Rectangle** tool.
   - A circle.
   - An ellipse.

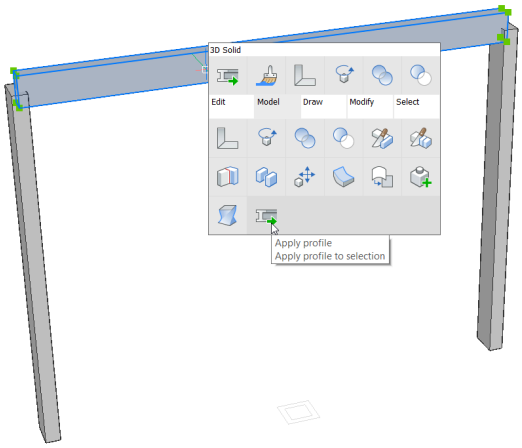
2 Optionally, use the **Offset** tool to create a hollow profile: select the outline, then specify the offset.



3 Hold down the **Ctrl** key, then select one or more linear solids. The linear solid(s) highlight(s).

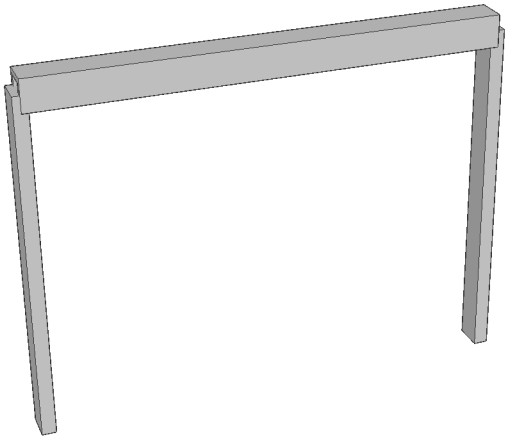
4 Click the **Apply profile** tool in the **Model** tab on the **Quad**.

Prompts you: Select profile [selection options (?)]:



5 Do the following:

- Select the custom profile.
- Use the window overlap selection method to select the profile outline and the offset when applying a hollow profile.
- A preview of the result displays.
- Prompts you: Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>:



6 Do one of the following:

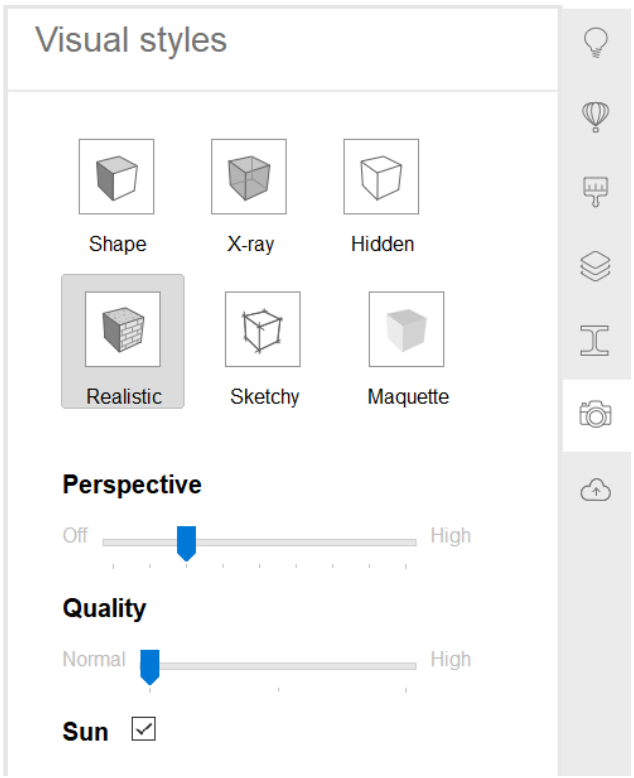
- Right click to apply the profile.
- Choose an option.
- Press the Esc key to quit.

23.9 Visualization in BricsCAD Shape

23.9.1 Visual styles panel

Visual styles control the appearance of your model.

The **Visual Styles** panel can be accessed by clicking the camera icon at the right side of the screen.



The look of the entire model will be changed when selecting another style.

The available visual styles are:

- Shape
- X-ray
- Hidden
- Realistic
- Sketchy
- Maquette

Perspective

You can change the perspective view by moving the **Perspective** slider.

Nota: Moving the perspective slider all the way to the left will turn the perspective off.

Quality

To control the on-screen rendering quality of the model, move the **Quality** slider.

Sun

To make the shadows of the sun appear, toggle the **Sun** checkbox.

23.9.2 Hide/Isolate/Show entities

Hide

When model elements block your view, you can hide them easily.

To hide one or more elements:

- Select the entities you want to hide.
- Click the **Hide Entities** icon in the **Quad** or in the toolbar.

Isolate

The opposite of hiding an element is isolating it, which will hide all the non-highlighted elements in your model.

To isolate entities:

- Select the entities you want to isolate.
- Click the **Isolate Entities** icon in the **Quad** or in the toolbar.

Show

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:

- Select the **Show Entities** icon in the “No Selection” Quad or in the toolbar.

23.9.3 Sections

You can use Sections to cut through your model dynamically.

To create a section:

- In the “No Selection” Quad, select the **Define Section** tool, found under the **Model** tab.
- Select any solid face in your model to create the section, parallel to the selected solid face.

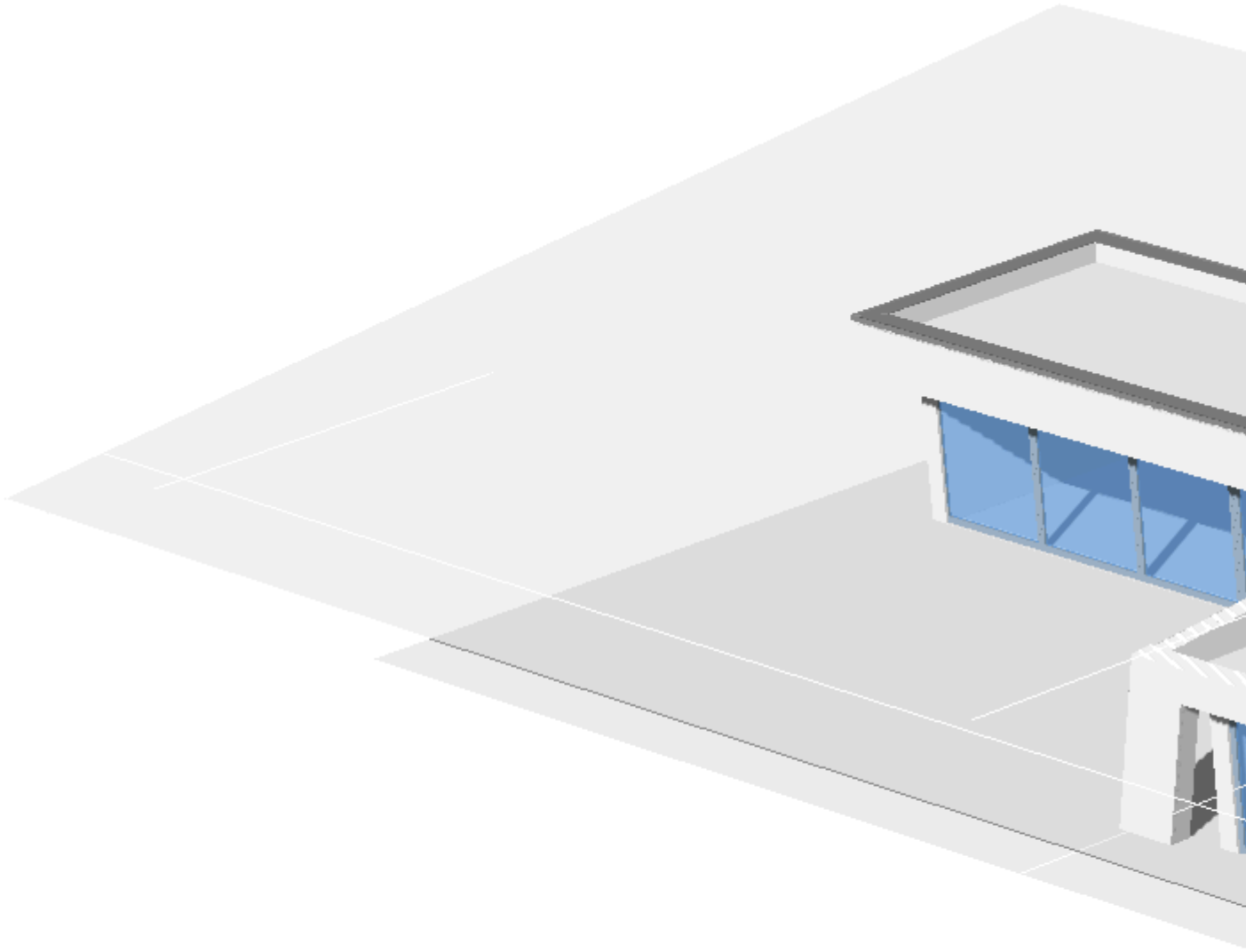
Nota: If you don’t choose a surface to attach the Section, a horizontal Section plane will be created.

You can toggle a section on and off by:

- double-clicking its handle
- highlighting the section plane and using **Clip Display** tool 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 click the arrow displayed near the midpoint grip, you will flip the clipping direction of the Section.



23.10 Solids in BricsCAD Shape

23.10.1 Drawing a box

1 Start creating a box by accessing the *BOX* command through the command line or by clicking the **Box** icon in the Quad or toolbar.

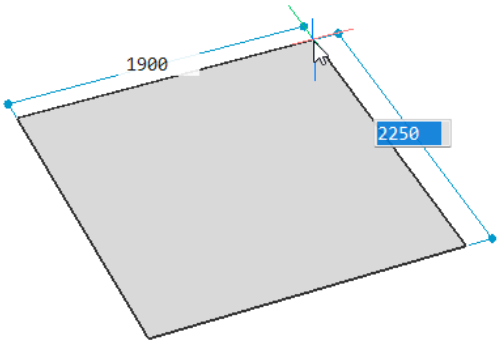
2 Specify a corner for the base of the box.

3 Specify the opposite corner for the base of the box to apply both the length and width.

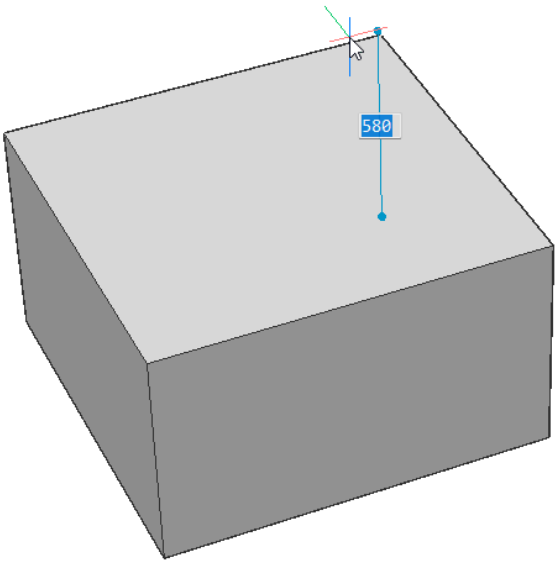
4 You can do this dynamically, by hovering the cursor at the opposite corner, or by entering the desired values in the dynamic input fields.

The box is created parallel to the x and y axes.

Nota: The blue dynamic field is the active one.



5 Specify the height of the box.



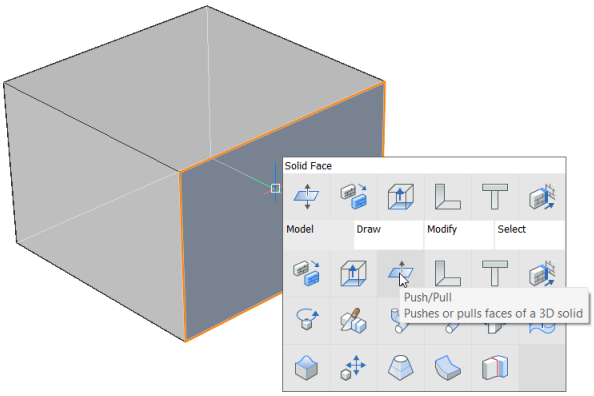
23.10.2 Manipulate a solid face

To manipulate a solid face:

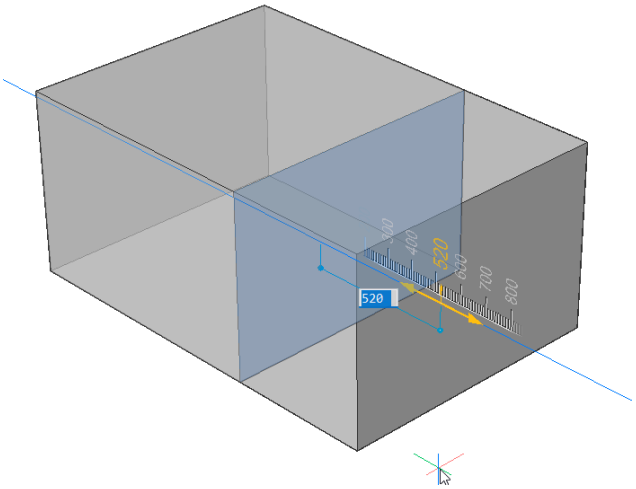
1 Hover the cursor over a face until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *DMPUSHPULL* command can be accessed.



4 Specify the displacement value or move with your cursor to define the end position of the face.



Nota: When selecting multiple faces and accessing the command, both faces will be pushed or pulled over same distances.

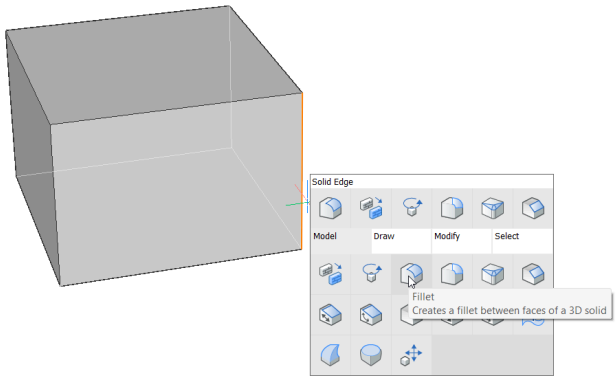
23.10.3 Manipulating a solid edge

To manipulate a solid edge:

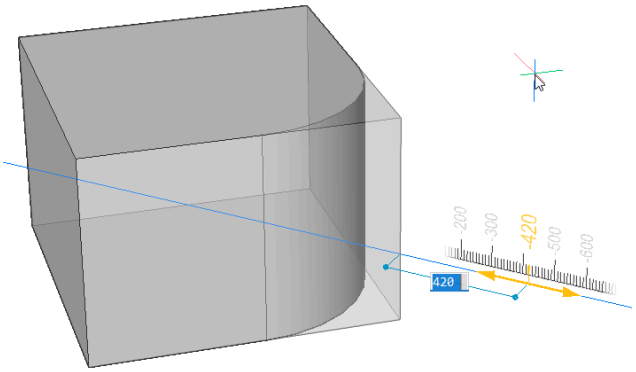
1 Hover your cursor over an edge while holding the **CRTL** key until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *FILLET* command can be accessed.

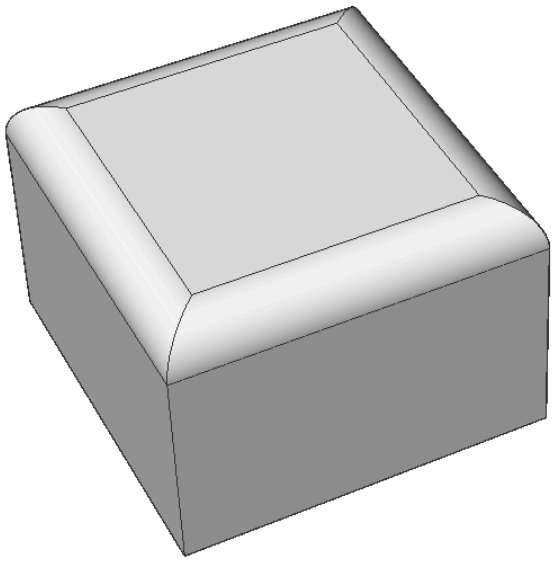


4 Specify the displacement value or move with your cursor to define the end position of the face.



Nota: When selecting multiple edges and accessing the command, all selected edges will have a fillet of the same radius.

Nota: When using a selection window to select edges, make sure you press the **CRTL** key twice so the edge selection icon is selected at the bottom of your screen.



23.10.4 Extrude solids

Selected boundaries can be extruded to create solids or surfaces. For example, a floor slab can be extruded between existing walls.

The extrude command can be accessed by the Quad or in the toolbar.

Nota: The extrude command can also subtract geometry also.

23.11 Creating walls in BricsCAD Shape

There are two ways to create a simple wall:

- Using the *POLYSOLID* command.
- Using the *BIMQUICKDRAW* command.

Nota: For a detailed explanation about *BIMQUICKDRAW*, read the article Quickdraw in BricsCAD Shape article.

23.11.1 About the Polysolid tool

The *POLYSOLID* command allows you to create wall solids by selecting a start point, an endpoint and entering a height.

23.11.2 Procedure: Create a simple wall

1 Launch the *Polysolid* command:

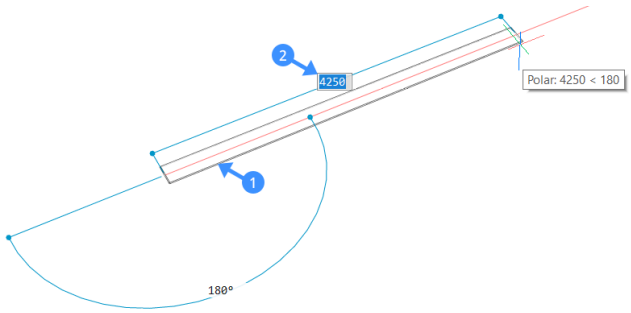
- Click the **Polysolid** icon

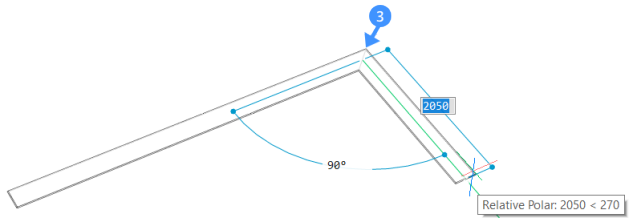
  in the toolbar
- Type *POLYSOLID* in the command line.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

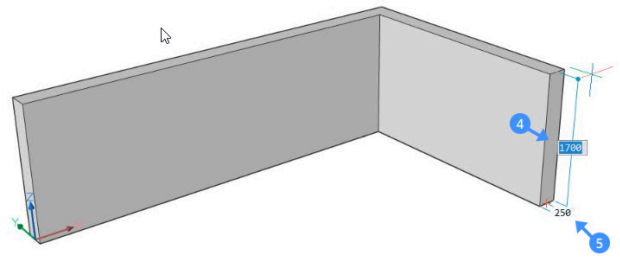
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



4 Select an endpoint or enter a value in the dynamic input field and press **Enter** key. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.



5 Press **Enter** once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing **Tab** key.



6 To change the justification of the wall hit **CTRL** while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.



Nota: The width and height of the previous wall will be the new default values of the next Polysolid.

23.11.3 Procedure: Using an existing 2D plan to create walls

1 Launch the *POLYSOLID* command.

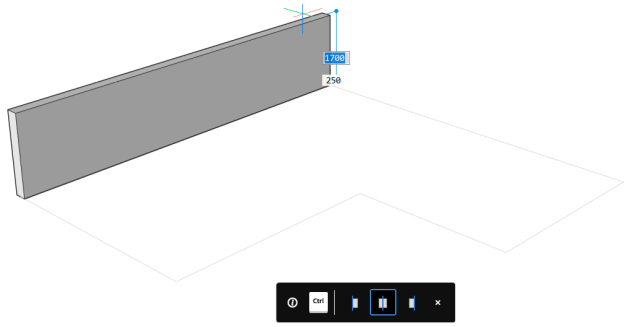
2 Type E in the command line or select **Entity** in the prompts menu.

3 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 polysolid base.

4 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.

5 The current width of the wall appears in the **Width Field**. Press **Tab** switch between the Height and the Width field.

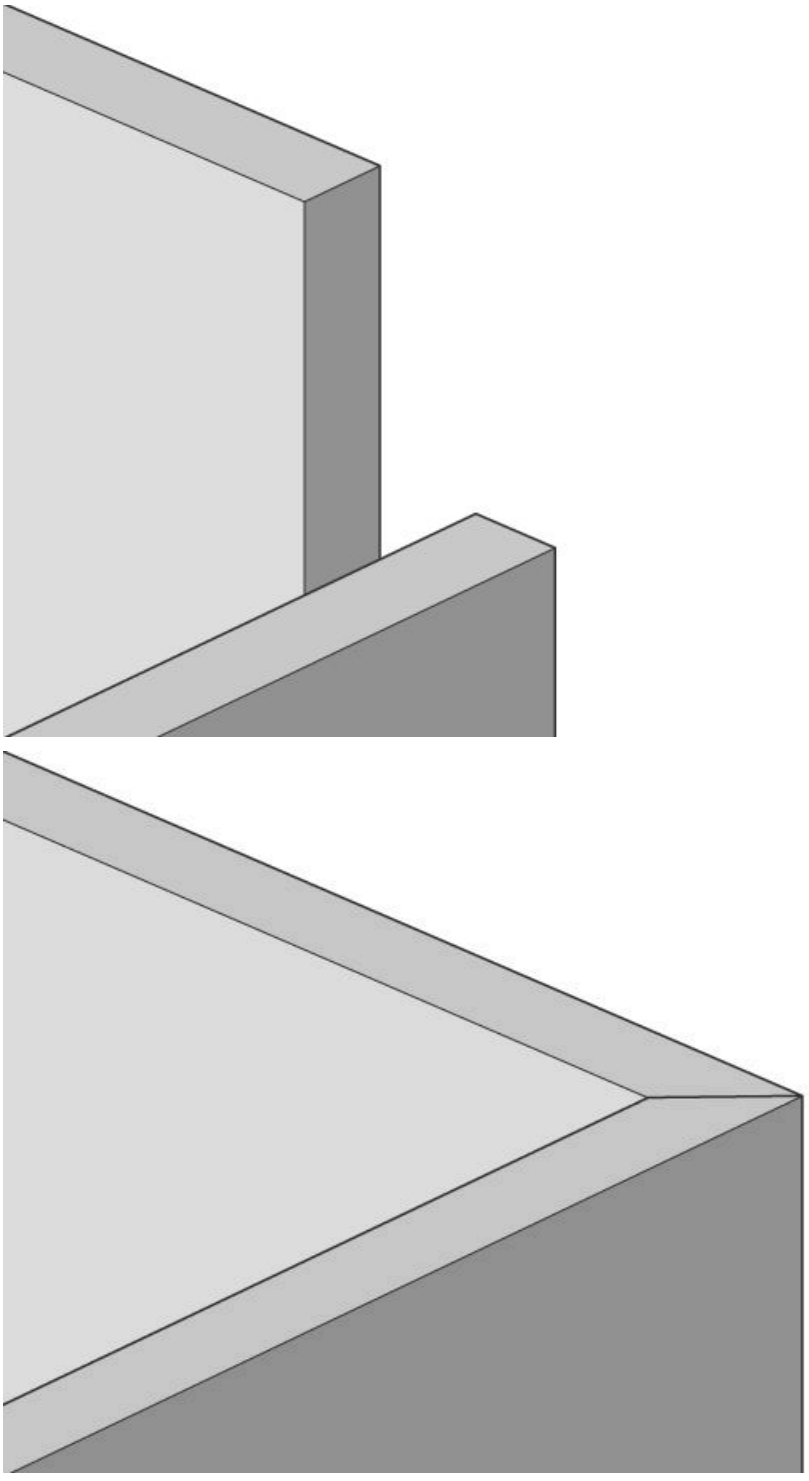
6 The **Hotkey Assistant** appears and displays the possible justification options. Press **CTRL** to cycle through the wall justification options.



23.11.4 Procedure: Creating L-connections between two walls

1 Launch the **LConnect** tool, by typing *LConnect* in the Command line and press Enter.

Select the two wall elements you want to connect.



Nota: When two walls intersect, a mitered connection will be created by default.

2 Change the layout of the wall connections using the 'LConnect HotKeyAssistant widget'.

3 The selected connection type is indicated by a blue frame.



The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.

Press **Enter** to accept the current connection type or press **CTRL** to cycle through the possible connection options.

23.11.5 Procedure: Creating T-connections between two elements

1 Launch the *TConnect* command.

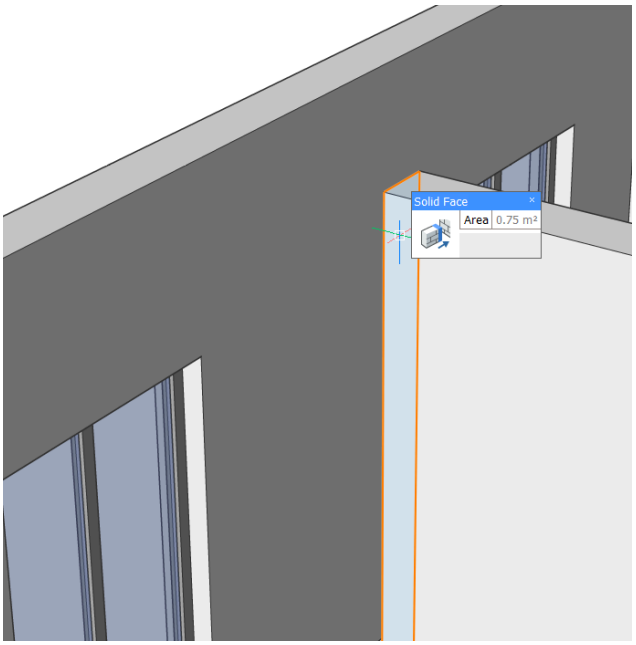
The tool allows you to connect the minor face of a wall to the major face of another element or wall. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or side face of a slab to a wall.

2 Select the faces of the objects that you would like to connect and press **Enter**.

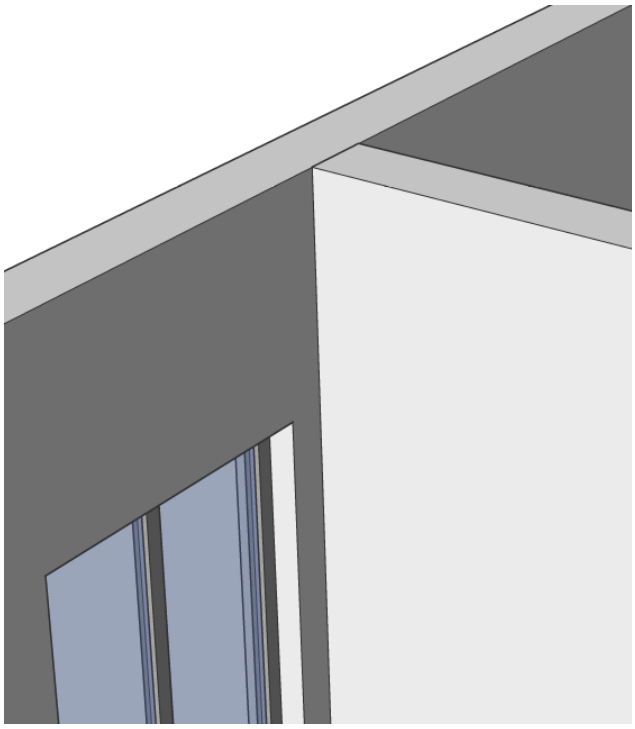
3 Select **Connect to nearest** in the prompt menu.

You can also use the **Quad** menu to create a T-connection between two walls or other elements:

- In the drawing area, highlight the face you want to connect to another wall.
- Use the **Connect with Nearest** tool from the **Quad**.



The Connect with Nearest tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



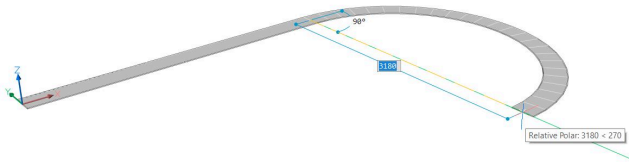
23.11.6 Procedure: Creating a curved wall

1 Launch the *POLYSOLID* command.

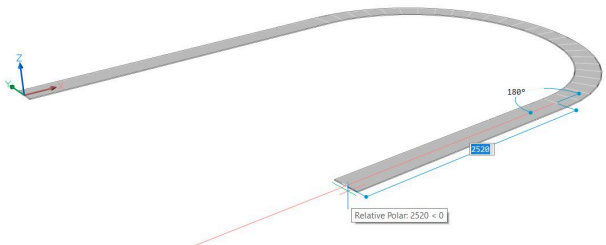
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

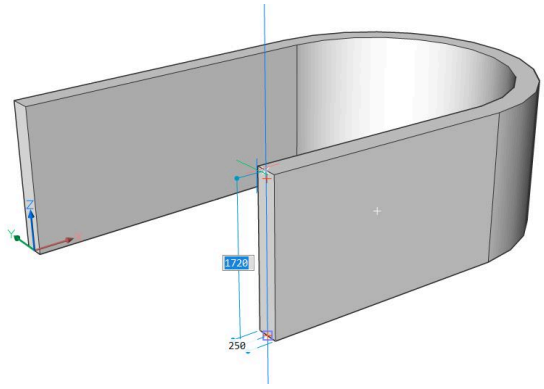
4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press **Enter**.



5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.

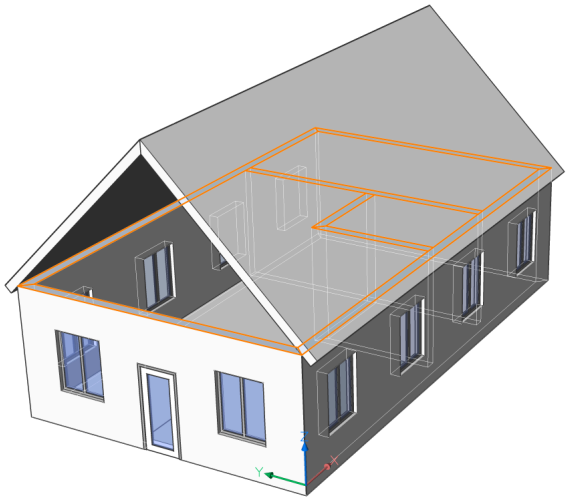


6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

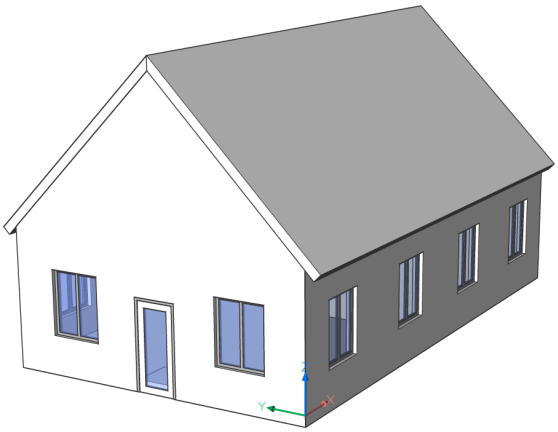


23.11.7 Procedure: Connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



23.12 Quickdraw in BricsCAD Shape

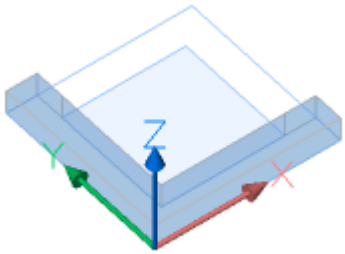
23.12.1 About the Quickdraw tool

The **Quickdraw** tool allows you to get started with conceptual modeling easily. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, Quickdraw will remain active until you explicitly exit the command or until another command is executed.

23.12.2 The magnetic snapping feature

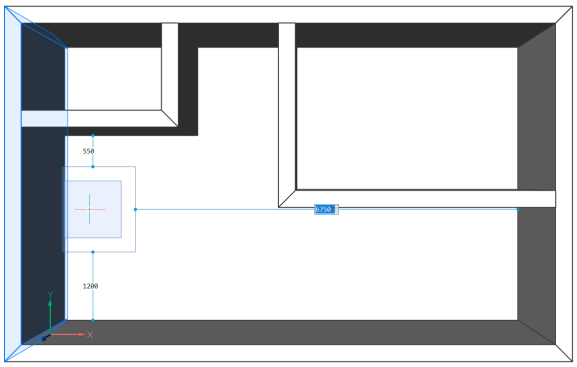
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object. You have the following options:

1 Align to the origin

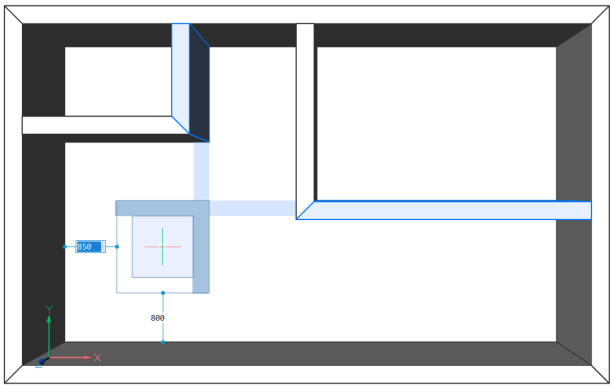


2 Align to existing walls

- To the wall itself:



- The extends of the wall:



23.12.3 Procedure: Creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

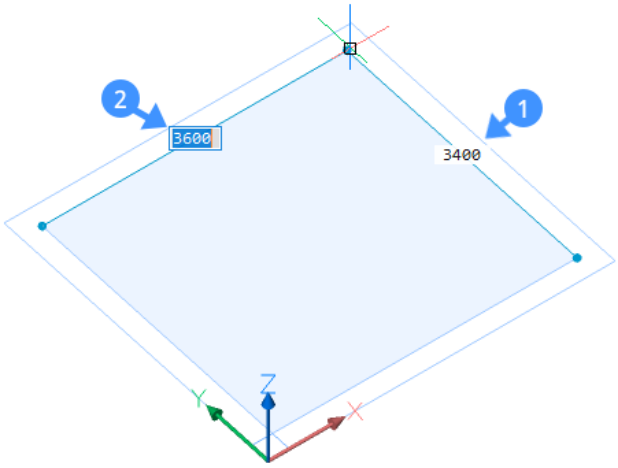
You are prompted: Select first point or [floorHeight/wallWidth/slabThickness/autoAdoptsizes]:

Nota: Select one of the options in the command bar to change the wall width, floorHeight and slab thickness of the roombeforeyou click and then specify the first corner of a building.

Nota: When the **autoAdoptsizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to. If you don't want this to happen, you can turn this option off in the command line.

2 Select any point in the drawing area to set the first corner.

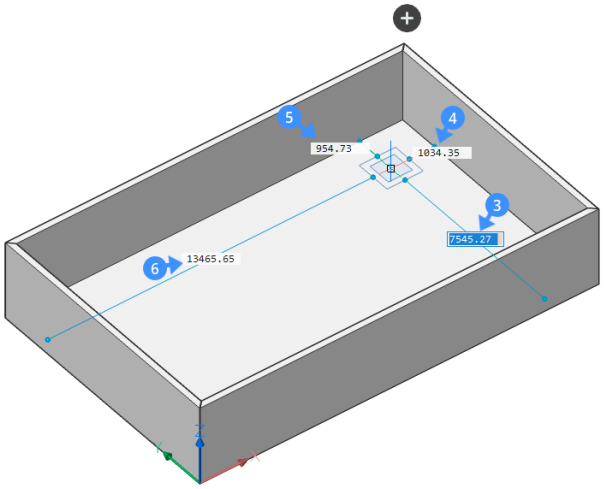
3 Drag the mouse cursor to define the footprint of the building.



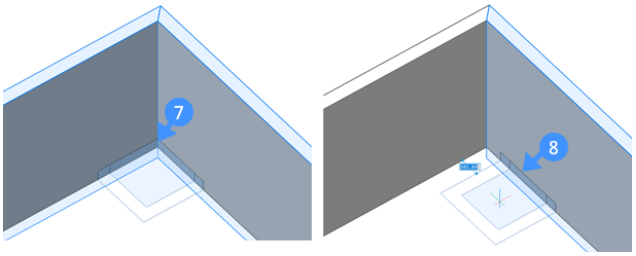
Specify the values of the room width and length in the Dynamic dimension field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).



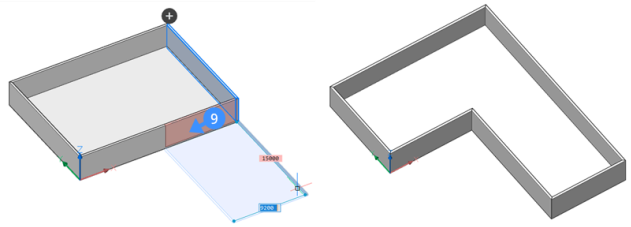
If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

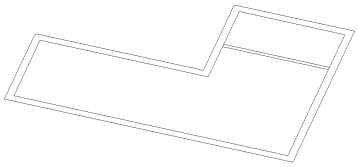


Nota: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.

23.12.4 Procedure: using Quickdraw starting from a 2D layout

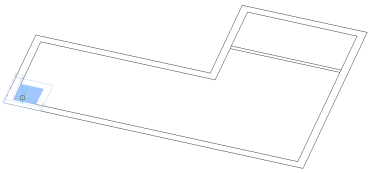
1 Open a 2D floorplan of a rectangular or L-shaped building.



Nota: It is also possible to draw a simple 2D floor plan with the POLYLINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

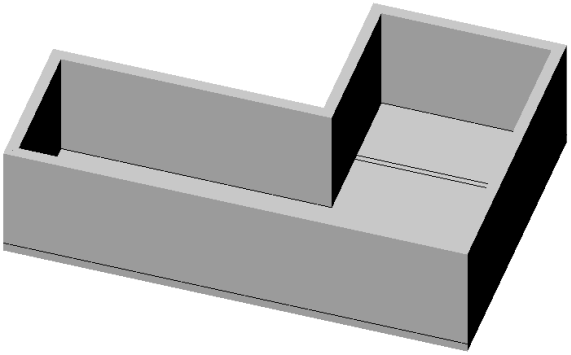
2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

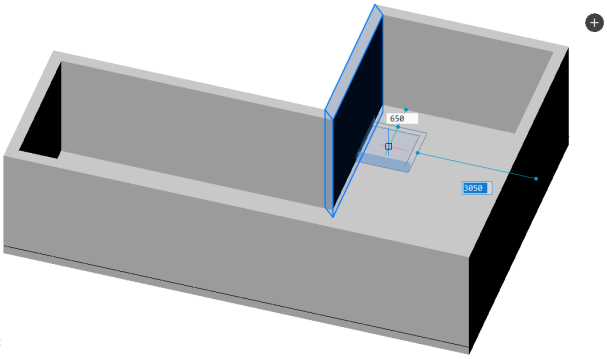


Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

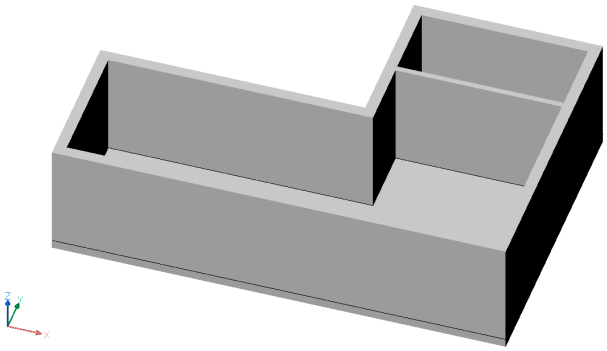


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



23.13 Creating windows and doors in BricsCAD Shape

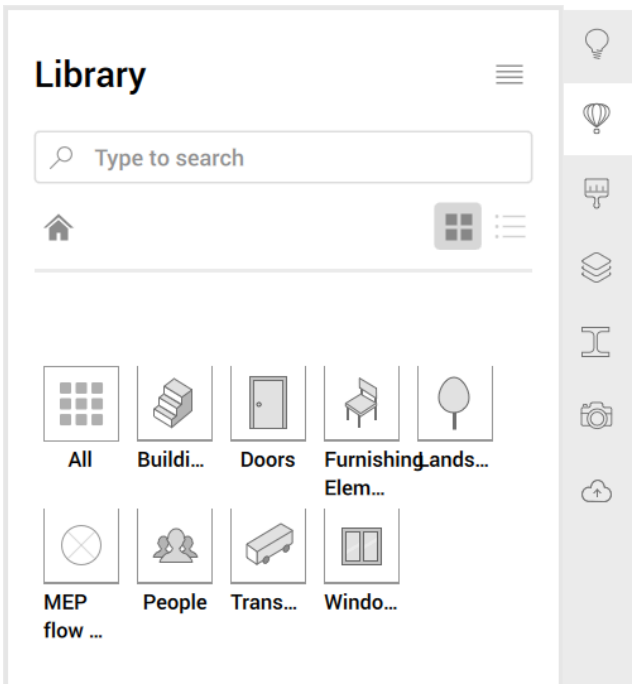
There are two ways to create windows or doors in BricsCAD Shape.

- Pick a door or window from the **Library** panel.
- Create your own custom window.

23.13.1 Procedure: Pick windows or doors from the library

1 Open the **Library** panel by clicking the airballoon icon on the right hand edge of the screen.

2 Click the **Doors** section in the **Library** panel.



3 Select a door from the panel.

4 Move the cursor into the model.

5 Click to place the component.

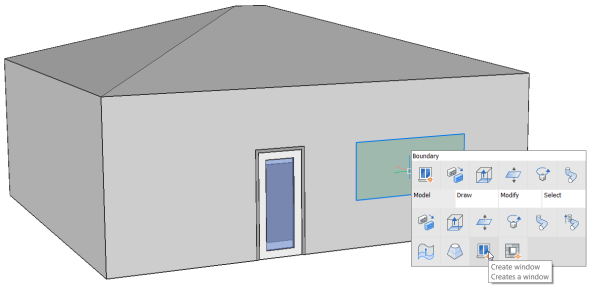
6 Press the **CTRL** key to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the **Tab** key. When you're done sizing and placing the door, press the **Enter** key.

Nota: This is the same for a window. Change Doors to Window section. Drag a window into the wall. You can change the size and location of a window just as you did with the door.

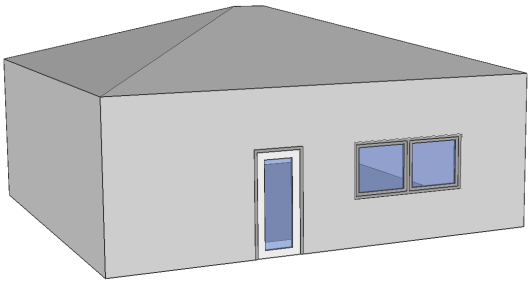
23.13.2 Procedure: Create a custom window

1 Use the either the **Polyline** tool, the **Polygon** tool or the **Rectangle** tool to draw the outline of the window on the wall face.

2 Move the cursor inside the window outline and, when the boundary is highlighted, choose **Create window** from the Quad.



3 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 also create a custom window from a wall boundary:

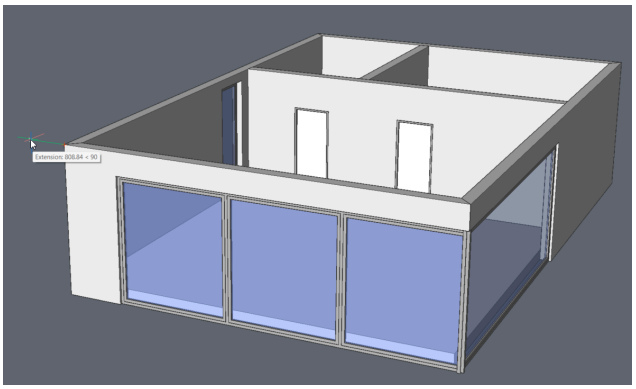
1 Select face of the outer wall and use the **Offset** tool from the Quad.

2 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.

23.14 Creating roofs in BricsCAD Shape

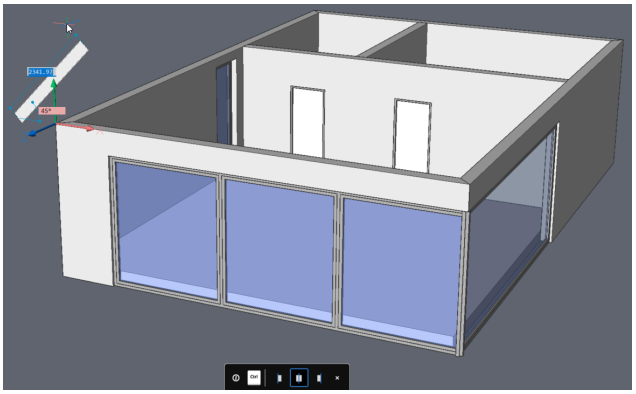
Launch the *POLYSOLID* command trough the toolbar or the "No selection" Quad.

1 To align the drawing lane to the wall surface, press the **SHIFT** button while hovering over the wall surface. The alignment of the slab can be changed by pressing the **CRTL** button.

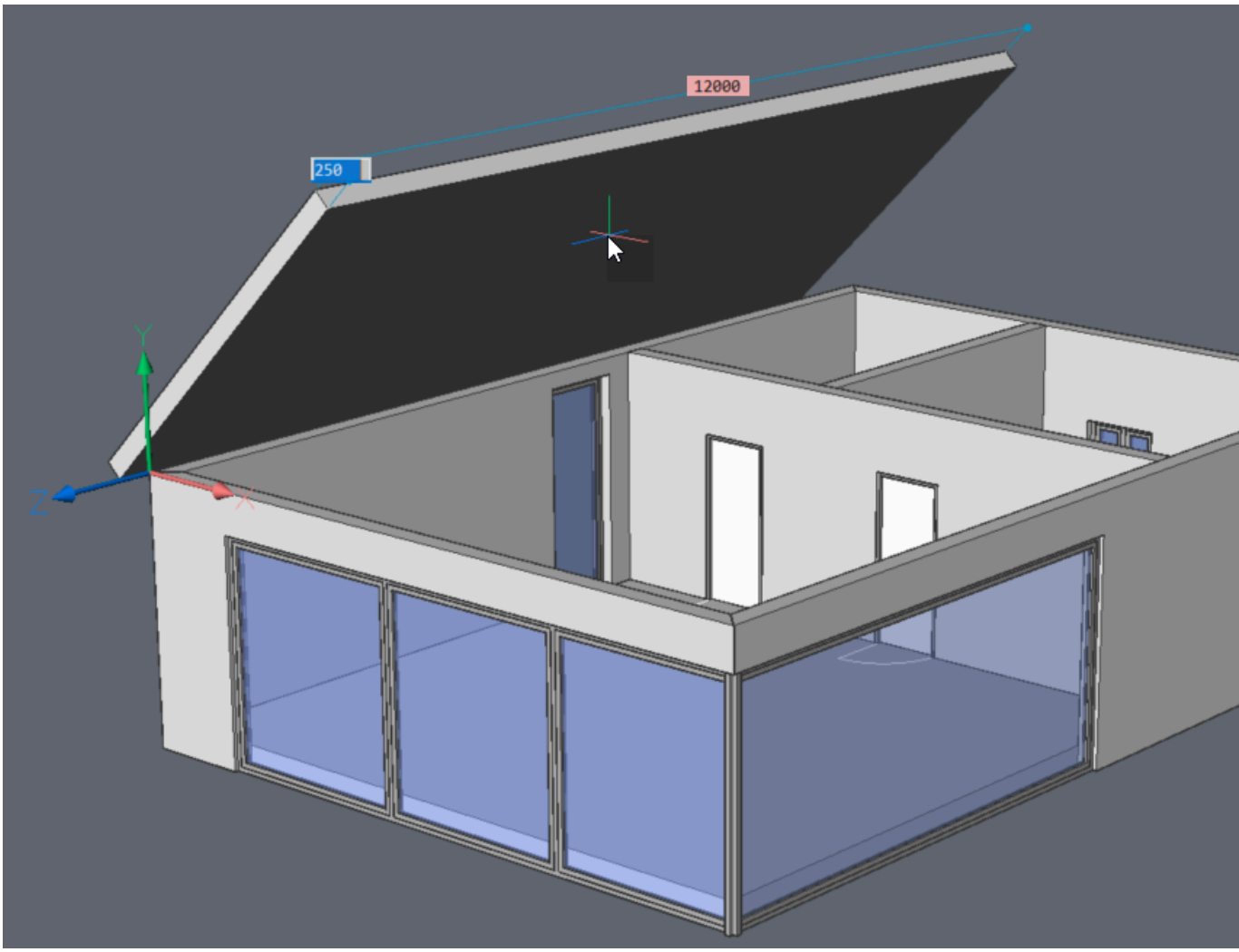


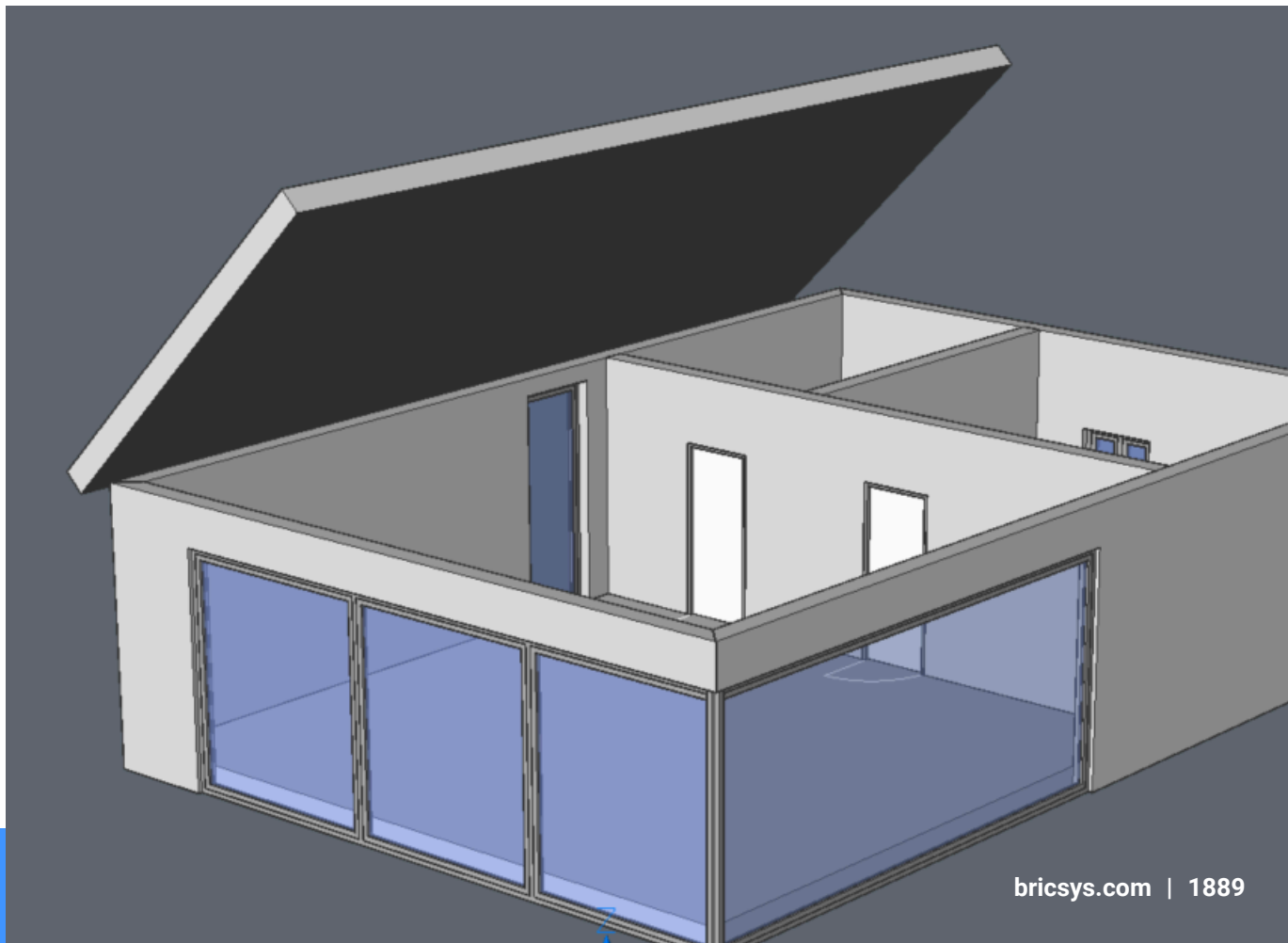
2 Pick an origin point to start drawing.

3 To lock the inclination, press TAB until the dynamic dimension of the inclination turns blue. Enter a value and press Enter.

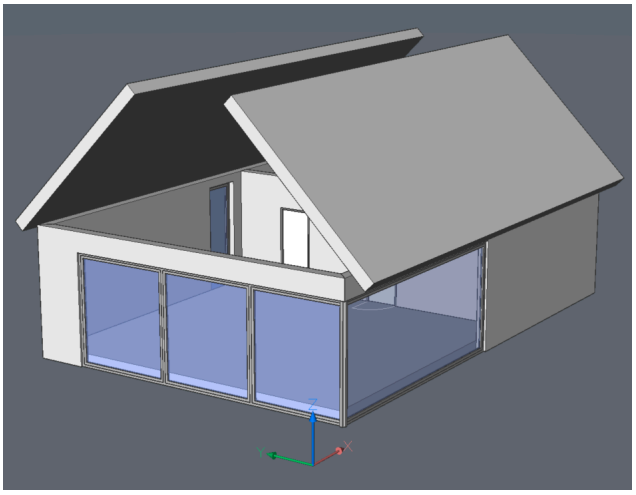


4 The profile thickness can be changes by pressing the TAB key and entering the desired value.

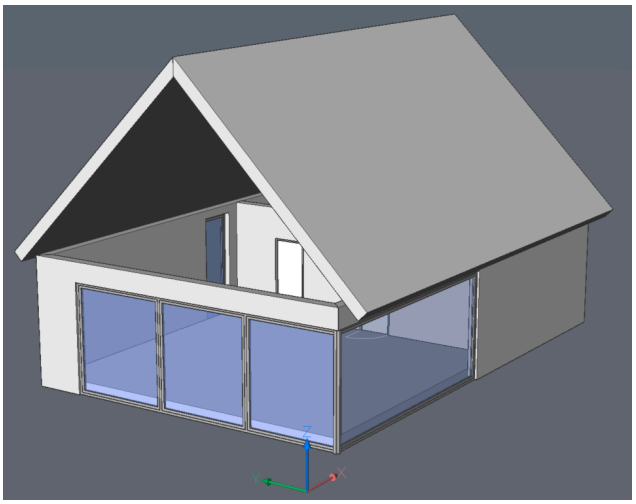

12000
250
Y
Z
X



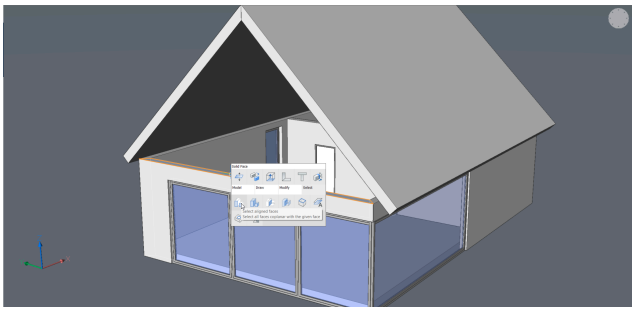
5 Repeat the same process on the other side of the building. Another way to have identical slabs on both sides is by 3D mirroring the first slab.



6 Access the **L connect** tool to connect both slabs.



7 To connect the walls to the roof, select one of the top surfaces of a wall. Trough the Quad, the **Select aligned faces** option can be accessed in the **Select** tab.



8 Once the top surfaces are selected, the **Connect with nearest** command can be accessed trough the Quad in the **Model** tab. Walls are extended up to the roof.

