Bricsys®

BricsCAD V21

Dokumentacja produktu



Bricsys®

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1. Co nowego

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 stronie 32

Productivity na stronie 35

Drafting na stronie 49

Modeling na stronie 51

Civil tools na stronie 60

Performance and compatibility na stronie 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 stronie 958

Modeling techniques na stronie 959

Building data na stronie 966

Project collaboration na stronie 975

Design documentation na stronie 976

Point clouds na stronie 985

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:

Przegląd na stronie 1529

Menedżer BOM na stronie 1530

Biblioteka części na stronie 1533

Parametry i ograniczenia na stronie 1534

Części standardowe na stronie 1534

Arkusz Blachy na stronie 1534

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

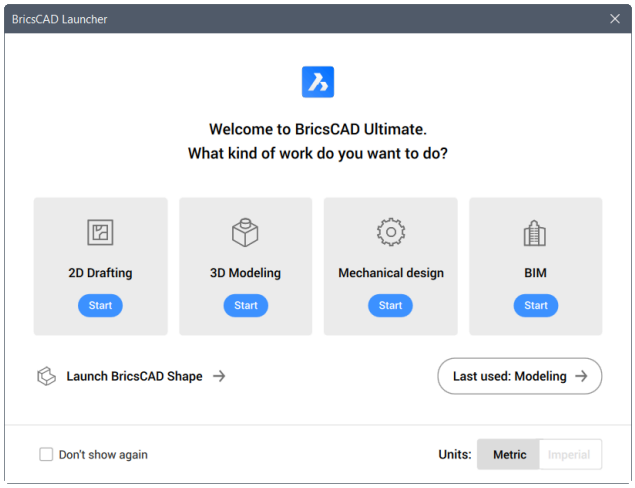
BricsCAD® V21 offers a variety of user interface enhancements to help increase your productivity without disrupting your existing workflow.

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.

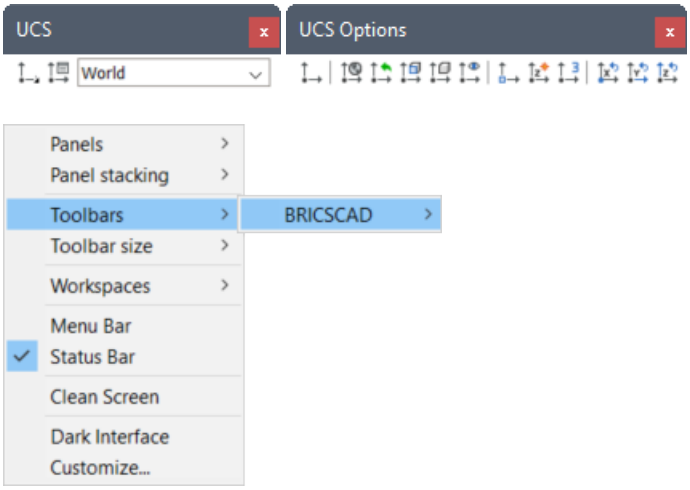
1.2.3 Ribbon

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

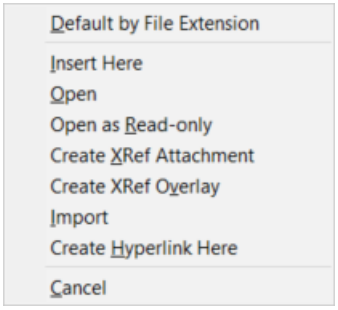
New UCS and UCS Options toolbars offer easy access to UCS commands and options. You an access these toolbars by right-clicking on a toolbar, a panel, or the ribbon and expanding the **Toolbars** > **BricsCAD** menu.



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

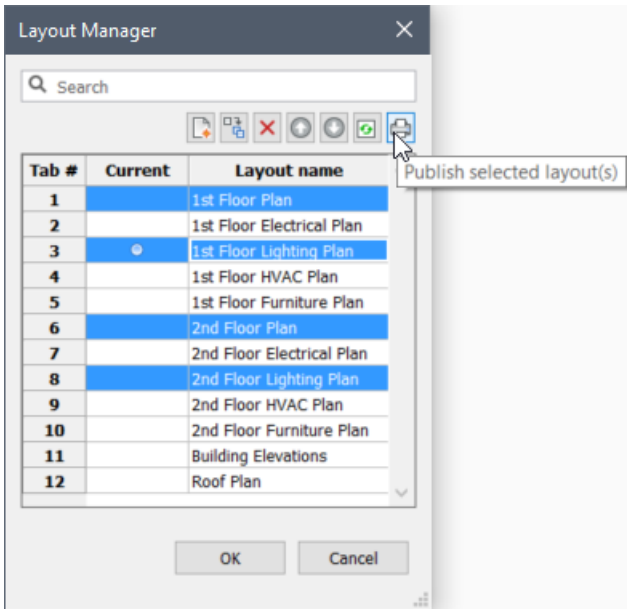
1.2.5 Menus

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**.

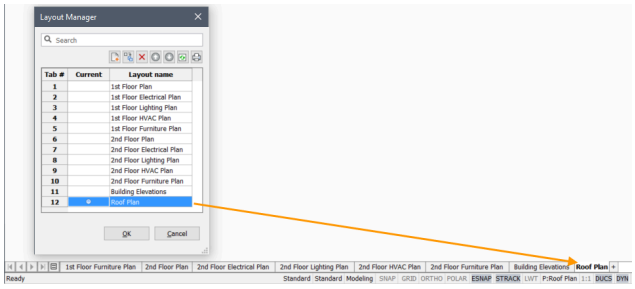


1.2.6 Layouts

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

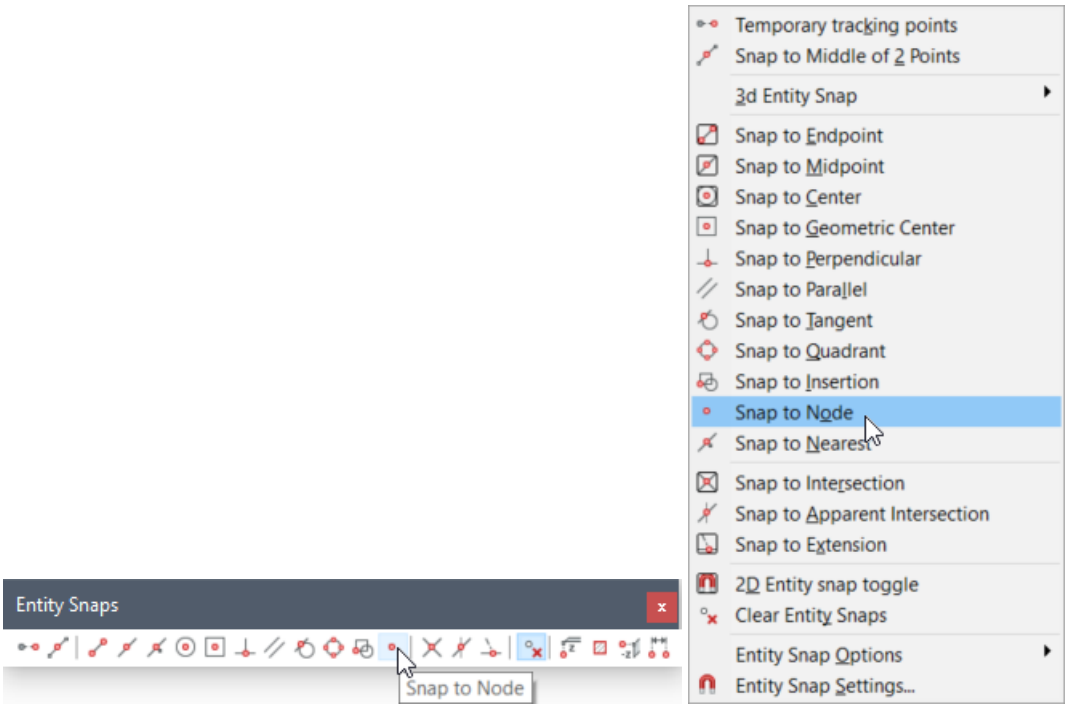


When you set a layout current in the Layout Manager, the active layout tab automatically scrolls to remain visible in the layout tab list.



1.2.7 Entity Snaps

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

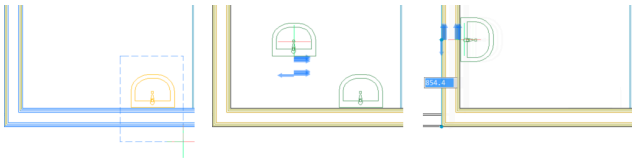


1.3 Productivity

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!

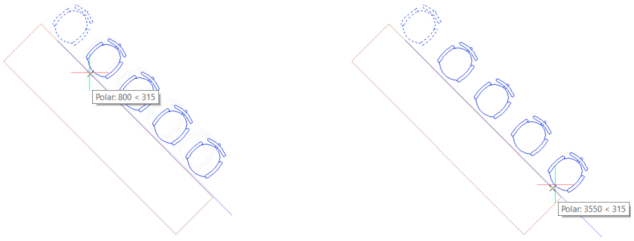


For more flexibility, you can first select the entities you want to move and then launch the MOVEGUIDED command. The pre-selected entities are used for the Move selection set and you are prompted to select entities to use as reference curves.

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.



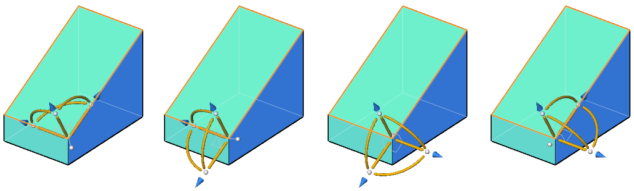
1.3.3 Undo per entity

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

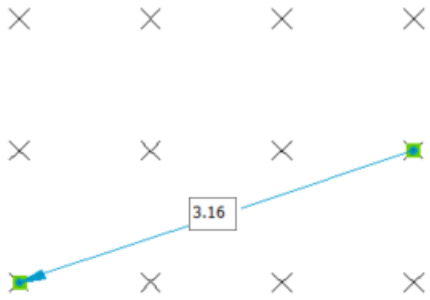
Press the SHIFT key to rotate the Manipulator 90 degrees about its normal axis.



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.

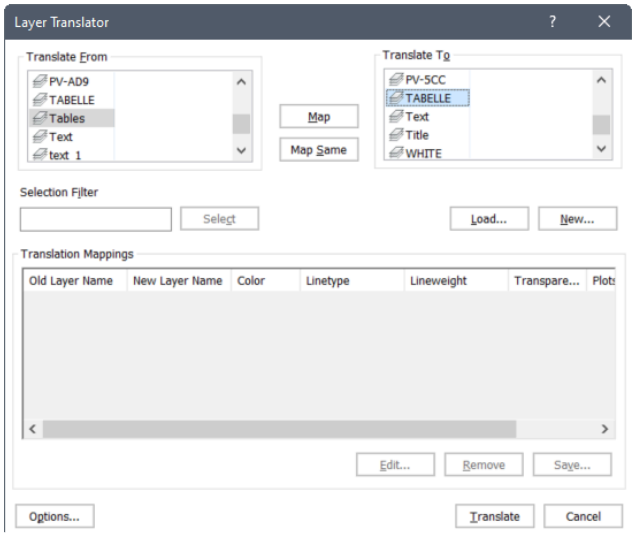


1.3.6 Layers

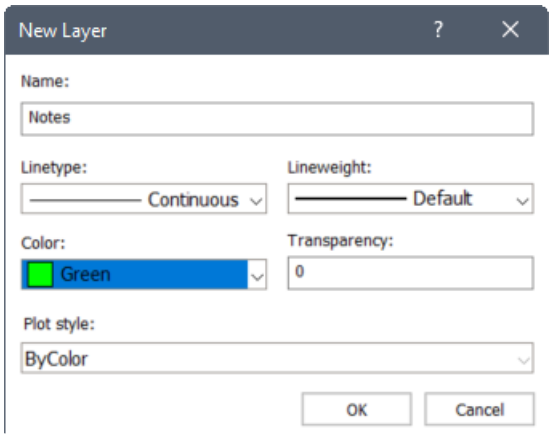
BricsCAD® V21 offers several layer enhancements to help increase your productivity.

Layer Translator

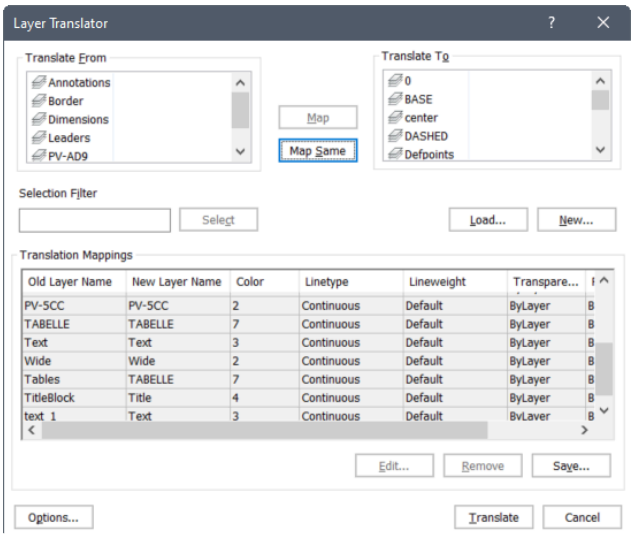
The new LAYTRANS command enables you quickly apply layer names and properties from another drawing file to the current drawing. When you launch the LAYTRANS command, the Layer Translator dialog displays. All the layer names in the current drawing are displayed in the Translate From list. You can specify 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.

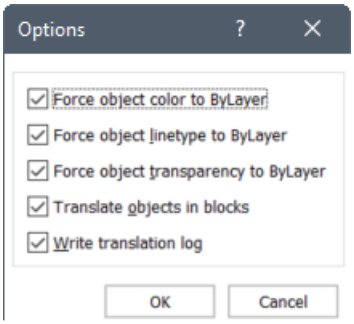


You can select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose **Map Same**, any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right.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

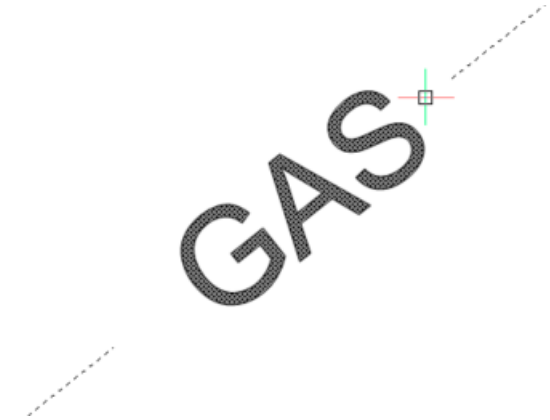
Layer properties by Viewport

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.

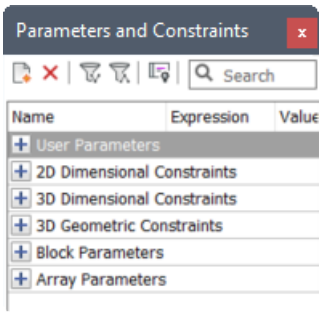


1.3.8 Parameters & Constraints

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

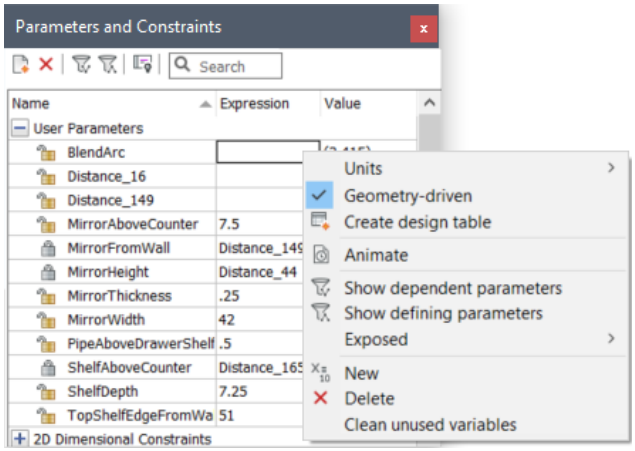
Parameters & constraints panel

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.

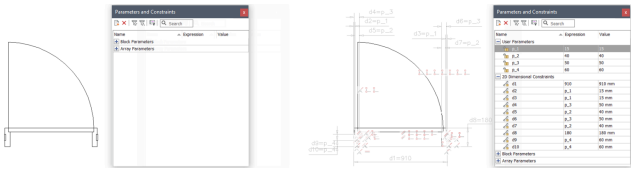


The right-click menu also enables you to control if a parameter is exposed for editing when inserted as a block.

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.

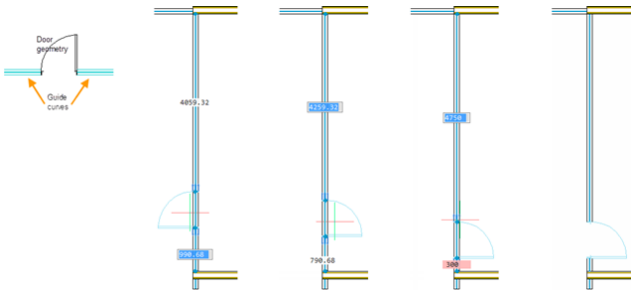


Reference curves

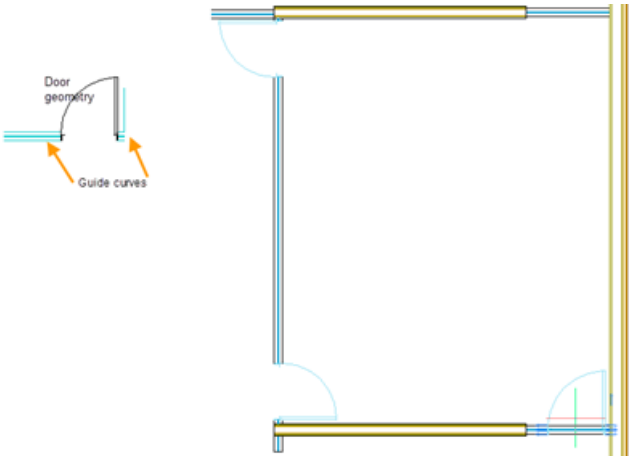
The new REFERENCECURVES command enables you to specify reference geometry for blocks. When you select geometry to use for reference, BricsCAD® creates a REFERENCE_CURVES layer (if it doesn't already exist) and adds the selected entities to that layer.

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.



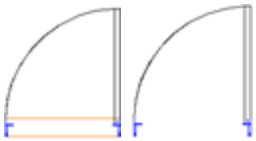
Visibility states

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.

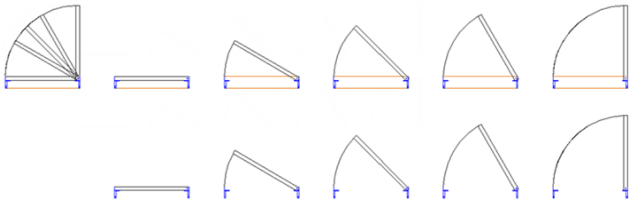
The SwingAngle parameter includes 5 visibility states: Closed, 30, 45, 60, and 90.



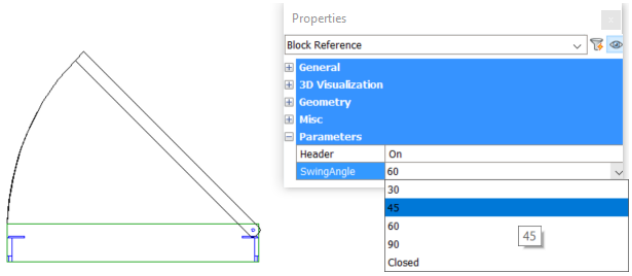
The Header parameter includes two visibility states: **On** and **Off**.



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.

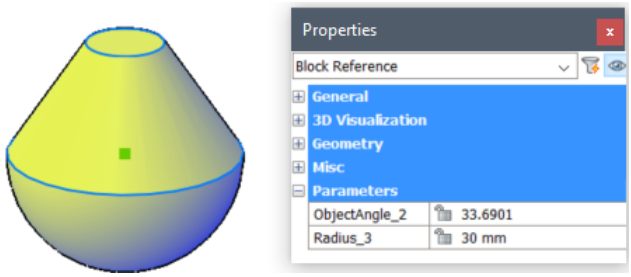


You can easily change the visibility states for a selected block insertion using the Properties panel.



3D Radius constraints

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

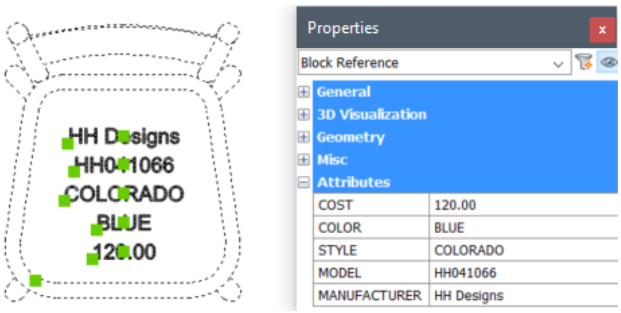


1.3.9 Blocks

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

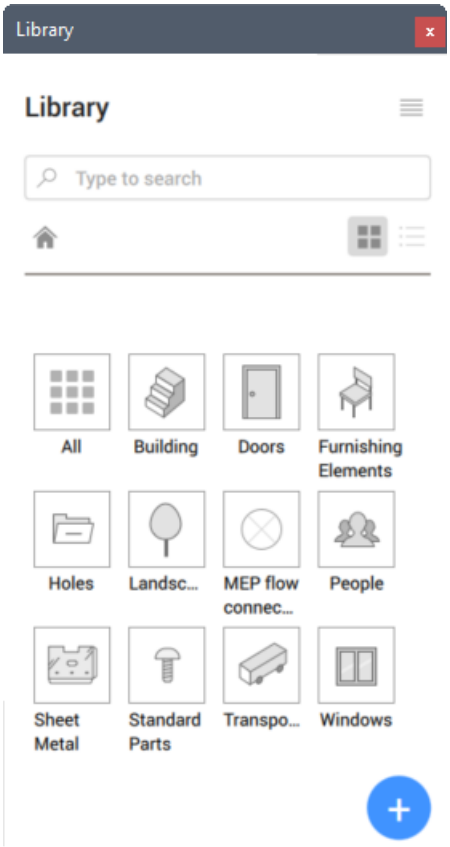
Block attribute properties

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.

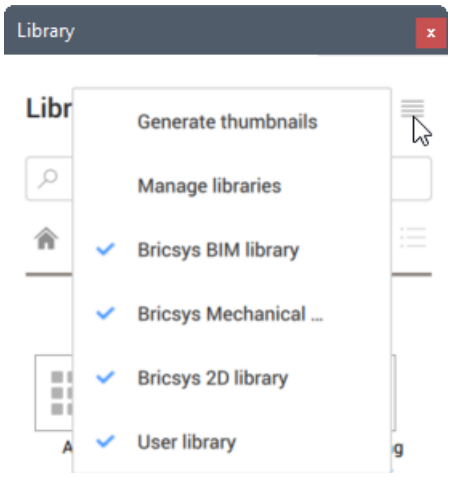


Library panel

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.

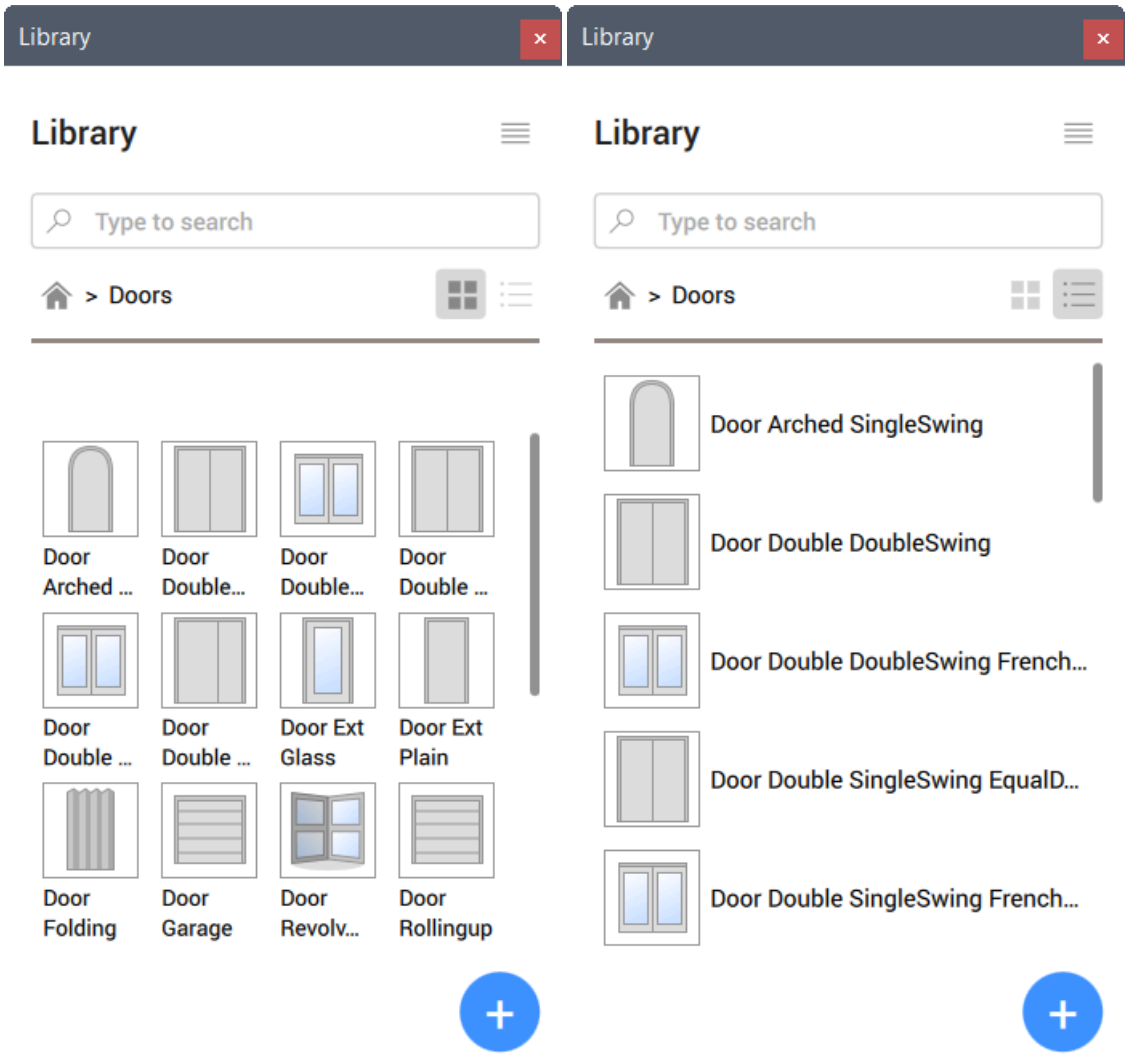


A menu in the upper right corner allows you to control which content is displayed in the Library panel. It may include pre-defined content from Bricsys libraries as well as the User Defined library with your own content.

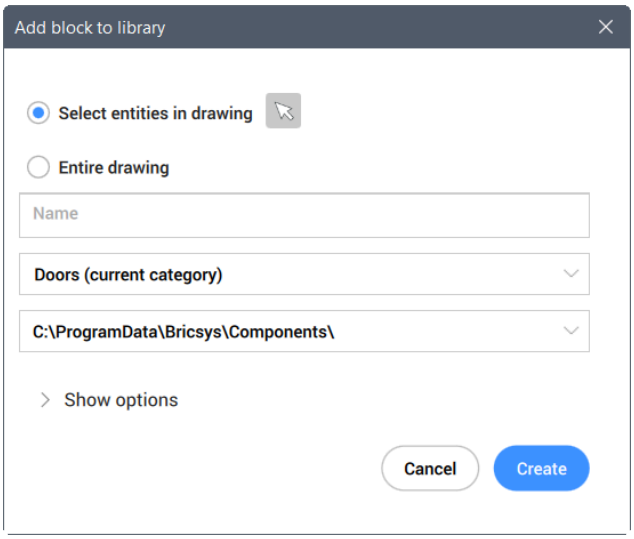


You can specify where BricsCAD® looks for library content by selecting the Manage libraries option. It offers quick access to the COMPONENTSPATH setting so you can view and modify library folder locations. Another menu option automatically generates thumbnail images for any that are missing.

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.

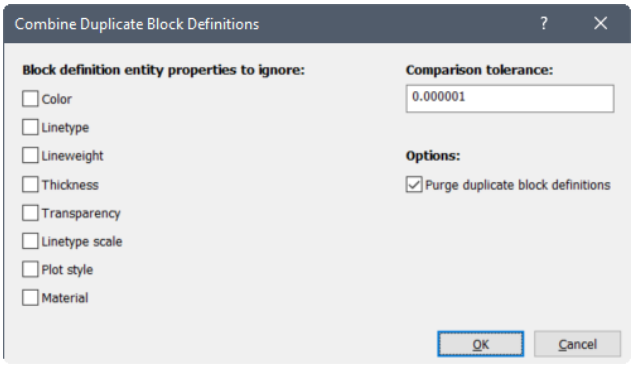
Audit

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

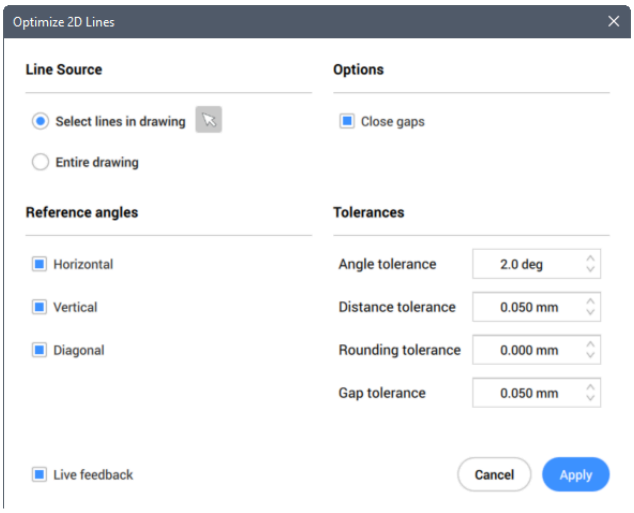
The OVERKILL command enables you to choose if you want to delete duplicate entities or move them to a specified layer. You can specify the layer name using the new OVERKILLLAYER setting or leave the default name, Duplicate Entities.

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.

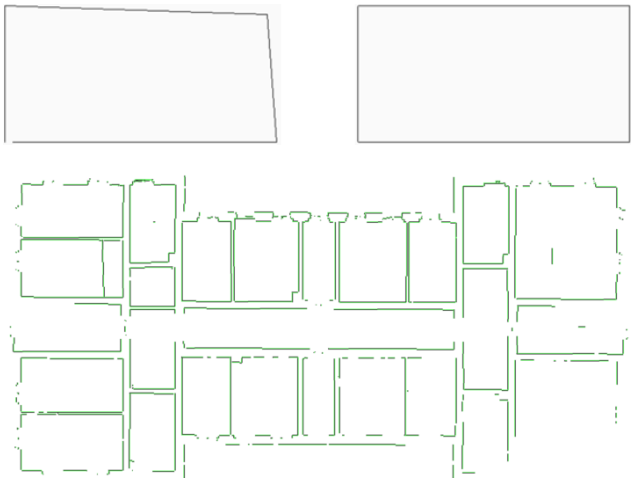


Optimize

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.

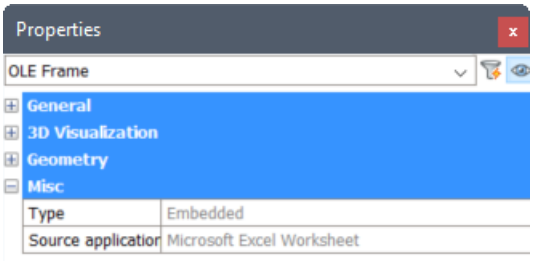


The Optimize tool is useful for cleaning up any 2D lines. For example, you can use it to quickly clean up line geometry extracted from point clouds.



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.

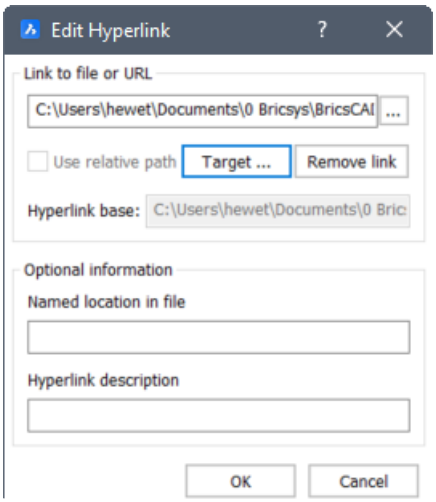


1.3.12 Images

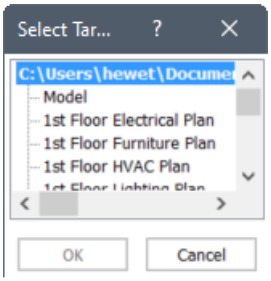
Image manipulation is enhanced in BricsCAD® V21. When you manipulate an attached image using grips and other editing tools the image content displays and updates as you edit the image frame. This enables you to position, align, and scale the image using the image content for reference. The selected image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

1.3.13 Hyperlinks

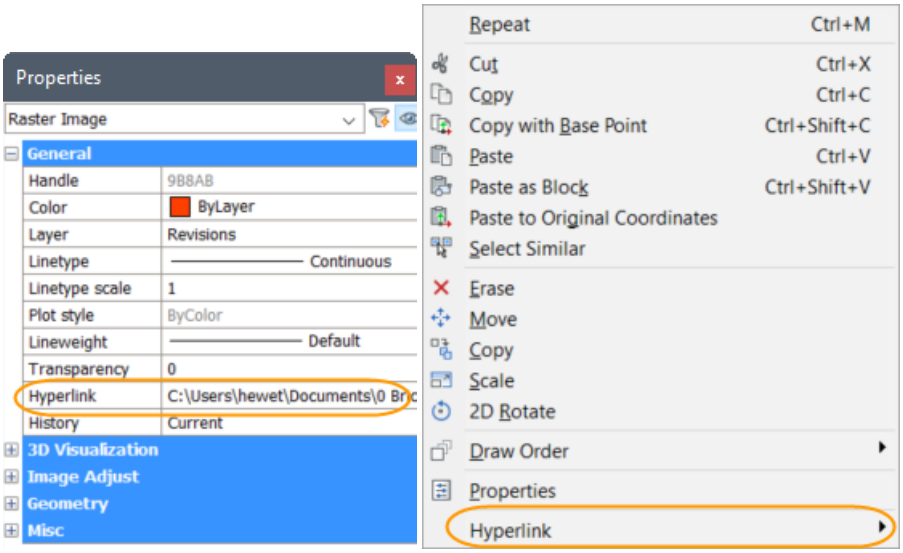
The Edit hyperlink dialog box now enables you to link to layouts and views inside a drawing. After selecting the drawing to which you want to link, choose **Target** to open the Select Target View dialog box.



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.

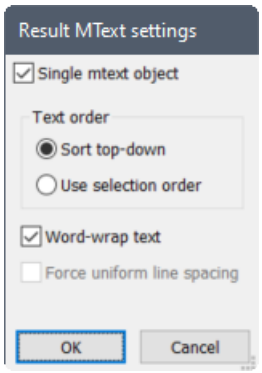


1.4 Drafting

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

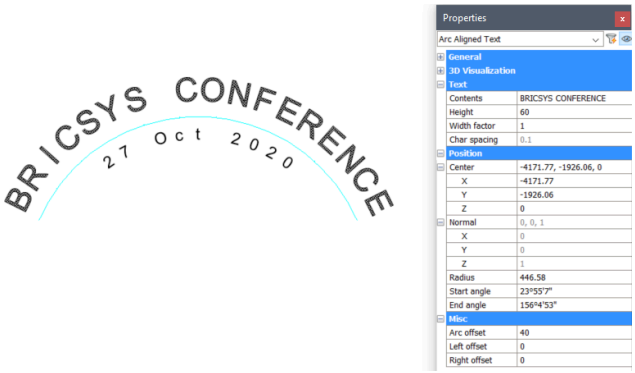
1.4.1 Combine Text to Mtext.

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.



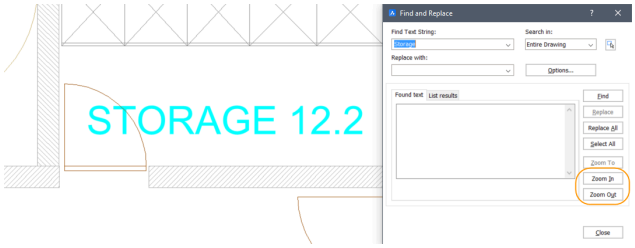
1.4.2 Arc Aligned Text

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



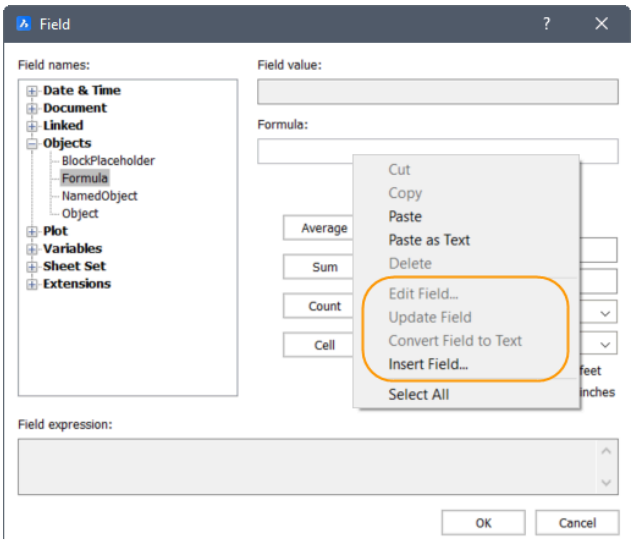
1.4.3 Find and Replace Text

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.

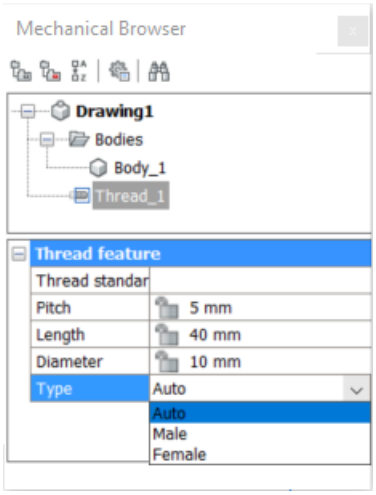
1.5.1 Dynamic UCS

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.

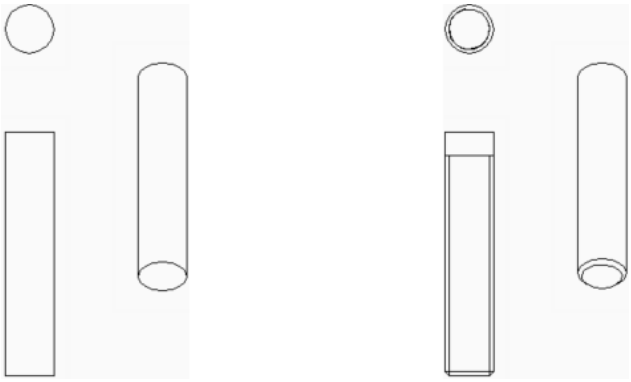
1.5.2 Direct modeling

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.

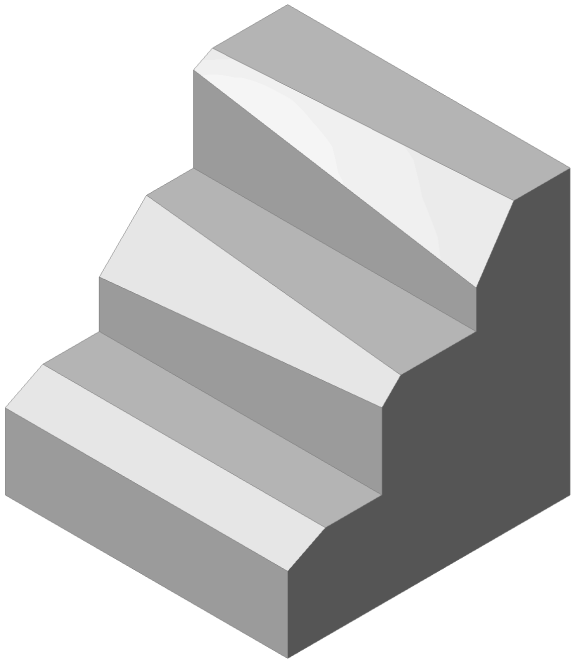


Drawing views created with the VIEWBASE command, the VIEWPROJ command or the VIEWSECTION command represent the threads following drafting standards. The following image show views of a cylinder before and after adding threads.



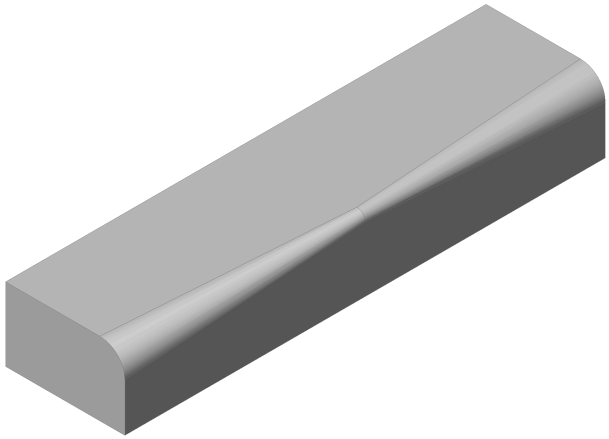
Chamfers

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.



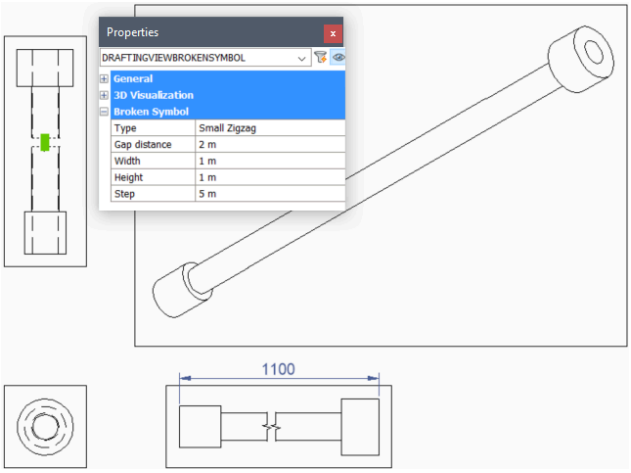
Push Pull

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.

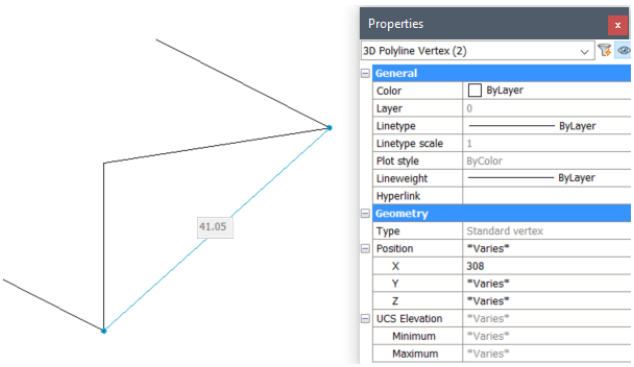


1.5.4 3D Entity snaps

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.

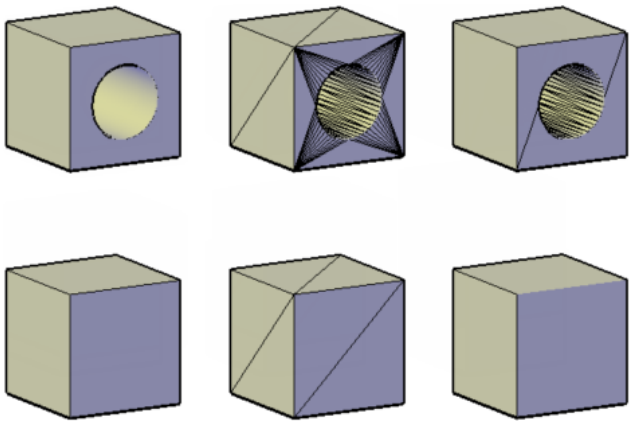
1.5.7 3D display

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

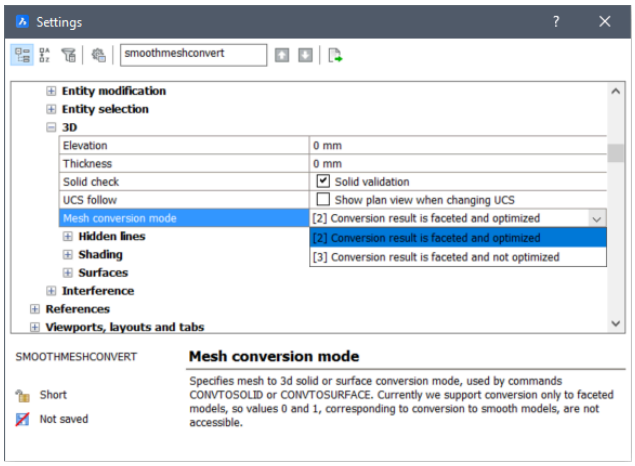
Convert to mesh

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).

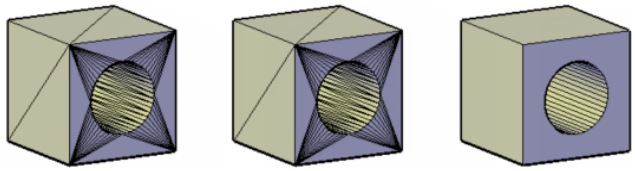


Convert to solid

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

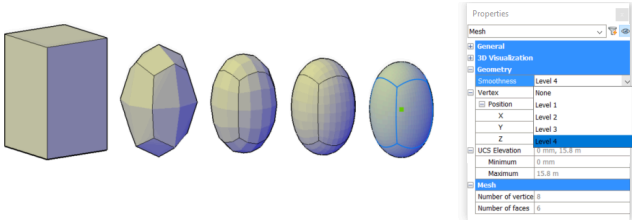


When set to optimized, coplanar and adjacent faces are merged into a single face. As an example, the original mesh on the left is converted to a solid in BricsCAD® V20 (center) and BricsCAD® V21 (right).

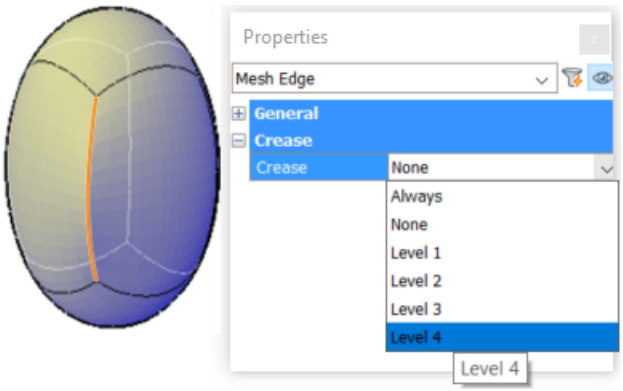


Mesh properties

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

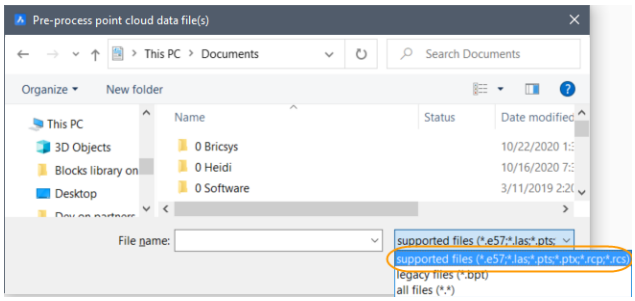
The Animation Editor is enhanced to support the animating of exploded views in model space.

1.5.12 Point clouds

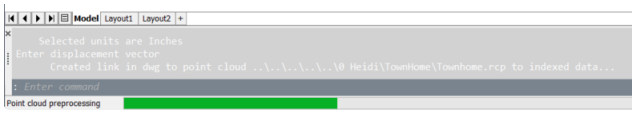
Point cloud functionality is significantly faster and more powerful in BricsCAD® V21.

Pre-processing

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



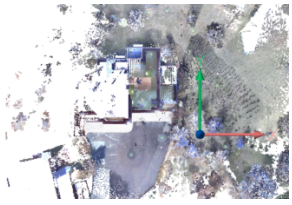
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD® V21 than in V20. A progress meter at the bottom of the display shows the conversion progress.



Alignment

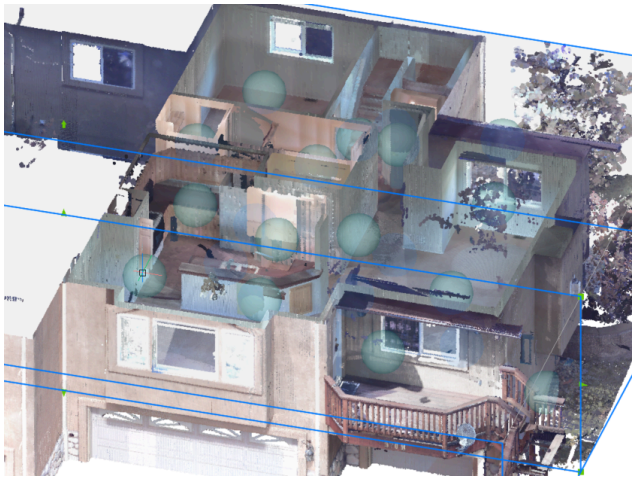
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment it can analyze the entire point cloud or you can specify the most relevant area.





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.



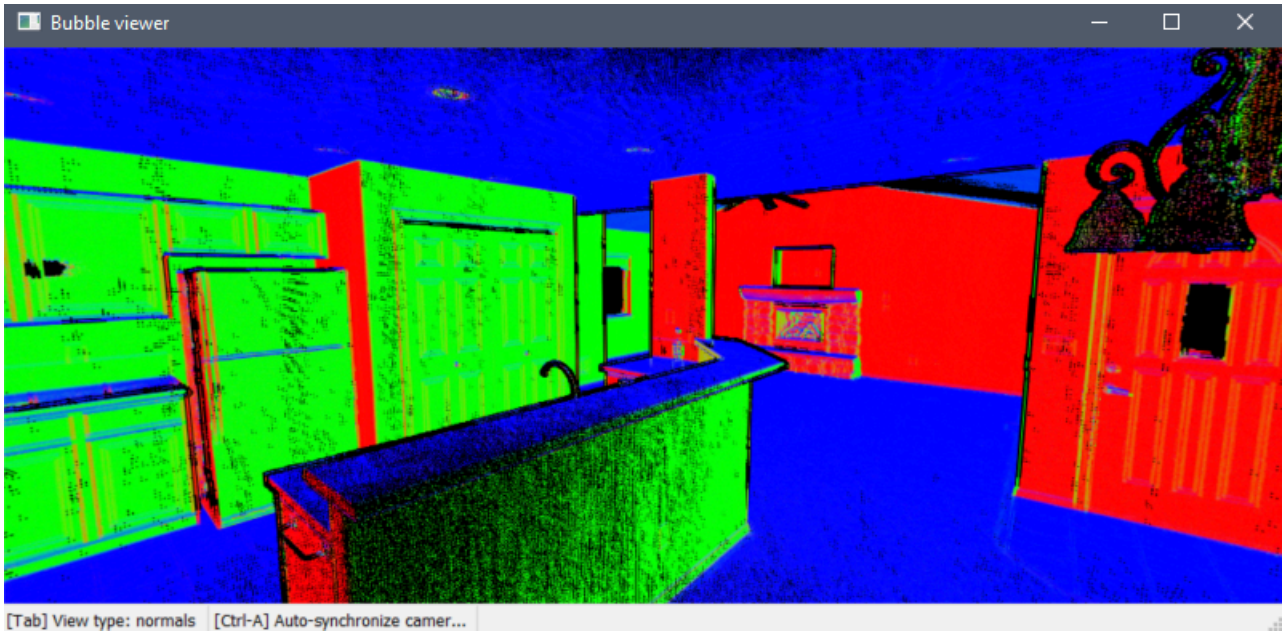


Press the Tab key to cycle between three different visual modes.

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

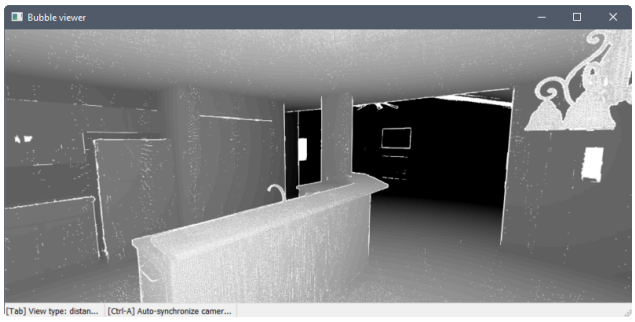


The second mode displays the points as red, green, or blue according to their normal vectors. The colors correspond to the UCS axes.





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.



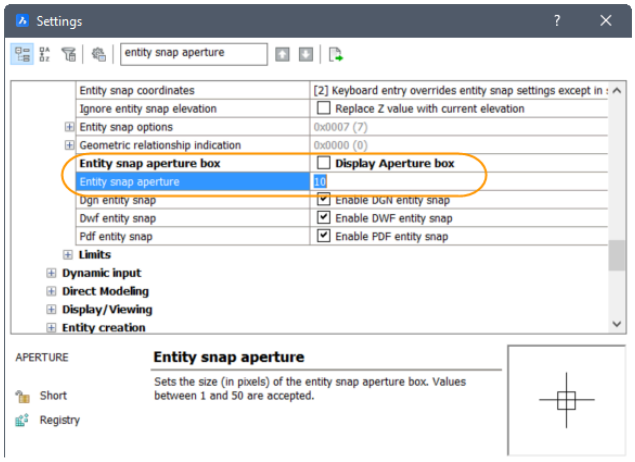
Entity snaps

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.

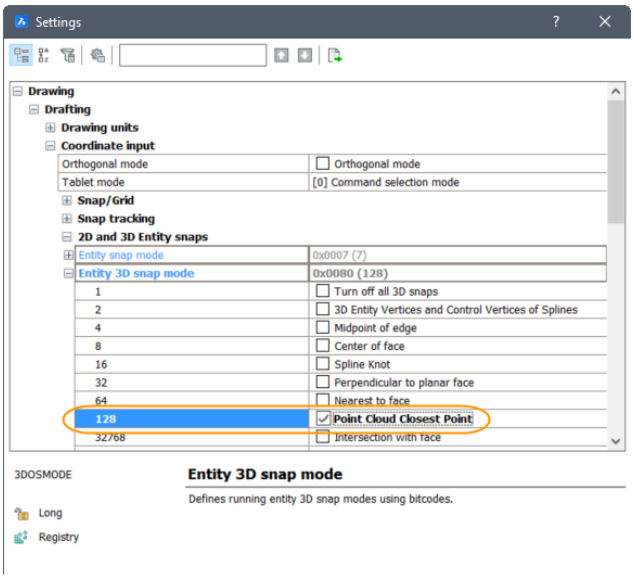




The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.



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

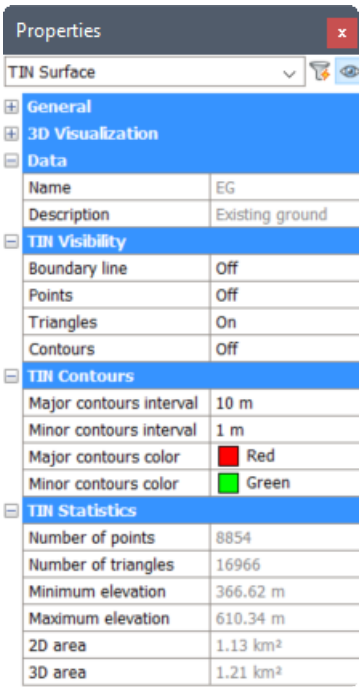
The new CIVIL3DIMPORT command enables you to create multiple civil entities in your BricsCAD® drawing, from an external Autodesk® Civil 3D drawing, in one operation.

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

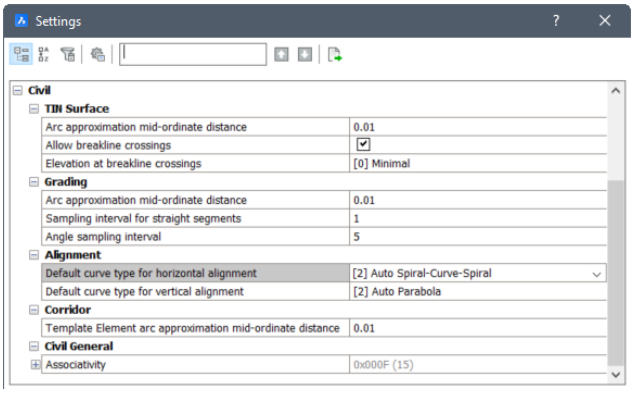
All Civil entities now include Name and Description properties in the Properties panel. Name and description are in the read-only mode in the Properties panel. They can be edited with the API.



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

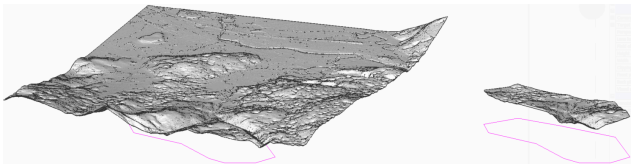
New and enhanced tools for creating and editing TIN surfaces provide more flexibility and a faster workflow.

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.

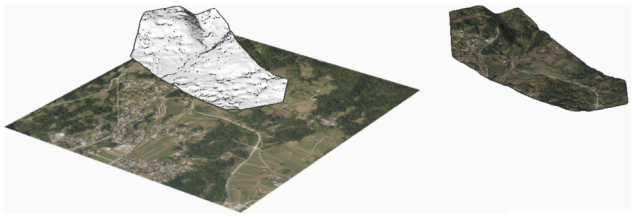
Clipping

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



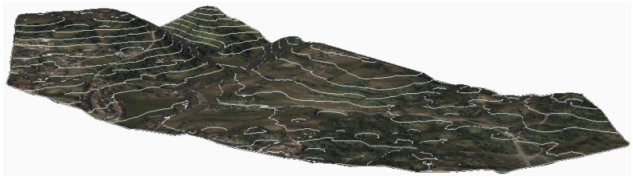
Raster Images

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

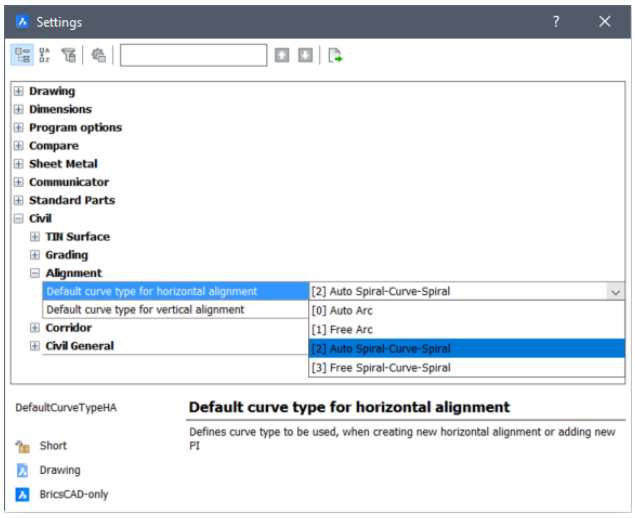
The new TINWATERDROP command enables you to draw a water drop's path from a point on a TIN Surface.

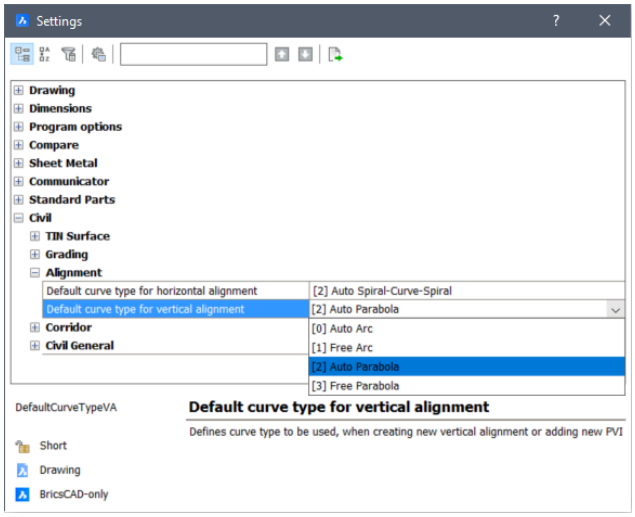


1.6.5 Alignments

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.



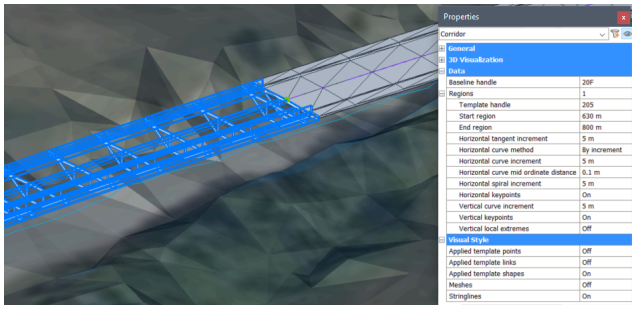
Then use the new CORRIDORTEMPLATE command to create a corridor template from template elements.



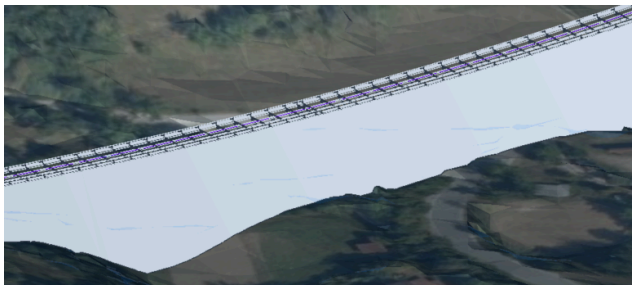
Finally, use the new CORRIDOR command to create a corridor from a corridor template, 3D alignment and/or TIN Surface. You can create multiple regions using different corridor templates. And, you can add or remove regions using the new CORRIDOREDIT command.



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

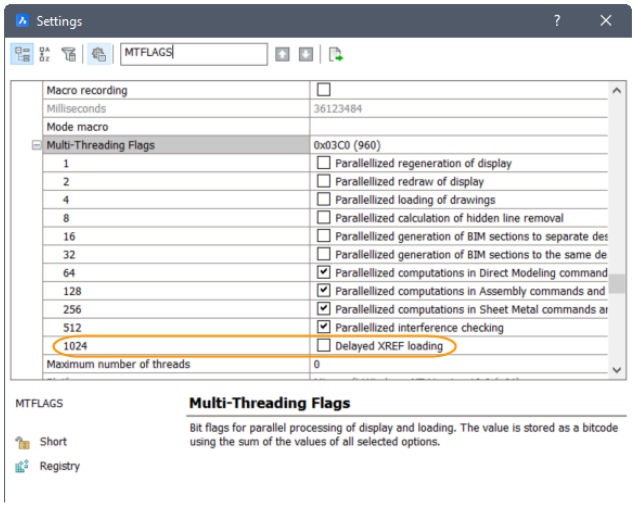
BricsCAD® V21 offers new and enhanced tools to improve performance and compatibility as you work individually or in a collaborative environment.

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.

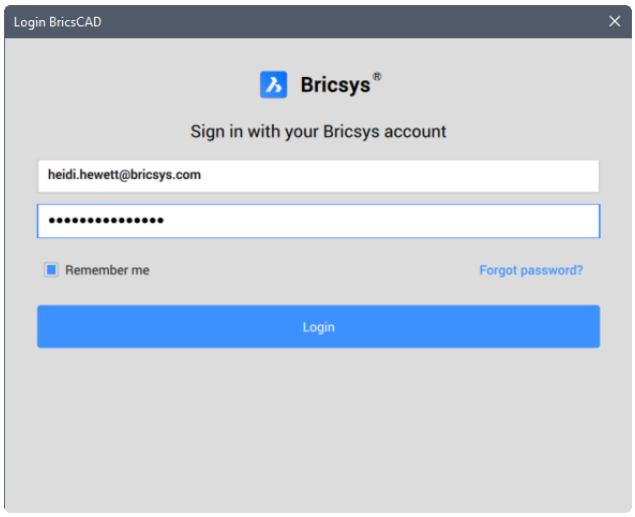
6 Choose Send Support Request.

VERSIONCONTROL Command

The new VERSIONCONTROL command offers the following options:

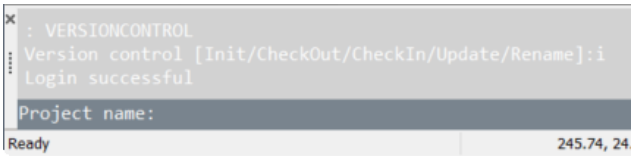
- Init
- Checkout
- Checkin
- Update
- Rename

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

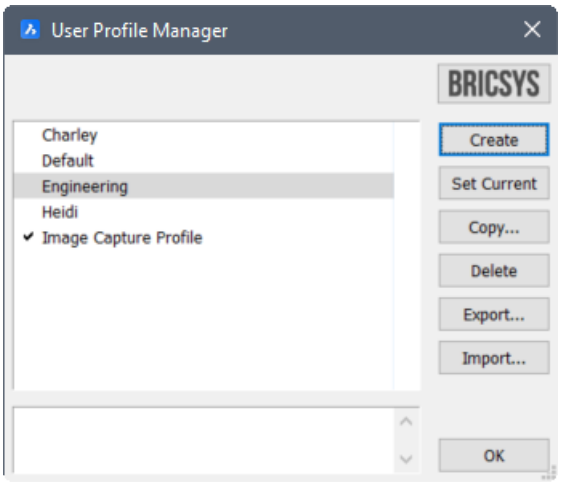
A new DGNEXPORT command enables you to export the current drawing to DGN file format.

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. Instalacja i licencjonowanie

2.1 Instalowanie BricsCAD®

2.1.1 Minimum system requirements and recommendations

System requirements and recommendations for Intel and AMD-based Microsoft Windows & Linux PCs, and Intel-based Apple Macs.

Supported Operating Systems	• Windows 10 (x64) • Windows 8.1 (x64) • Ubuntu V18.04 or higher • openSUSE builds, later than April 2018 • Fedora builds, later than April 2018 • Mac OSX 10.13 or higher	***Uwaga:*** Windows 8.1 will reach the end of Microsoft's Extended Support program on 10 January 2023. ***Uwaga:*** Mac OSX is supported only on Intel-based hardware at this time.
Deprecated Operating Systems	• Windows Server 2008 R2 • Windows Server 2008 (with SP2 or higher) • Windows 7 (x64)	***Uwaga:*** we do not recommend running mission-critical CAD software on unsupported operating systems, as your PC may become more vulnerable to security risks. • On 14 January 2020, Microsoft ended support for Windows Server 2008 and 2008 R2. • On 14 January 2020, Microsoft ended the Windows 7 Extended Support initiative.

Minimum CPU: • Intel® Core™ i5 • AMD Ryzen™ 5 • 2.5 GHz or higher	**Recommended CPU:** • Intel® Core™ i7 • Intel® Core™ i9 • AMD Ryzen™ 7 • AMD Ryzen™ 9 • 3.0 GHz or higher	The CPU with the highest single-thread performance rating will always excel with BricsCAD®. Additional CPU cores can be a bonus, because BricsCAD® supports the concept of 'helper threads' for many data- and calculation-intensive tasks. **Note about Intel® Xeon® and AMD Ryzen™ Threadripper™ CPUs**: these top-tier workstation CPUs support large on-chip memory cache, more PCIe® I/O lanes and multi-channel, high speed memory architectures. Will they help BricsCAD® run faster? Maybe. You will need to do testing in your specific environment, to determine if the additional cost of these systems is justifiable.
Minimum System Memory: 8 GB	**Recommended System Memory:** • 16 GB • 32 GB	For maximum performance, determine the memory configuration of your PC and ensure that each memory channel on the motherboard is populated with at least one memory module (SIMMs/DIMMs). ***Uwaga:*** the number of available memory channels may not correlate 1:1 with the number of RAM sockets on your PC's motherboard. Questions? Please check with your hardware supplier before you buy system memory.
Minimum Disk Space: 2 GB for BricsCAD® Ultimate, installed	**Recommended Disk Space:** (As needed)	We recommend Solid-State Drives (SSDs) as system drives, for both BricsCAD® and Windows. Drawings and support files can reside on slower mechanical hard disk drives, if necessary. ***Uwaga:*** solid-state disks can be up to 10x faster reading and 20x faster writing data than a mid-range hard disk drive (for example Seagate Barracuda 5.4k RPM, SATA 6.0).

Minimum Display System: • 1920 x 1080 True Color display and a graphics card with 1 GB VRAM • Apple Mac Displays	**Recommended Display System:** • Multiple 3840 x 2160 (4k) True Color displays, with a PCIe® graphics card w/ GPUs and 4 GB VRAM • Apple Mac Retina (4K and 5K) Displays	On Microsoft Windows systems, most graphics cards & GPUs from NVIDIA, AMD and Intel® are supported. On Linux systems, most graphics cards and GPUs from NVIDIA and AMD are supported. On Mac systems, native Apple display hardware is supported. BricsCAD®'s non-wireframe display modes are generated by RedSDK technology from Redway3D®. For maximum RedSDK performance in these modes, a GPU is required. ***Uwaga:*** We suggest that you install the latest drivers available for your display hardware, as recommended by the manufacturer. **Note to Linux users:** Hardware acceleration of 3D graphics on Linux systems is NOT supported on: • Intel® graphics chipsets / cards • Laptops with dual graphics adapters. However, recent driver developments by NVIDIA show promise in this area. For more information about supported graphics hardware for BricsCAD®, see the Redway3D GPU chipset reference page.

2.1.2 Pobieranie BricsCAD®

2.1.2.1 Pobieranie najnowszej wersjiBricsCAD®

1 Jeśli jeszcze tego nie zrobiłeś, wejdź na stronę Bricsys, zarejestruj się i zaloguj .

2 Kliknij **Pobierz BricsCAD** .

3 Wybierz platformę, język i wersję (32-bitową lub 64-bitową).

4 Zaznacz **Zgadzam się z warunkami użytkowania** pole wyboru.

5 Kliknij przycisk **Pobierz**.

2.1.2.2 Pobieranie starej wersji BricsCAD®

1 Jeśli jeszcze tego nie zrobiłeś, wejdź na stronę Bricsys, zarejestruj się i zaloguj .

2 Kliknij **Pobierz BricsCAD** .

3 Kliknij łącze **Pokaż stare wersje**.

4 Wybierz platformę i język.

5 Wybierz wersję **BricsCAD®**

6 Wybierz wersję (32-bitowa lub 64-bitowa).

7 Zaznacz **Zgadzam się z warunkami użytkowania** pole wyboru.

8 Kliknij przycisk **Pobierz**.

2.1.3 Przegląd opcji instalacji

- **BricsCAD®** jest dostarczane jako pojedynczy plik do pobrania dla wszystkich poziomów licencji, w tym **BricsCAD® Shape** .
- Gdy po raz pierwszy instalujesz nową **BricsCAD®** , możesz uruchomić 30-dniowy tryb próbny.
- Możesz zainstalować każdą nową wersję główną obok poprzedniej wersji głównej.

2.1.3.1 Instalacja interaktywna

W przypadku normalnej (cichej) instalacji użytkownik podaje niezbędne informacje dotyczące instalacji za pośrednictwem okien dialogowych.

Interfejs użytkownika Instalatora Windows wysyła zapytanie do systemu docelowego i wyświetla Kreatora instalacji. Kreator instalacji umożliwia użytkownikowi zmianę różnych opcji, które mają wpływ na instalację.

2.1.3.2 Cicha instalacja

W przypadku cichej lub cichej instalacji interfejs użytkownika Kreatora instalacji jest wyłączony, a działania podczas sekwencji interfejsu użytkownika nie są wykonywane.

2.1.3.3 Instalacja interaktywna

1 Kliknij dwukrotnie plik instalatora.

2 Kliknij **Dalej**.

3 Zaznacz pole wyboru, aby zaakceptować **Umowa licencyjna** .

4 Kliknij **Dalej**.

5 Wykonaj jedną z następujących czynności:
   - Zaakceptuj domyślny folder instalacyjny.
   - Kliknij przycisk **Zmień...** i wybierz folder instalacyjny.

6 Kliknij **Dalej**.

7 Kliknij **Zainstaluj**, aby rozpocząć instalację.

8 Wybierz, czy chcesz dodać skrót na pulpicie i / lub wyświetlać informacje o wersji po zakończeniu procesu instalacji.

9 Kliknij **Dalej**.

10 Kliknij**Zakończ**, aby zakończyć proces instalacji.

Result: **BricsCAD®** jest zainstalowane.

2.1.3.4 Cicha instalacja (tylko Windows)

Zanim rozpoczniesz: Do cichej instalacji potrzebne są uprawnienia administratora.

1 Otwórz **PowerShell** lub **wiersz poleceń** i zmień katalog (cd) na lokalizację instalatora **BricsCAD®**

pliku .msi.

Uwaga: W systemie Windows 10 możesz uzyskać dostęp do menu kontekstowego **Otwórz okno PowerShell tutaj**, klikając prawym przyciskiem myszy folder w **Eksploratorze**. Dzięki temu możesz otworzyć **PowerShell** w bieżącym oknie bez konieczności zmiany katalogu.

2 Wpisz **msiexec/i** i wykonaj jedną z następujących czynności:
   - Wpisz nazwę pliku instalatora **BricsCAD®** (na przykład BricsCAD-V20.2.09-1-en_US(x64).msi.
   - Przeciągnij instalator **BricsCAD®** z okna dialogowego Eksploratora Windows do okna poleceń.

3 Wpisz **/ qn** w wierszu polecenia.

4 (opcjonalnie) Dodaj **ADDDESKTOPSHORTCUT = ""**, aby pominąć dodanie skrótu na pulpicie.

5 (opcjonalnie) Dodaj **SHOWRELEASENOTES = ""**, aby wyłączyć wyświetlanie uwag do wydania po zakończeniu instalacji.

6 (opcjonalnie) Dodaj **APPLICATIONFOLDER="twój_folder_aplikacji"** , aby zainstalować **BricsCAD®** w folderze niestandardowym.

7 (opcjonalnie) Dodaj **BRXLICENSEDESTFILE = "ścieżka_do_lic_pliku"**, aby określić folder dla pliku licencji, jeśli są zdefiniowane jakiekolwiek właściwości **BRXLICENSEKEY** , **BRXLICENSEFILE** lub **BRXLICENSESERVER** .
Domyślnie jest to **APPLICATIONFOLDER\BricsCAD.lic**.

8 (opcjonalnie) Dodaj **BRXLICENSEKEY = "twój_klucz_licencji"**, aby aktywować licencję i zapisać ją w folderze określonym przez **BRXLICENSEDESTFILE**.

9 (opcjonalnie) Dodaj **BRXLICENSEFILE = "twój_plik_licencji"**, aby skopiować plik licencji do folderu określonego przez **BRXLICENSEDESTFILE**.

10 (opcjonalnie) Dodaj **BRXLICENSESERVER = "host"** lub **BRXLICENSESERVER = "port@host"** (*), aby utworzyć plik licencji w folderze określonym przez **BRXLICENSEDESTFILE**, zawierającym specyfikację hosta serwera.
(*) Jeśli Twój serwer licencji używa innego portu niż domyślny port 5053.

11 (opcja tylko w wersji 32-BIT) Dodaj **INSTALLVBA = "0"** lub **INSTALLVBA = ""**, aby nie instalować VBA.
Pełny wpis w wierszu poleceń może wyglądać następująco: **msiexec / i "BricsCAD-Vxx.x.xx-x-en_US (x64) .msi" / qn ADDDESKTOPSHORTCUT = "" SHOWRELEASENOTES = ""**.

12 Naciśnij klawisz **Enter**, aby rozpocząć instalację.

Result:

Uwaga:

- Aby uzyskać więcej opcji, wpisz: **msiexec /?** w wierszu poleceń.
- W witrynie firmy Microsoft można znaleźć więcej informacji na temat polecenia **msiexec**.

2.1.4 Tryb próbny

BricsCAD® może działać w trybie próbnym przez 30 dni. Po tym okresie musisz aktywować ważną licencję komercyjną.

2.1.4.1 Uruchamianie BricsCAD w trybie próbnym

Za każdym razem, gdy zaczynasz **BricsCAD®** w trybie próbnym, **BricsCAD - Bezpłatny okres próbny** , wskazując pozostałe dni.

Wykonaj jedną z następujących czynności:

- Kliknij przycisk **Kontynuuj** , aby otworzyć **BricsCAD®** w trybie próbnym.
- Kliknij przycisk **Wprowadź licencję** ... , aby aktywować licencję pojedynczą lub zbiorczą albo użyć licencji sieciowej .

2.1.4.2 Ustawianie poziomu licencji

Po uruchomieniu **BricsCAD®** w trybie próbnym dostępne są wszystkie funkcje. Działa jako licencja **BricsCAD® Ultimate**

Polecenie SETLICENSELEVEL umożliwia wybranie niższego poziomu licencji.

2.2 Aktywacja i licencjonowanie BricsCAD®

2.2.1 Przegląd opcji licencjonowania

BricsCAD® oferuje różne opcje licencjonowania.

2.2.1.1 Poziomy licencji

Istnieje 5 poziomów licencji dla **BricsCAD®** :

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

1 BricsCAD® Lite

Jeśli tworzysz tylko ogólne projekty 2D, **BricsCAD® Lite** może być dla Ciebie właściwym wyborem. W porównaniu z **AutoCAD LT®** , umożliwia także przeglądanie, przenoszenie, kopiowanie lub odbijanie lustrzane modeli 3D. Nie ma jednak możliwości edycji modelu ani tworzenia nowych elementów 3D.**BricsCAD® Lite** uruchamia również procedury Lisp. Lisp umożliwia **BricsCAD®** użytkownikom ładowanie niestandardowych procedur lub małych aplikacji, które będą działać w **BricsCAD®** .

2 BricsCAD® Pro

BricsCAD® Pro obejmuje bezpośrednie modelowanie 3D z pełnym systemem wiązań 3D, renderowanie z materiałami i oświetleniem oraz dostęp do aplikacji innych firm.

3 BricsCAD® BIM

BricsCAD® BIM zawiera wszystkie **BricsCAD® Pro** funkcje. Możesz wykorzystać swoje obecne umiejętności CAD, aby płynnie przejść do tworzenia rzeczywistych modeli informacji o budynku w rekordowym czasie. Jeden produkt, jeden przepływ pracy, wszystko w standardowym formacie DWG.

4 BricsCAD® Mechanical

Oprócz wszystkich funkcji **BricsCAD® Pro BricsCAD® Mechanical** zawiera kompletny zestaw narzędzi do projektowania mechanicznego, wszystko w branżowym formacie DWG, w tym funkcjonalność arkusza blachy.

5 BricsCAD® Ultimate

BricsCAD® Ultimate łączy cechy **BricsCAD® BIM** i **BricsCAD® Mechanical**

2.2.1.2 Typ licencji:

Istnieją 2 typy licencji dla **BricsCAD®** :

- Dożywotnia licencja.
- Licencja czasowa.

1 Dożywotnia licencja

Kupując licencję wieczystą, jesteś właścicielem oprogramowania.**BricsCAD®** edycje działają w systemach Microsoft Windows, macOS i Linux.

Istnieją 2 opcje maintenance:

- Opcja Maintenance obejmuje automatyczne aktualizacje i wsparcie.
- Możesz kupić aktualizacje według własnego uznania.

2 Licencja czasowa

Wybierając opcję wypożyczenia, masz dostęp do najnowszej wersji **BricsCAD®** przez 1 rok. Obejmuje priorytetowe wsparcie i dostęp do wszystkich nowych wersji, które zostaną wydane w czasie, gdy subskrypcja jest aktywna.

2.2.1.3 Dodatkowe opcje licencji

Dodatkowe opcje licencji obejmują licencje akademickie, licencje dla jednego użytkownika, sieciowe i zbiorowe.

1 Licencja akademicka

Studenci, wydziały, instruktorzy i instytucje edukacyjne mają bezpłatny dostęp do całego**BricsCAD®** oprogramowania BricsCAD przez 12 miesięcy. Możesz ją co roku odnawiać za pomocą aktualnej legitymacji studenckiej. Zarejestruj się jako uczeń lub szkoła / nauczyciel, korzystając ze swoich danych akademickich.

2 Licencja dla jednego użytkownika

Umowa licencyjna dla jednego użytkownika dotyczy jednego użytkownika. Każda licencja dla jednego użytkownika umożliwia 2 aktywacje. Możesz go zainstalować na dwóch komputerach, na przykład na stacji roboczej w biurze i na laptopie w terenie. Jednak jednocześnie może być używana tylko jedna licencja. Licencje dla jednego użytkownika są dostępne w przypadku licencji wieczystych lub licencji czasowych. Licencje akademickie są zawsze licencjami dla jednego użytkownika.

3 Licencja sieciowa

Umowa licencyjna sieciowa umożliwia wielu użytkownikom dostęp **BricsCAD®**do BricsCAD w sieci LAN (Local Area Network). Liczba dostępnych licencji sieciowych określa liczbę użytkowników, którzy mogą jednocześnie korzystać z oprogramowania. Opcja licencji sieciowej jest dostępna w przypadku licencji wieczystej lub czasowej i obejmuje roczną umowę serwisową. Licencje sieciowe są również dostępne dla instytucji korzystających z licencji akademickiej.

4 Licencja wielostanowiskowa

Umowa licencjonowania zbiorowego umożliwia korzystanie z **BricsCAD®** w ramach zdefiniowanej bazy użytkowników. Oprogramowanie można zainstalować przy użyciu jednego klucza licencyjnego ważnego dla każdego użytkownika. Opcja licencji zbiorczej jest dostępna w przypadku licencji wieczystej lub wynajmu i obejmuje roczną umowę serwisową. Jest to ustawienie domyślne dla instytucjonalnych licencji akademickich.

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

Drawings							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
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	√	√	√	√	√	√	√

Interface							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Command Line (Autocomplete, Midstring, Clickable options)	√	√	√	√	√	√	√
Start tab	√	√	√	√	√	√	√
File (Drawing) tabs	√	√	√	√	√	√	√
Model and Layout tabs	√	√	√	√	√	√	√
Layout Manager	√	√	√	√	√	o	o
Menu bar	√	√	√	√	√	√	√
Toolbars	√	√	√	√	√	√	√
Ribbon	√	√	√	√	√	√	√
Tool Palettes	√	√	√	√	√	√	√
Contextual Right-click Menus	√	√	√	√	√	√	√

Interface							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Contextual Quad Menu	√	√	√	√	√	o	o
Status Bar	√	√	√	√	√	√	√
Workspaces	√	√	√	√	√	√	√
Searchable Settings (Variables) Dialog Box	√	√	√	√	√	o	o
Lockable Interface	√	√	√	√	√	√	√
Clean Screen Mode	√	√	√	√	√	√	√
Drawing Explorer	√	√	√	√	√	o	o
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	√	√	√	√	√	√	√
Zintersection 3D snap	√	√	√	√	√	o	o
Nearest Distance	√	√	√	√	√	o	o
Adaptive Gridsnap	√	√	√	√	√	o	o
Real-time boundary detection	√	√	√	√	√	o	o
2D Dynamic UCS	√	√	√	√	√	o	o

Selection, snap & track							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
3D Dynamic UCS	√	√	√	√	√	o	√

Drawing tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Line, Polyline, 3D Polyline, Spline, Ray, Xline	√	√	√	√	√	√	√
Multiline and Multiline Styles	√	√	√	√	√	o	√
Circle, Arc, Ellipse, Elliptical Arc	√	√	√	√	√	√	√
Polygon, Rectangle, Region, Donut, Point	√	√	√	√	√	√	√
Helix	√	√	√	√	√	o	√

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

Annotation tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Hatches, Gradients, Hatch/Gradient Editor	√	√	√	√	√	√	√
Wipeouts	√	√	√	√	√	√	√
Revision Clouds	√	√	√	√	√	√	√
Attributes, Attribute Editor	√	√	√	√	√	√	√
Tables, Table Styles	√	√	√	√	√	√	√
Fields	√	√	√	√	√	√	√
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	√	√	√	√	√	√	√
CopyGuided	√	√	√	√	√	o	o
MoveGuided	√	√	√	√	√	o	o
Grip Edit	√	√	√	√	√	√	√
Properties panel	√	√	√	√	√	√	√
Manipulator (3D Gizmo)	√	√	√	√	√	o	√
Drawordder	√	√	√	√	√	√	√
Undo Per Entity (Property History)	√	√	√	√	√	o	o

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

Reference tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
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

Visualization tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Lights	o	√	√	√	√	o	√
Sun Properties	o	√	√	√	√	o	√
Materials	o	√	√	√	√	o	√
Render	o	√	√	√	√	o	√
Animation Editor	o	√	√	√	√	o	o
Walkthrough navigation	o	√	√	√	√	o	√
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	√

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

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Extract Edges	o	√	√	√	√	o	√
Direct modeling	o	√	√	√	√	o	√
Deformable (Freeform) Modeling	o	√	√	√	√	o	√
Audit, Simplify	√	√	√	√	√	o	√
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

Modeling tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Section planes	o	√	√	√	√	o	√
Drawing Views	o	√	√	√	√	o	√
3D Threads	o	√	√	√	√	o	o
Point Cloud Align	o	√	√	√	√	o	o
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

Civil tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Corridors	o	√	√	√	√	o	o
Civil 3D Object Enabler	√	√	√	√	√	o	√
Architectural Desktop Object Enabler	√	√	√	√	√	o	√
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	√

Architectural tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Site, Building and Stories	o	o	√	o	√	o	√
Windows and Doors	o	o	√	o	√	o	√
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

Architectural tools							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Point Cloud Geometry Detection	o	o	√	o	√	o	o
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
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

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

Import, export, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
PC3 Printer Configuration Files	√	√	√	√	√	√	√
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	√	√	√	√	√	√	√

Import, export, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
RVT Import	o	o	√	o	√	o	o
DWF Export	√	√	√	√	√	√	√
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	√

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

Programming							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
TX (Teigha eXtensions)	√	√	√	√	√	o	o
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	√	√	√	√	√	√	√

Licensing							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Full
Network Multi-user	√	√	√	√	√	o	o
Volume (Perpetual)	√	√	√	√	√	o	o
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 Wersja próbna: aktywacja online

Zanim rozpoczniesz: **BricsCAD®** wymaga połączenia z Internetem.

1 Uruchom **BricsCAD®**.

2 Kliknij **Rozpocznij próbę** w oknie dialogowym **BricsCAD Launcher**.

Result: Licencja próbna jest aktywowana przez Internet.

2.2.4 Wersja próbna: aktywacja offline (ręczna)

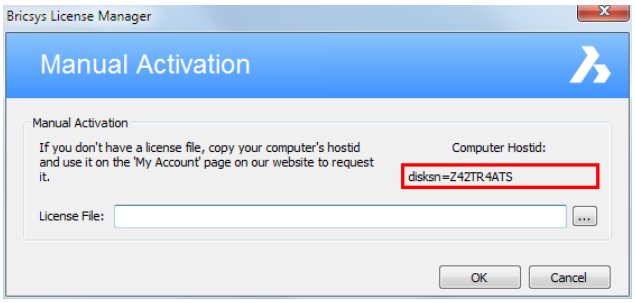
Jeśli Twój komputer nie jest połączony z Internetem, możesz aktywować licencję ręcznie za pomocą innego komputera podłączonego do Internetu.

1 Uruchom **BricsCAD®** .

2 Kliknij **Aktywuj licencję** w oknie dialogowym BricsCAD Launcher .

3 Kliknij**Activate Manually...** w oknie dialogowym **Menedżer licencji Bricsys** .

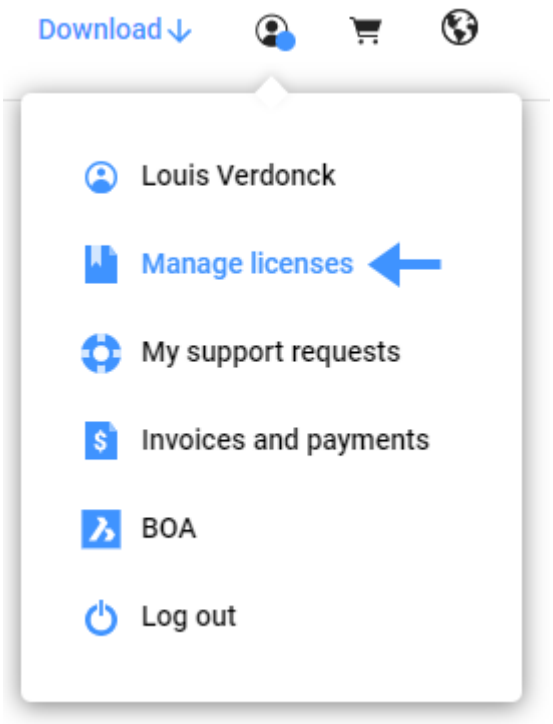
Result: Otworzy się okno dialogowe **Aktywacja ręczna**.



4 Skopiuj pełną zawartość pola **Computer Hostid** .

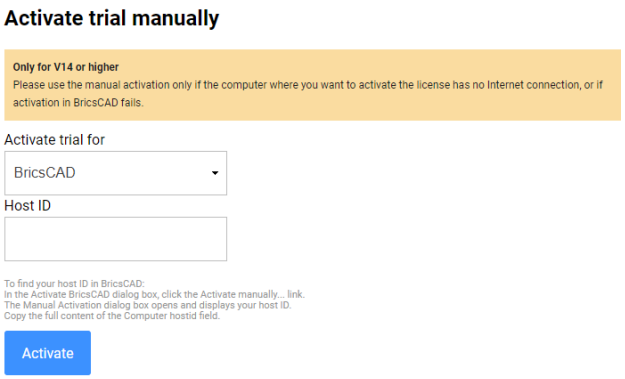
5 Zaloguj się na stronie Bricsys na komputerze podłączonym do internetu.

6 Kliknij ikonę **Moje konto** ( ) i wybierz z menu opcję **Zarządzaj licencjami** .



7 Wybierz opcję **Aktywuj BricsCAD ręcznie** .

8 W polu **Identyfikator hosta** wklej identyfikator hosta komputera, na którym chcesz zainstalować **BricsCAD®** a następnie kliknij przycisk **Aktywuj** .



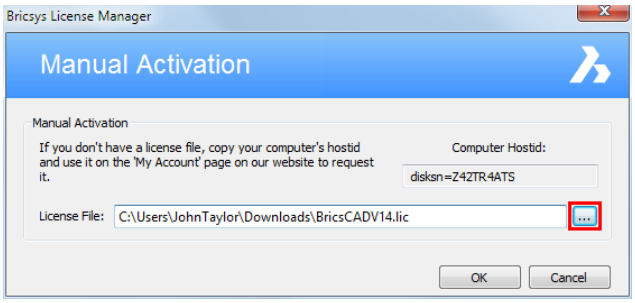
Result: Plik licencji jest generowany i udostępniany do pobrania.

9 Kliknij przycisk **Pobierz plik licencji** .

Result: Plik licencji jest pobierany do domyślnego folderu pobierania.

10 W oknie dialogowym **Aktywacja ręczna** kliknij przycisk **przeglądania** (...).

11 Otwórz folder, do którego skopiowałeś plik licencji, i wybierz plik.



12 Kliknij przycisk **OK** .

Result: Uruchomiono **BricsCAD®**

2.2.5 Licencja pojedyncza lub zbiorcza: aktywacja online

Zanim rozpoczniesz: **BricsCAD®** wymaga połączenia z Internetem.

1 Uruchom **BricsCAD®**.

Uwaga: Po wygaśnięciu okresu próbnego kliknij **Aktywuj licencję** na **Launcher BricsCAD** i przejdź do kroku 5.

2 Uruchom polecenie LICENSEMANAGER.

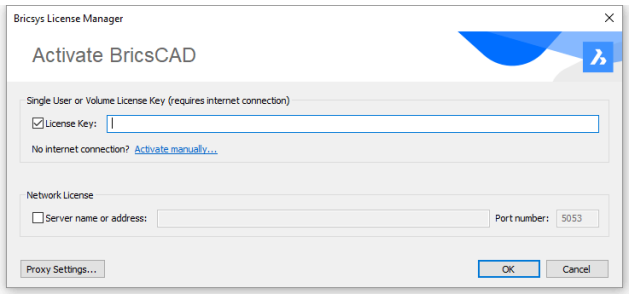
Result: Zostanie wyświetlone okno dialogowe Menedżer licencji BricsCAD.

3 Kliknij **Zarządzaj licencją**.

Result: Wyświetlone zostanie okno dialogowe **Informacje licencyjne**.

4 Kliknij przycisk **Modyfikuj**.

Result: ten **Menedżer licencji BricsCAD** > **Aktywuj BricsCAD** wyświetla się okno dialogowe:



5 Wklej swój klucz licencyjny w polu **Klucz licencyjny**.

6 Kliknij przycisk **OK** .

Result:

- Twój klucz licencyjny jest rejestrowany przez Internet.
- Po pomyślnej aktywacji klucza licencyjnego zostanie wyświetlone okno dialogowe.

7 Przeczytaj tekst w oknie dialogowym i kliknij przycisk **Rozumiem**.

2.2.6 Licencja pojedyncza lub zbiorcza: aktywacja offline (ręczna)

Jeśli Twój komputer nie jest połączony z Internetem, możesz aktywować licencję ręcznie za pomocą innego komputera podłączonego do Internetu.

Uwaga: Nie można dezaktywować licencji aktywowanych ręcznie.

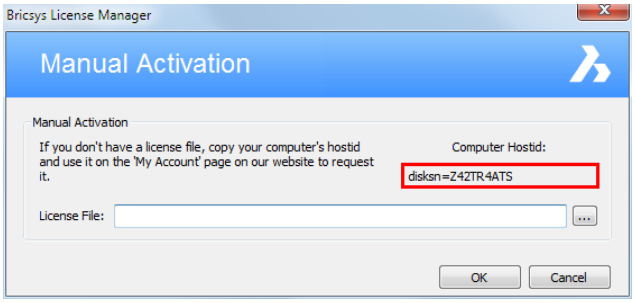
1 Uruchom **BricsCAD®** .

2 Wykonaj jedną z następujących czynności:

- Jeśli zaczynasz od **BricsCAD®** po raz pierwszy, przejdź do następnego kroku.
- Jeśli uruchomisz **BricsCAD®** w trybie próbnym, kliknij **Wprowadź licencję...** w oknie dialogowym **Bezpłatna wersja próbna**
- Jeśli wymieniasz lub aktualizujesz istniejącą licencję, uruchom polecenie LICENSEMANAGER , a następnie kliknij **Zarządzaj licencją** i kliknij przycisk **Modyfikuj** ...

3 Kliknij **Aktywuj ręcznie...** w oknie dialogowym **Aktywuj BricsCAD**.

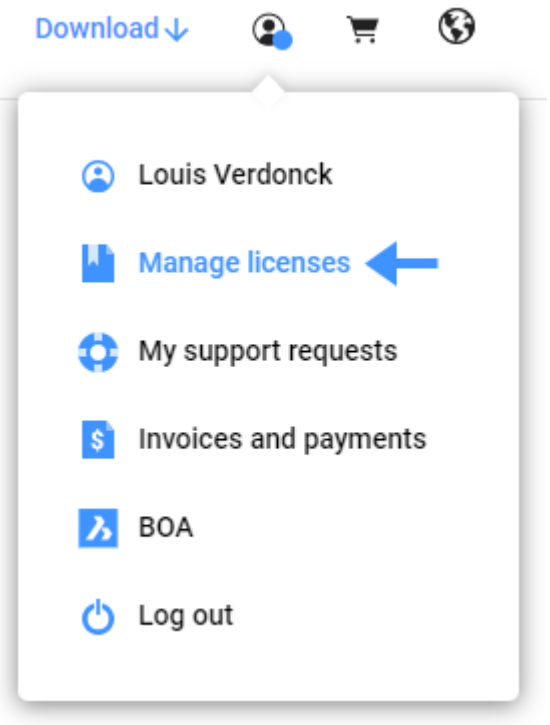
Result: Otworzy się okno dialogowe **Aktywacja ręczna**.



4 Skopiuj pełną zawartość pola **Computer Hostid** .

5 Zaloguj się na stronie Bricsys na komputerze podłączonym do internetu.
Upewnij się, że logujesz się na konto Bricsys, które zawiera Twoje licencje.

6 Kliknij ikonę **Moje konto** ( ) i wybierz z menu opcję **Zarządzaj licencjami**.



7 Najedź kursorem na klucz licencyjny, który chcesz aktywować i kliknij ikonę ołówka ( ).



Result: Wyświetlane są **szczegóły licencji** .

8 Kliknij opcję **Aktywacja ręczna** .

9 W polu **Identyfikator hosta** wklej identyfikator hosta komputera, na którym chcesz zainstalować **BricsCAD®** .

10 Kliknij przycisk **Aktywuj** .

Result: Plik licencji jest generowany i udostępniany do pobrania.

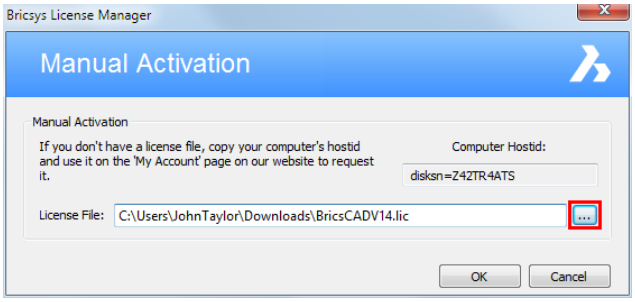
11 Kliknij przycisk **Pobierz plik licencji** .

Result: Plik licencji **BricsCADVxx.lic** jest pobierany do domyślnego folderu pobierania.

12 Skopiuj plik licencji na komputer, na którym chcesz zainstalować **BricsCAD®** .

13 W oknie dialogowym **Aktywacja ręczna** kliknij przycisk przeglądania ( ). Otwórz folder, do którego

skopiowałeś plik licencji **BricsCADVxx.lic** i wybierz plik.



14 Kliknij przycisk **OK** .

Result: Uruchomiono **BricsCAD®**

2.2.7 Korzystanie z licencji sieciowej na komputerze Klienta

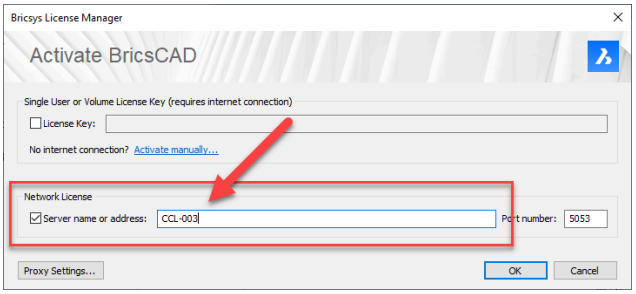
Po aktywowaniu licencji sieciowej na serwerze licencji można z niej korzystać na komputerach klienckich.

1 Wykonaj jedną z następujących czynności:

- Jeśli uruchamiasz **BricsCAD®** po raz pierwszy, kliknij przycisk **Aktywuj licencję** w oknie dialogowym BricsCAD Launcher.
- Jeśli wymieniasz lub aktualizujesz istniejącą licencję, uruchom polecenie LICENSEMANGER, kliknij **Zarządzaj licencją** i kliknij **Modyfikować** .

2 Zaznacz opcję **Licencja sieciowa** w oknie dialogowym **Aktywuj BricsCAD**.

3 Wpisz nazwę hosta lub adres IP serwera licencji w polu **Nazwa lub adres serwera**.



4 (opcjonalnie) Wpisz numer portu w polu **Numer portu**.

5 Jeśli komputer jest połączony z serwerem licencji przez serwer proxy, kliknij przycisk **Ustawienia proxy**, aby skonfigurować ustawienia proxy dla licencji sieciowej.

6 Kliknij przycisk **OK** .

2.3 Konfigurowanie serwera licencji sieciowej

2.3.1 Serwer licencji sieciowej

Aby udostępnić licencję sieciową w BricsCAD**BricsCAD®** na komputerze klienta, administrator sieci musi najpierw zainstalować serwer licencji sieciowej z Reprise Software.

Menedżer Licencji Sieciowej Bricsys to osobny program, który instaluje serwer. Po uruchomieniu serwera licencji sieciowej można go skonfigurować przez interfejs WWW.

Tak jak**BricsCAD**® BricsCAD potrzebuje pliku licencji do uruchomienia, tak serwer licencji sieciowej potrzebuje pliku licencji sieciowej, który możesz aktywować online lub ręcznie.

Licencję sieciową można aktywować tylko raz i jest ona przypisana do serwera.

Po aktywacji licencji sieciowej i uruchomieniu serwera, możesz użyć licencji sieciowej **BricsCAD**®w BricsCAD na komputerze klienta.

Uwaga: Reprise Software nie oferuje serwera licencji sieciowej dla komputerów Mac. Możesz jednak zainstalować serwer licencji sieciowej Windows lub Linux, aby obsługiwać licencje na komputerach klienckich Mac.

2.3.2 Krótki przewodnik dotyczący korzystania z licencji sieciowej

Przede wszystkim upewnij się, że w proces jest zaangażowany administrator sieci/IT.

2.3.2.1 1. Akcje maszyny działające na serwerze

1 Pobierz Bricsys Network License Manager i zainstaluj z domyślnymi ustawieniami:

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

Informacje związane z polityką bezpieczeństwa serwera :

po instalacji, RLM.exe musi mieć możliwość korzystania z 3 portów: **5053** dla licencji, **5054** dla interfejsu internetowego, **dynamicznie przydzielanego portu** przez system Windows dla serwera ISV, który można ustawić jako naprawiony po aktywacji, jak opisano w artykule Konfiguracje zaawansowane.

2 Kiedy pojawi się okno aktywacji, aktywuj używając klucza licencyjnego z Twojego konta Bricsys.

Informacje związane z polityką bezpieczeństwa serwera :

aplikacja aktywatora jest *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager \actNetworkLicense.exe* i musi być w stanie dotrzeć do **license.bricsys.com/actpro** port **80**

3 Strona administracyjna menedżera licencji RLM znajduje się na **lokalnym hoście: 5054**

2.3.2.2 2. Akcje maszyny klienta

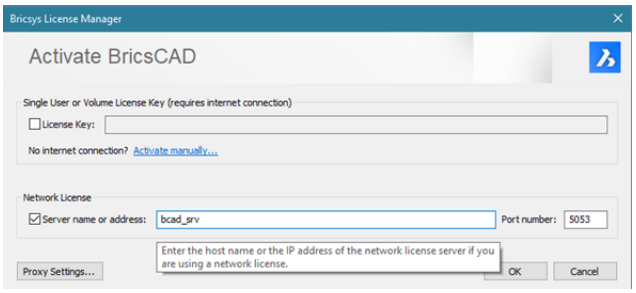
Maszyna klienta musi być:

- W tej samej **sieci LAN** z maszyną RLM, fizycznie lub przez połączenie VPN.
- Potrafi komunikować się z maszyną RLM na porcie **5053** za **pomocą** protokołu TCP.

1 Zainstaluj **BricsCAD**® z ustawieniami domyślnymi.

2 Uruchom **BricsCAD**® i aktywuj licencję sieciową: <nazwa serwera>/<numer ip> i <port>

(może nie być takiej potrzeby, ponieważ **BricsCAD**® sprawdzi sieć pod kątem serwera licencji na początku)



2.3.3 Konfigurowanie serwera licencji sieciowej Windows

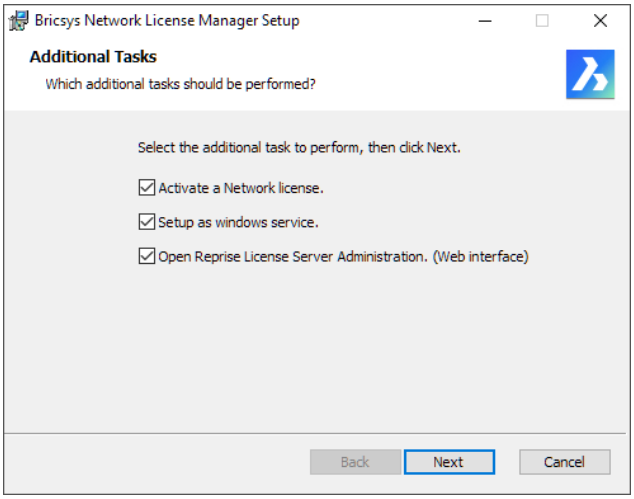
2.3.3.1 Pobieranie Menedżera licencji sieciowej Bricsys

Pobierz Menedżera licencji sieciowej Bricsys (Windows) z następującego łącza:

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

2.3.3.2 Nowa instalacja Bricsys Network License Manager

W **Dodatkowe zadania** w oknie dialogowym, upewnij się, że zaznaczyłeś wszystkie pola wyboru (zobacz obraz Konfiguracja Menedżera Licencji Sieciowej Bricsys).



- **Aktywuj licencję sieciową**

  Po instalacji, **Bricsys Network License Manager** otwiera się i pyta o aktywację licencji sieciowej. Możesz otworzyć **Menedżera Licencji Sieciowej Bricsys** później, klikając skrót Aktywacji Licencji w menu Start.
- **Skonfiguruj jako usługę Windows**

  Za każdym razem, gdy uruchamiasz system Windows, serwer licencji jest automatycznie uruchamiany jako usługa. Możesz to zrobić później za pomocą następującego polecenia w wierszu polecenia z podwyższonym poziomem uprawnień: rlm.exe-dlog rlm.log-install_service
- **Administracja serwera licencji Open Reprise (interfejs internetowy)**

Po zakończeniu instalacji otworzy się interfejs WWW Administracji serwera licencji Reprise. Pozwala on na zarządzanie licencją sieciową. Administrację Serwera Licencji Reprise można otworzyć później, klikając skrót Administracja serwera licencji w menu Start.

2.3.3.3 Aktualizacja Menedżera Licencji Sieciowej Bricsys

1 Pobierz i zainstaluj Bricsys Network License Manager na komputerze używanym jako serwer licencji sieciowej.

2 Upewnij się, że odznaczyłeś wszystkie pola wyboru obok **Dodatkowe zadania** .

Uwaga: Instalacja restartuje serwer licencji.

2.3.3.4 Instalacja z wiersza poleceń

Menedżera Licencji Sieciowej można również uruchomić z okna poleceń.

Domyślnie opcje **Aktywuj licencję sieciową**, **Zainstaluj jako usługę Windows** i **Otwórz Administrację Serwera Licencji Reprise** (interfejs WWW) są wyłączone podczas instalacji z wiersza poleceń.
Aby włączyć te opcje, użyj odpowiednich parametrów ACTIVATELICENSE, INSTALLSERVICE i OPENWEBINTERFACE.

Na przykład **msiexec /qb /i Bricsys-NetworkLicenseManager.msi ACTIVATELICENSE=1 INSTALLSERVICE=1** wyświetli okno dialogowe aktywacji i zainstaluje jako usługę bez wyświetlania interfejsu internetowego.

Uwaga:

- Aby uzyskać więcej opcji, wpisz: **msiexec /?** w wierszu poleceń.
- W witrynie firmy Microsoft można znaleźć więcej informacji na temat polecenia **msiexec**.

2.3.4 Konfigurowanie serwera licencji Linux

Zanim rozpoczniesz: Pobierz i zainstaluj Bricsys Network License Manager na komputerze, który będzie używany jako serwer licencji sieciowej.

1 Pobierz Menedżera licencji sieciowej Bricsys (Linux)

2 Rozpakuj **Bricsys-NetworkLicenseManager.tgz** do dowolnego folderu (wiersz poleceń: tar xvzf Bricsys-NetworkLicenseManager.tgz).

3 Uruchom aplikację rlm w tym folderze (wiersz poleceń w tym folderze: ./rlm).

4 Przejdź do strony internetowej **http://localhost:5054** .

5 Kliknij przycisk **System Info** po lewej stronie na witrynie internetowej. Skopiuj ID hosta Ethernet i użyj go do aktywacji licencji na stronie **Moje konto** na stronie Bricsys.

6 Kiedy pobierasz plik licencyjny ze strony Bricsys, przechowuj go w folderze, w którym wyodrębniony został Bricsys-NetworkLicenseManager.tgz.

7 Kliknij przycisk **Wczytaj ponownie/Uruchom ponownie serwery**.

8 Kliknij przycisk **Status**, aby zobaczyć status serwera, na następnej stronie kliknij przycisk **bricsys** w tabeli, aby wyświetlić licencje Bricsys.

9 Kliknij przycisk **RLM Manual...**, aby dowiedzieć się, jak szczegółowo skonfigurować serwer licencji. Strona wyjaśnia między innymi, jak uruchomić rlm automatycznie po uruchomieniu serwera.

Result: Po aktywacji licencji sieciowej i uruchomieniu serwera, jest on gotowy do obsługi licencji. Teraz można używać licencji sieciowej w **BricsCAD®** na komputerze klienckim.

2.3.5 Licencja sieciowa: aktywacja online

Aktywacja licencji **BricsCAD®** wymaga połączenia z Internetem.

1 Kliknij skrót **License activation** w menu **Start** systemu operacyjnego.

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

Otworzy się okno dialogowe **Bricsys License Manager - Activate Network License** .

2 Wprowadź nową licencję sieciową w polu **Klucz licencyjny** i kliknij **OK**.

3 Kliknij skrót do **administrowania serwerem licencji** w menu **Start** systemu operacyjnego, aby otworzyć **Administrację serwera licencji Reprise**.

4 Kliknij opcję **Reread / Restart Servers** w lewym menu.

2.3.6 Licencja sieciowa: aktywacja offline (ręczna)

Jeśli Twój komputer nie jest połączony z Internetem, możesz aktywować licencję ręcznie za pomocą innego komputera podłączonego do Internetu.

Uwaga: Nie można dezaktywować licencji aktywowanych ręcznie.

1 Kliknij skrót **Aktywacja licencji** w menu **Start** systemu operacyjnego.

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

Otworzy się okno dialogowe **Bricsys License Manager - Activate Network License** .

2 Skopiuj pełną zawartość pola **Computer Hostid** .

3 Zaloguj się na Stronie Bricsys na komputerze podłączonym do internetu.

4 Kliknij ikonę **Moje konto** ( ) i wybierz z menu opcję **Zarządzaj licencjami** .

Upewnij się, że logujesz się na konto Bricsys swojej firmy, które zawiera licencję sieciową. W razie potrzeby skontaktuj się z opiekunem klienta Bricsys.

5 Najedź kursorem na klucz licencyjny, który chcesz aktywować i kliknij ikonę ołówka ( ).

Wyświetlane są **szczegóły licencji** .

6 Kliknij opcję **Aktywacja ręczna** .

7 W polu **Host ID** wklej identyfikator hosta komputera, na którym chcesz zainstalować program **BricsCAD®**.

8 Kliknij przycisk **Aktywuj** .

Plik licencji jest generowany i udostępniany do pobrania.

9 Kliknij przycisk **Pobierz plik licencji** .

Plik licencji **BricsCADVxx.lic** jest pobierany do domyślnego folderu pobierania.

10 Skopiuj plik licencji na serwer, do folderu instalacyjnego **Bricsys Network License Manager** (domyślnie: *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager*).).

11 Przeprowadź **ponowne wczytanie/restart serwera** z interfejsu WWW RLM: localhost: 5454, aby menedżer licencji Reprise używał rozszerzenia pliku .lic.

2.3.7 Administracja serwerem licencji sieciowej

Gdy serwer licencji sieciowej (nazywany również serwerem licencji RLM) jest uruchomiony, można go skonfigurować w interfejsie WWW Administracji serwera licencji Reprise.

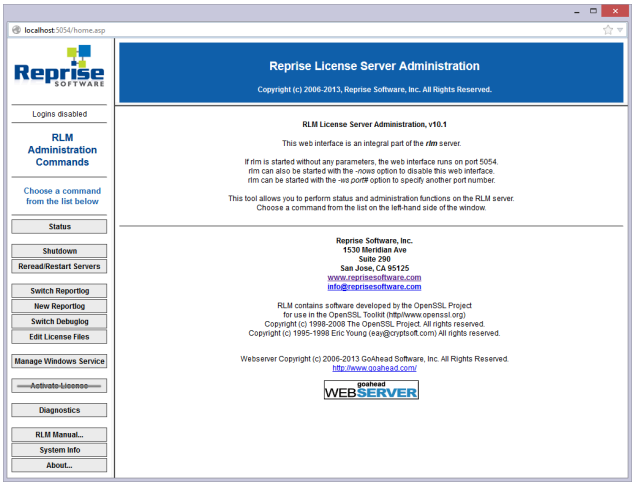
2.3.7.1 Otwarcie interfejsu internetowego

1 Jeśli serwer licencji sieciowej nie jest uruchomiony, kliknij skrót **Serwer Licencji** w menu Start (w systemie Windows Vista lub nowszym: kliknij prawym przyciskiem myszy > Uruchom jako administrator).

Result: Otworzy się okno konsoli.

2 Przejdź do **http://localhost:5054** .

Result: Zostanie otwarty interfejs WWW Administracji serwera licencji Reprise.



Uwaga:

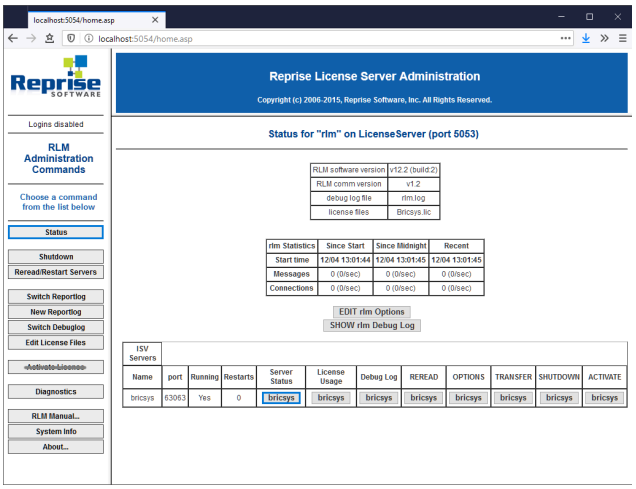
- Nie używaj przycisku **Aktywuj licencję**!
- Aby uzyskać przegląd wszystkich opcji, kliknij **Instrukcja RLM...** przycisk po lewej stronie lub przejdź do Podręcznik administrowania licencjami RLM.

2.3.7.2 Sprawdzanie licencji i statusu serwera

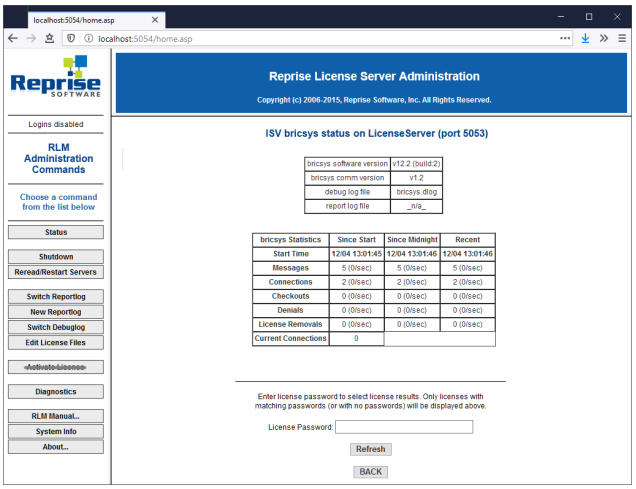
Możesz sprawdzić stan wszystkich licencji sieciowych na serwerze.

1 Kliknij przycisk **Stan** po lewej stronie.

Result: Zostanie otwarta strona **Status dla „rlm" na [LicenseServer] (port 5053)**.



2 Kliknij przycisk **bricsys** w kolumnie **Server Status** tabeli Serwery ISV, aby zobaczyć status licencji sieciowej Bricsys.

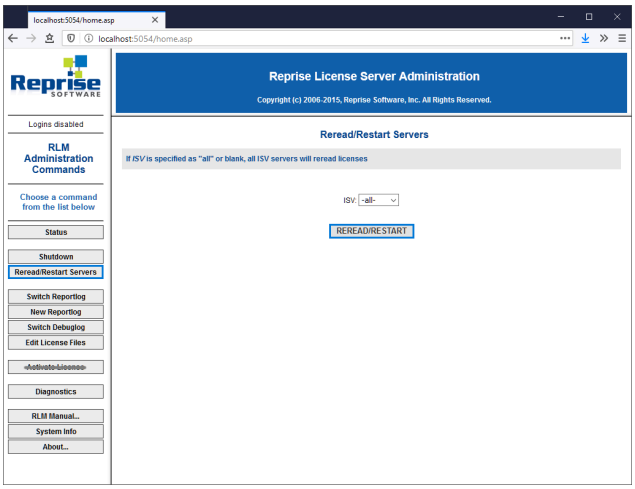


2.3.7.3 Ponowne wczytanie licencji / Ponowne uruchomienie serwera

Po aktywowaniu licencji sieciowej, gdy serwer był uruchomiony, należy ponownie wczytać licencję / zrestartować serwer.

1 Kliknij przycisk **Wczytaj ponownie/Uruchom ponownie serwery** po lewej stronie.

Result: Otworzy się strona **Odczytaj ponownie/Uruchom ponownie serwery.**



2 Kliknij przycisk **Wczytaj ponownie/Uruchom ponownie serwery**.

Result: Serwer uruchomi się ponownie i ponownie wczyta licencję sieciową.

2.3.8 Zaawansowane konfiguracje

Po skonfigurowaniu serwera licencji sieciowej **BricsCAD**® może zgłosić błąd -17 podczas próby połączenia z serwerem.

Przyczyną może być obecność oprogramowania antywirusowego i/lub zapory i/lub serwera proxy.

2.3.8.1 Antywirus

Jeśli na serwerze działa oprogramowanie antywirusowe i blokuje program serwera licencji (**rlm.exe**w systemie Windows, **rlm** w systemie Linux/Mac), należy odblokować usługę RLM w programie antywirusowym.

Zobacz swoją konfigurację antywirusową, jak odblokować usługi.

2.3.8.2 Zapora

Jeśli firewall jest zainstalowany pomiędzy serwerem a **BricsCAD**®, dwa numery portów muszą być otwarte w firewallu: numer portu 5053 i numer portu przypisany do serwera licencji Bricsys.

Ten numer portu jest przypisywany dynamicznie podczas uruchamiania serwera licencji. Aby otworzyć ten port w zaporze, należy go najpierw ustawić w pliku Bricsys.lic na serwerze:

1 Otwórz interfejs WWW administracji serwera licencji pod adresem **http://localhost:5054**.

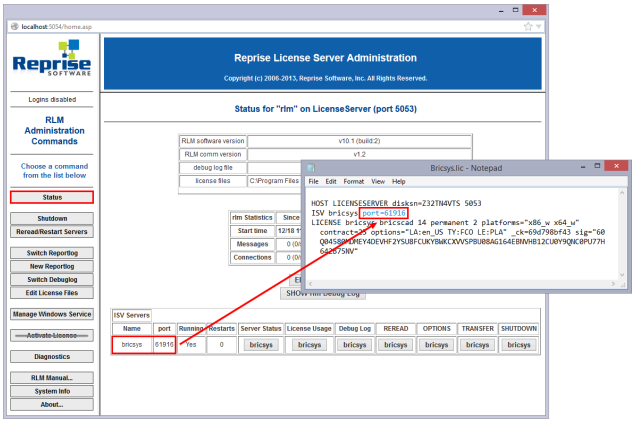
2 Kliknij przycisk **Stan** po lewej stronie. Zostanie otwarta strona **Status dla „rlm" na [LicenseServer] (port 5053)**.

3 Wyszukaj numer portu w tabeli Serwery ISV. Numer portu jest wyświetlany w wierszu bricsys w drugiej kolumnie.

4 Skopiuj ten numer portu.

5 Otwórz plik Bricsys.lic w edytorze tekstu (na przykład Notatnik, Gedit, ...). Domyślna lokalizacja pliku to C:\Program Files (x86)\Bricsys\Bricsys Network License Manager.

Możesz naprawić numer portu, modyfikując linię ISV bricsys na: ISV bricsys port=[numer portu], gdzie [numer portu] należy zastąpić numerem portu skopiowanym z tabeli Serwery ISV.



6 W zaporze otwórz oba numery 5053 i numer portu dodany do pliku Bricsys.lic.

7 Zamknij i uruchom ponownie program License Server.

Uwaga: Jeśli to nie rozwiąże problemu, dodaj program serwera licencji (rlm.exe w systemie Windows, rlm w systemie Linux/Mac) jako wyjątek do wyjątków zapory i ponownie uruchom serwer licencji.

2.3.8.3 Serwer proxy

Jeśli serwer proxy istnieje między **BricsCAD®** a serwerem licencji, ustawienia proxy należy skonfigurować w **BricsCAD®** .

Zobacz Ustawienia proxy, jak skonfigurować ustawienia proxy na stałe do korzystania z licencji sieciowej.

2.4 Zarządzanie licencjami

2.4.1 Modyfikacja licencji BricsCAD®

Użyj tej procedury, aby zaktualizować lub wymienić licencję.

1 Uruchom polecenie LICENSEMANAGER.

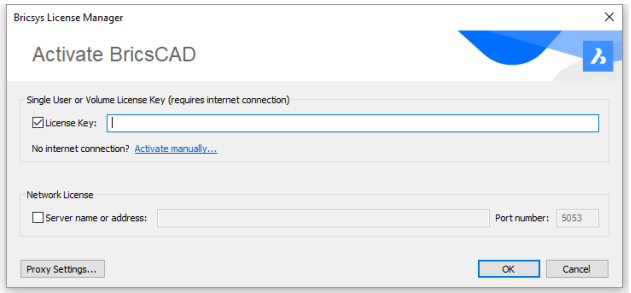
Result: Zostanie wyświetlone okno dialogowe Menedżer licencji BricsCAD.

2 Kliknij **Zarządzaj licencją**.

Result: Wyświetlone zostanie okno dialogowe **Informacje licencyjne**.

3 Kliknij przycisk **Modyfikuj**.

Result: Wyświetlane jest okno dialogowe **Menedżer licencji BricsCAD** - **Aktywuj BricsCAD**.



4 Wklej swój klucz licencyjny w polu **Klucz licencyjny**.

5 Kliknij przycisk **OK** .

Result:

- Twój klucz licencyjny jest rejestrowany przez Internet.
- Po pomyślnej aktywacji klucza licencyjnego zostanie wyświetlone okno dialogowe.

6 Przeczytaj tekst w oknie dialogowym i kliknij przycisk **Rozumiem**.

2.4.2 Dezaktywacja lub cofnięcie licencji BricsCAD®

Użyj tej procedury, aby:

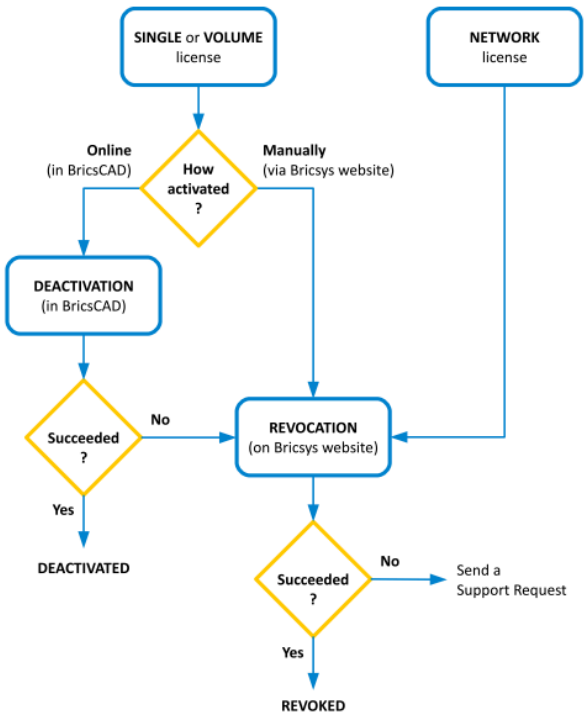
- Przenieś licencję na inny komputer.
- Trwale usuń **BricsCAD®** z komputera.
- Aktywuj ponownie **BricsCAD®** po zainstalowaniu nowego dysku twardego, zmianie partycji lub uaktualnieniu systemu operacyjnego.

Aby dezaktywować **BricsCAD®** , wymagane jest połączenie internetowe. Nie można ręcznie dezaktywować licencji za pośrednictwem innego komputera podłączonego do Internetu.

Nie możesz dezaktywować:

- Licencje pojedyncze i zbiorcze aktywowane ręcznie.
- Licencje sieciowe.
- Licencje próbne.

Tabela wyjaśnia, jak dezaktywować licencję **BricsCAD®** **BricsCAD®** nie jest możliwa, możesz ją unieważnić.



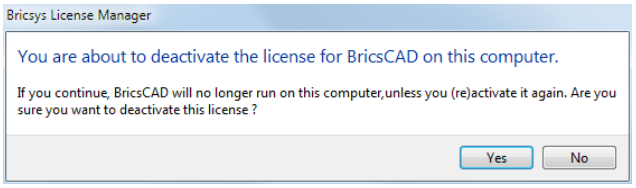
2.4.2.1 Dezaktywacja licencji

1 Uruchom polecenie LICENSEMANAGER. Kliknij **Zarządzaj licencją** opcja na **Menedżer licencji Bricsys** Okno dialogowe.

Result: Wyświetlone zostanie okno dialogowe **Informacje licencyjne**.

2 Jeśli komputer łączy się z serwerem licencji przez serwer proxy, kliknij **Ustawienia proxy**, aby skonfigurować ustawienia proxy.

3 Naciśnij **Dezaktywuj ...**



4 Kliknij przycisk **Tak** w oknie dialogowym potwierdzenia, aby dezaktywować licencję.

Result: Licencja jest dezaktywowana i można ją aktywować na innym komputerze.

2.4.2.2 Cofnięcie licencji

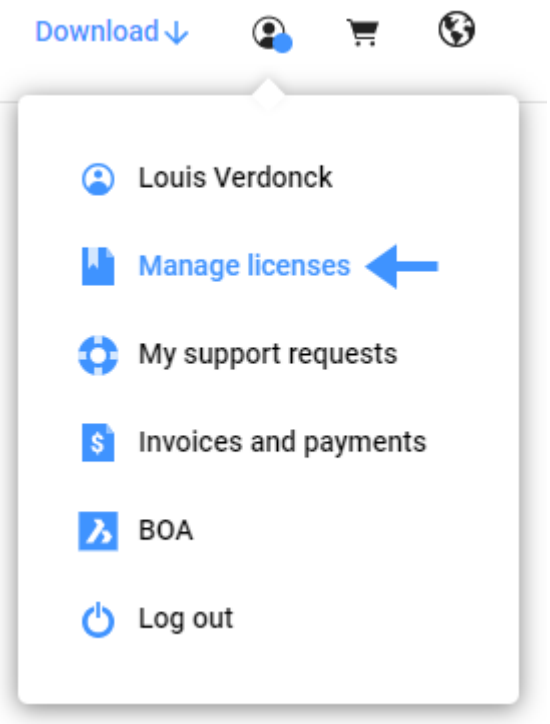
Odwołaj licencję tylko wtedy, gdy dezaktywacja nie jest możliwa.

Użyj tej procedury, aby:

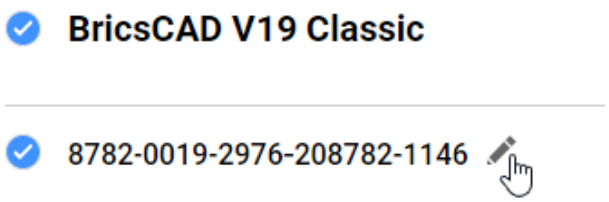
- Ręcznie dezaktywuj licencję sieciową lub licencję pojedynczą lub zbiorczą, która jest aktywowana ręcznie.
- Reaktywuj **BricsCAD®** po awarii dysku twardego.
- Aktywuj ponownie **BricsCAD®** po zainstalowaniu nowego dysku twardego, zmianie partycji lub uaktualnieniu systemu operacyjnego.
- Przenieś licencję na inny komputer.
- Trwale usuń **BricsCAD®** z komputera.

1 Zaloguj się na stronie Bricsys.
Upewnij się, że logujesz się na konto Bricsys zawierające twoje licencje.

2 Kliknij ikonę **Moje konto** ( ) i wybierz z menu opcję **Zarządzaj licencjami**.



3 Najedź kursorem na klucz licencyjny licencji, którą chcesz cofnąć, i kliknij ikonę ołówka ( ).



4 Przewiń w dół i kliknij ikonę **X**, aby cofnąć licencję.



2.4.3 Migracja licencji pojedynczej lub zbiorczej na inny komputer

Wszystkie Twoje ważne licencje możesz znaleźć na stronie Bricsys (wymagane logowanie): https://www.bricsys.com/protected/account/licenses.

1 Dezaktywuj licencję na starym komputerze.

2 Uruchom **BricsCAD®** na nowym komputerze.

3 Aktywuj licencję na nowym komputerze.

2.4.4 Migracja licencji sieciowej na inny serwer

Wykonaj następujące czynności:

1 Anuluj aktywację licencji.

2 Aktywuj licencję na nowym serwerze.

3 Zamknij menadżera licencji na starym serwerze i usuń plik licencji na starym serwerze.

Uwaga: Liczba anulacji jest ograniczona.

2.4.5 License pooling

2.4.5.1 Editing the options

You can edit the options in 2 ways:

- In the bricsys.opt file, located in the Network License Manager installation folder.
- On the web interface of the Network License Manager.
  - Browse to *http://localhost:5054* on the license server.
  - Click **Status** in the menu on the left side.
  - Click **bricsys** in the OPTIONS column for the bricsys ISV row.

Once edited, Reread/Restart the license server.

2.4.5.2 Commonly used options

Groups can be used to define a list of usernames, hostnames or IP-addresses.

The name can later be used to apply rules for that group.

		Syntax	Example
GROUP	Defines a group of usernames.	GROUP name list-of-usernames	GROUP engineers userA userB userC
HOST_GROUP	Defines a group of hostnames.	HOST_GROUP name list-of-hostnames	HOST_GROUP firstFloor machineA machineB machineC
INTERNET_GROUP	Defines a group of IP-addresses.	INTERNET_GROUP name list-of-ip-addresses	INTERNET_GROUP networkGhent 192.168.1.* 192.168.2.* The wildcard character (*) can be used in IP addresses.

		Syntax	Example
RESERVE	Reserve a number of licenses to a specific (group of) user(s).	RESERVE num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	RESERVE 20 bricscad GROUP engineers
MAX	Limit the number of licenses available to a specific (group of) user(s).	MAX num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	MAX 5 bricscad HOST_GROUP firstFloor
INCLUDE	Only allow licenses for a product to a specific (group of) user(s). Anyone not specified by the INCLUDE line is not allowed to use the product.	INCLUDE product userhost|group|host_group|internet|internet_group|project who [id=nnn]	INCLUDE bricscad GROUP bricscadusers
EXCLUDE	Disallow licenses for a product to a specific (group of) user(s).	EXCLUDE product user|host|group|host_group|internet|internet_group|project who [id=nnn]	EXCLUDE bricscad INTERNET_GROUP 1.1.*.*

More options can be found in the RLM end user manual, under the paragraph **The ISV Options File**.

2.4.5.3 Identifying a license file

Since V15, a new feature in the Reprise license manager allows you to add an '_id' field in your license file.

With this '_id', you can identify each license file as a separate license pool.

When specifying an [id=nnn] parameter for an added OPTION, that OPTION will only have effect on the corresponding license file.

Usage Example: You have 2 licenses, one with 10 seats forBricsCAD® Lite, and one with 10 seats for BricsCAD® Pro, and want to specify which users can use which level.

THE LICENSE FILES

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=""

Now the Lite license is defined by id=1, and the Pro license by id=2.

THE OPTIONS

If now, for example, you would like to only allow engineers in your firm to use the Pro version, this can be done by adding the following to the options.

GROUP engineers userA userB userC

INCLUDE bricscad GROUP engineers id=2

Symptoms

I want to adjust the way my licenses are pooled.

More information: http://www.reprisesoftware.com/RLM_Enduser.html

2.4.5.4 License pooling by RLM_PROJECT

The following applies to cases where multiple levels of network licenses are activated on the same RLM server. Consequently, the users are able to decide what license level to acquire when activating BricsCAD®.

The user can obtain a specific BricsCAD® license level, based on the RLM_PROJECT environment variable which should match the ISV server options. The license seats will be reserved on the ISV server for the specific PROJECT users by an IDentifer.

Uwaga: Please be aware that the following actions are to be executed by an IT department person.

1. RLM server actions

1 Add _id=<number> identifier to license files, just before the _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 Add RESERVE option for PROJECTs and license IDs to ISV server, for all the seats of the licenses:

RESERVE 5 bricscad PROJECT bc_pro id=1

RESERVE 5 bricscad PROJECT bc_lite id=2

2. BricsCAD® user actions

1 Create a script that SETs a temporary environment variable RLM_PROJECT on the user machine, which will allow BricsCAD to negotiate with the RLM server the PROJECT license pool.

2 Executing a specific script will allow BricsCAD to pull the desired license level

It can be *.CMD (command prompt) files:

- @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"
- or *.PS1 (powershell) files:
  $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"

Since all license seats are RESERVED in the RLM options, BricsCAD® will not be able to acquire a license when the RLM_PROJECT variable does not exist, or when it has a value which does not match the ISV server options.

Raising the available seats above 5 or lowering the RESERVE count below 5 would let some licenses be available freely.

The RLM_PROJECT also works with the other (EXCLUDE/INCLUDE/…) options so a lot of configurations are possible.

2.4.6 Raport wykorzystania licencji sieciowych

Istnieje kilka możliwości sprawdzenia wykorzystania licencji sieciowej:

- Z internetowego interfejsu użytkownika RLM
  przejdź do localhost: 5054/home.asp> Status> License Usage > bricsys

- Z pliku "bricsys.dlog", który zawiera informacje IN/OUT. Ten plik znajduje się w folderze RLM *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager*
- opcja REPORTLOG. Ta opcja jest opisana na stronach 77 i 103 w podręczniku RLM:
  http://www.reprisesoftware.com/RLM_License_Administration.pdf
  Powinien zostać dodany do pliku **bricsys.opt** w następujący sposób:
  REPORTLOG +D:\twoja_ścieżka\nazwa_raportu.log
  Następnie wykonaj ponowne wczytanie/zrestartowanie serwerów.
  Znak + oznacza, że nowe dane są dodawane do pliku, w przeciwnym razie plik zostanie nadpisany.
- Pakiet administracyjny licencji RLM
  https://reprisesoftware.com/admin/software-licensing-downloads.php
- Raport JTB Flex
  https://jtbworld.com/jtb-flexreport

2.5 Korzystanie z licencji sieciowej

2.5.1 Roaming

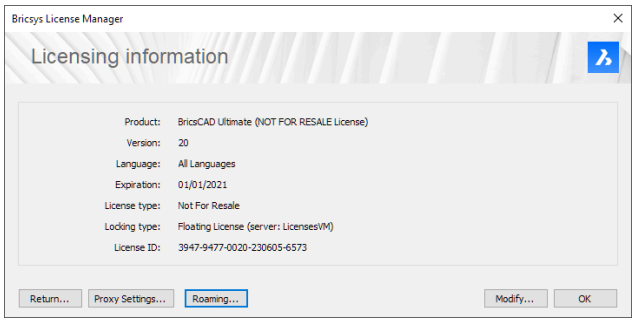
Dzięki licencji sieciowej możesz ustawić licencję pływającą na roaming do systemu, który następnie zostanie odłączony od sieci. Uzyskana licencja może być używana przez określoną liczbę dni, gdy została ustawiona na roaming. Po tym okresie licencja jest automatycznie ponownie wprowadzana. W razie potrzeby możesz wcześniej zwrócić licencję wypożyczoną do puli licencji.

Plik(i) licencji na serwerze licencji sieciowej zawiera(ją) parametr *max_roam_count*Jest to domyślnie ustawione na całkowitą liczbę stanowisk w licencji sieciowej.

2.5.1.1 Roaming licencji sieciowej na komputerze Klienta

Licencja sieciowa może być mobilna, aby używać **BricsCAD**® bez połączenia z serwerem licencji (praca poza biurem, w domu, ...).

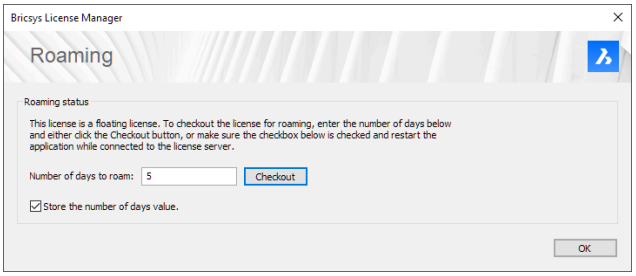
Kiedy licencja sieciowa jest używana na komputerze klienta, przycisk **Roaming ...** jest wyświetlany w oknie dialogowym Informacje licencyjne .



1 Jeśli komputer jest połączony z serwerem licencji przez serwer proxy, kliknij przycisk **Ustawienia proxy**, aby skonfigurować ustawienia proxy dla licencji sieciowej.

2 Kliknij przycisk **Roaming ...**

Result: Otworzy się okno dialogowe **Roaming**



3 Wprowadź liczbę dni, przez które chcesz sprawdzić licencję na roaming i kliknij przycisk **Wyloguj .**

4 (Opcjonalnie) Zaznacz to pole wyboru, jeśli chcesz automatycznie odświeżać okres roamingu za każdym razem, gdy komputer jest połączony z serwerem.

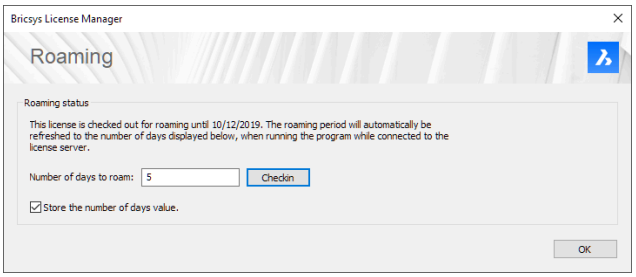
5 Uruchom ponownie **BricsCAD®** aby zastosować zmiany.

2.5.1.2 Zwracanie wypożyczonej licencji sieciowej na serwer

Jeśli masz połączenie z serwerem licencji, możesz zwrócić licencję mobilną na serwer.

1 Kliknij przycisk **Roaming ...** w oknie dialogowym Informacje licencyjne .

Result: Otworzy się okno dialogowe **Roaming**



2 Kliknij przycisk **Checkin** .

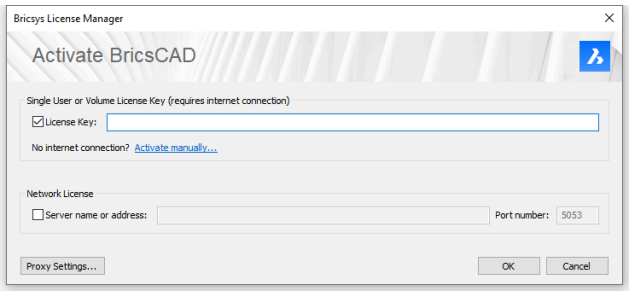
3 Uruchom ponownie **BricsCAD®** aby zastosować zmiany.

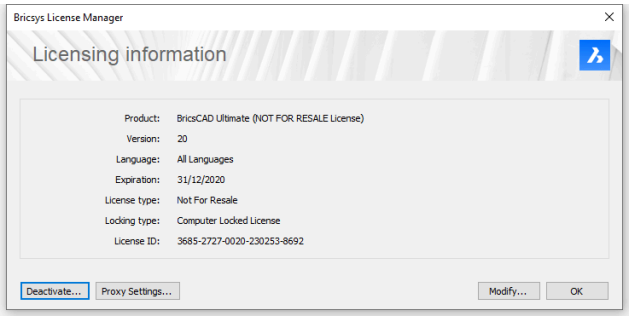
2.5.2 Ustawienia proxy na komputerze klienckim

Zanim rozpoczniesz: Niektóre komputery są połączone z siecią lub Internetem za pośrednictwem serwera proxy. Dlatego ustawienia proxy należy skonfigurować przed aktywacją lub dezaktywacją licencji.

2.5.2.1 Konfiguracja ustawień proxy

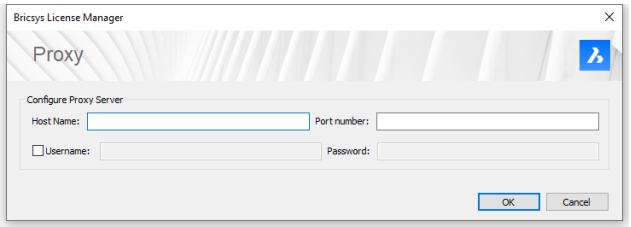
1 Kliknij przycisk **Ustawienia proxy...** w **Aktywuj BricsCAD** lub w oknie dialogowym **Informacje licencyjne**.





Result: ten **Pełnomocnik** otworzy się okno dialogowe.

2 Wypełnij wymagane pola w oknie dialogowym **Proxy**.



3 Kliknij przycisk **OK**, aby kontynuować.

Result:

Uwaga: Konfiguracja ustawień proxy pozostaje aktywna tylko dla bieżącej sesji **BricsCAD®** Jeśli będziesz musiał ponownie aktywować lub dezaktywować licencję w przyszłej sesji, będziesz musiał ponownie skonfigurować ustawienia proxy.

2.5.2.2 Konfiguracja ustawień proxy na stałe do użytku z licencją sieciową

Zanim rozpoczniesz: Jeśli używasz licencji sieciowej, **BricsCAD®** musi łączyć się z serwerem licencji za każdym razem, gdy sesja jest otwierana. Dlatego ustawienia proxy należy skonfigurować na stałe.

Postępuj w następujący sposób na komputerze, na którym działa **BricsCAD®**

1 Utwórz zmienną **HTTP_PROXY:** HTTP_PROXY=[Serwer Licencji]:[Numer Portu]

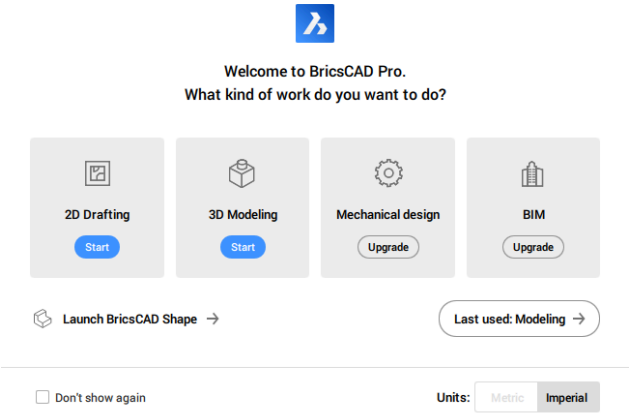
2 (Opcjonalnie) Jeśli serwer proxy obsługuje uwierzytelnianie podstawowe, utwórz zmienną **HTTP_PROXY_CREDENTIALS:** HTTP_PROXY_CREDENTIALS=[Nazwa Użytkownika]:[Hasło]

3. Pierwsze kroki zBricsCAD®

3.1 Uruchamianie BricsCAD®

3.1.1 Launcher BricsCAD

Jeśli zmienna systemowa GETSTARTED jest włączona, po uruchomieniu BricsCAD® pojawi się okno dialogowe BricsCAD Launcher. Tutaj możesz wybrać obszar roboczy dla bieżącej sesji. Dostępne obszary robocze zależą od poziomu licencji.



- **Rysowanie 2D** : uruchamia przestrzeń roboczą Rysowanie.BricsCAD® Lite wymagana jest licencja lub wyższa.
- **Modelowanie 3D** : uruchamia przestrzeń roboczą Modelowanie.BricsCAD® Pro wymagana jest licencja lub wyższa.
- **Roboczy Mechanical** : uruchamia obszar roboczy Mechanical. lub wymagana jest BricsCAD® Ultimate
- **BIM** : uruchamia przestrzeń roboczą BIM.BricsCAD® BIM lub wymagana jest BricsCAD® Ultimate
- **Uruchom BricsCAD Shape** : uruchamia BricsCAD® Shape . Licencja nie jest wymagana.
- **Ostatnio używane**: uruchamia ostatnio używany obszar roboczy.
- **Nie pokazuj ponownie** : ustawia zmienną systemową GETSTARTED na 0. Aby zresetować wyświetlanie okna dialogowego BricsCAD Launcher, wpisz **Zaczynaj** w linii poleceń i wybierz **1 dla ON** .
- **Jednostki** : ustawia jednostki dla domyślnego rysunku szablonu. Definiuje zmienną systemową BASEFILE.

Uwaga: Darmowe narzędzie do modelowania bryłowego, BricsCAD® Shape , jest wbudowane w pojedynczy instalator BricsCAD® Gdy BricsCAD® Ultimate wygasa trial 30 dni, to powraca do wieczystego BricsCAD® Shape licencji, które mogą być konwertowane do pełnej licencji komercyjnej poprzez zakup klucza licencyjnego. Nie musisz najpierw BricsCAD®

3.2 Odkrywanie interfejsu

3.2.1 Karta Start

Zakładka **Początek** znajduje się w górnej części obszaru rysunku, przed zakładkami dokumentu.

W zakładce **Start** możesz:

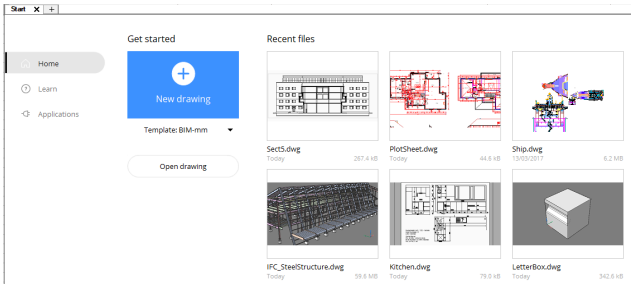
- **Otwórz ostatni rysunek**

  Kliknij miniaturę ostatniego pliku.

  Liczba dostępnych ostatnio plików jest kontrolowana przez wartość zmiennej systemowej RECENTFILES.
- **Utwórz nowy rysunek**

  Kliknij kafelek **Nowy rysunek**, aby utworzyć nowy rysunek przy użyciu bieżącego szablonu rysunku. Opcjonalnie otwórz listę **Szablon**, aby wybrać inny szablon rysunku.
- **Otwórz rysunek**

  Wyświetla okno dialogowe **Otwórz rysunek**.



Zakładka **Początek** jest domyślnie aktywna. Jeśli przypadkowo zamknąłeś zakładkę, możesz ją ponownie otworzyć za pomocą skrótu klawiaturowego **Ctrl + Home** lub za pomocą polecenia IDŹNASTART.

Uwaga: Korzystając ze skrótu klawiaturowego **Ctrl + Home**, pamiętaj o wyłączeniu Num lock.

Uwaga: Aby wyświetlić kartę **Start**, zmienna systemowa SHOWDOCTABS musi być **włączona**.

3.2.2 Zakładki dokumentu

Dla każdego aktualnie otwartego rysunku w górnej części obszaru rysunku znajduje się zakładka dokumentu.

Zakładki dokumentów umożliwiają:

- Łatwo przełączaj się między wszystkimi otwartymi rysunkami.
- Utwórz nowy rysunek: kliknij przycisk **+** po prawej stronie zakładek.
- Zamknij rysunek: kliknij **X** na karcie dokumentu.

3.2.3 Wybór obszaru roboczego

Domyślne obszary robocze to:

- **Kreślenie** : używa wstążki.
- **Szkicowanie (paski narzędzi)** : używa menu i pasków narzędzi.

- **Modelowanie**: wymaga **BricsCAD® Pro** lub wyższego poziomu licencji; używa wstążki.
- **Modelowanie (paski narzędzi)**: wymaga **BricsCAD® Pro** lub wyższego poziomu licencji; używa menu i pasków narzędzi.
- **Mechaniczny**: wymaga **BricsCAD® Mechanical** lub **BricsCAD® Ultimate** poziomu licencji.
- **BIM**: wymaga **BricsCAD® BIM** lub **BricsCAD® Ultimate** poziomu licencji.

Bieżący obszar roboczy konfiguruje ogólny układ okna aplikacji **BricsCAD®**

Obszar roboczy definiuje zawartość:

- Menu
- Paski Narzędzi
- Panele
- Wstążka
- Podstawy Quad

Możesz ustawić bieżący obszar roboczy w:

- Launcher BricsCAD
- Pole Bieżący obszar roboczy na pasku stanu.
- Pasek narzędzi Obszary robocze.
- Polecenie WORKSPACE.

3.2.4 Obszar rysowania

Na początku sesji rysowania początkowy obszar roboczy nosi nazwę **obszar modelu** . Przestrzeń modelu to obszar, w którym tworzy się dwuwymiarowe i trójwymiarowe elementy w oparciu o Globalny układ współrzędnych (GUW) lub lokalny układ współrzędnych (LUW).**BricsCAD®** zapewnia dodatkowy obszar roboczy zwany **przestrzenią papieru** . Każdy układ reprezentuje arkusz papieru. W jednym rysunku można utworzyć do 255 układów obszaru papieru.

Użyj karty **Model**, aby przeglądać i pracować w obszarze modelu. Kliknij jedną z kart **Układ** u dołu obszaru rysunku, aby przełączyć się do obszaru papieru.

3.2.4.1 Obszar modelu

Obszar modelu składa się z pojedynczego widoku wypełniającego ekran. W razie potrzeby można utworzyć dodatkowe widoki zwane rzutniami, które mogą pokazywać różne widoki rysunku lub modelu 3D. Możesz zarządzać rzutniami rysunkowymi za pomocą polecenia RZUTNIE.

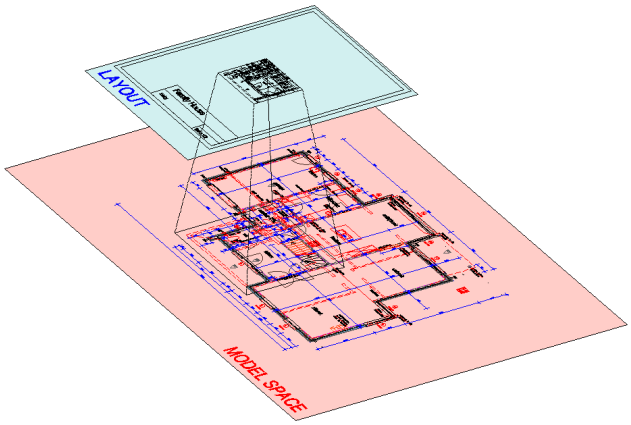
Rzutnie są wyświetlane jako kafelki. Możesz pracować jednocześnie tylko w jednej z tych rzutni, ale wszystkie rzutnie są aktualizowane jednocześnie. Kliknij rzutnię, aby ustawić ją jako bieżącą rzutnię.

3.2.4.2 Obszar Papieru

Obszar papieru reprezentuje układ papieru rysunku. W tym obszarze roboczym możesz tworzyć i rozmieszczać różne widoki rysunku, podobnie jak układasz rysunki na kartce papieru. W obszarze papieru

można również dodawać opisy indeksowe, adnotacje, ramki, tabelki rysunkowe i inne elementy związane z drukowaniem, których nie chcesz wyświetlać w obszarze modelu.

Każdy rysunek ma co najmniej jeden **układ**, w którym można mieć jedną lub więcej rzutni układu. Rzutnię układu można traktować jako okno widoku na rysunku w obszarze modelu:



Rzutnie układu można umieszczać w dowolnym miejscu układu obszaru papieru. Każda rzutnia układu ma własną skalę i widoczność warstw. Możesz drukować wszystkie rzutnie w tym samym układzie jednocześnie.

Możesz zarządzać rzutniami układu za pomocą polecenia WIDOK.

Przestrzeń papieru zapewnia wiele korzyści podczas drukowania rysunku:

- Możesz utworzyć wiele rzutni, aby wydrukować wiele widoków. Każdy z tych widoków może wyświetlać część rysunku w tej samej lub w różnych skalach.
- Możesz utworzyć wiele układów, aby wydrukować ten sam rysunek z różnymi ustawieniami drukowania, takimi jak szerokość pisaka, pliki konfiguracyjne drukarki, ustawienia grubości linii, skala rysunku ...
- Możesz dodać elementy związane z drukowaniem, które nie są istotne dla samego modelu, takie jak opisy odnośników, adnotacje, tabelki rysunkowe ...

3.2.5 Wiersz poleceń

Wiersz poleceń to opcjonalny dokowalny panel, który domyślnie znajduje się na dole obszaru rysunku.

Wiersz poleceń to szybka i wydajna metoda uruchamiania poleceń oraz ustawiania zmiennych systemowych i preferencji użytkownika.

BricsCAD® prosi o wprowadzenie danych przez użytkownika w linii poleceń i zwraca tutaj wyniki poleceń, takich jak ODLEGŁOŚĆ , OBSZAR lub LISTA .

Gdy wiersz poleceń jest zamknięty, wpis z klawiatury i monity poleceń są wyświetlane na pasku stanu .

3.2.6 Menu

Pasek menu jest domyślnie ukryty we wszystkich obszarach roboczych.

Zawartość menu jest zdefiniowana w bieżącym obszarze roboczym.

3.2.6.1 Sterowanie wyświetlaniem paska menu

1 Przesuń kursor na pasek narzędzi lub panel wstążki.

2 Kliknij prawym przyciskiem myszy i wybierz **Pasek menu** z menu kontekstowego.

3.2.7 Paski Narzędzi

Paski narzędzi są używane domyślnie w obszarach roboczych Rysowanie (paski narzędzi) i Modelowanie (paski narzędzi), ale można ich używać w dowolnym obszarze roboczym.

- Możesz wybrać małe (16 x 16), duże (32 x 32) lub bardzo duże (64 x 64) ikony na paskach narzędzi.
- W przypadku niektórych przycisków narzędzi mała czarna strzałka w prawym dolnym rogu ikony wskazuje, że dostępne jest menu rozwijane. W menu wysuwanym znajduje się grupa powiązanych narzędzi. Naciśnij i przytrzymaj lewy przycisk myszy, aby rozwinąć menu wysuwane i wybierz jedno z narzędzi wysuwanych. Ostatnio wybrane narzędzie pozostaje widoczne na zwiniętym pasku narzędzi.
- W każdej chwili możesz otwierać lub zamykać paski narzędzi.

3.2.8 Wstążka

Wstążka to zestaw pasków narzędzi z zakładkami, wypełnionych przyciskami narzędzi i innymi graficznymi elementami sterującymi, pogrupowanymi według funkcji.

Zawartość wstążki zależy od bieżącego obszaru roboczego.

Obszary robocze kreślenia, modelowania, mechaniki i BIM domyślnie korzystają ze wstążki.

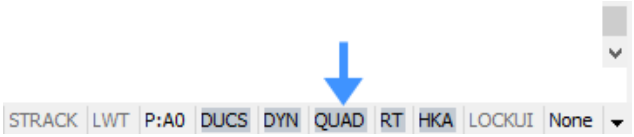
Użyj polecenia WSTĄŻKA i WSTĄŻKA ZAMKNIJ, aby otworzyć lub zamknąć wstążkę.

3.2.9 Podstawy Quad

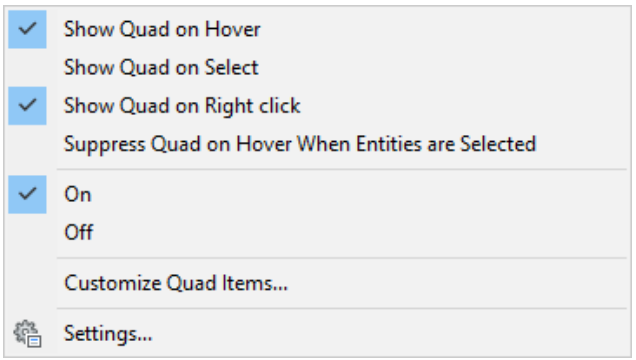
Quad to kompletny system poleceń umieszczony bezpośrednio przy wskaźniku myszy.

Podczas przesuwania kursora po ekranie lub gdy istnieje zbiór wskazań, Quad wykrywa typ elementu i przedstawia polecenia, które działają na tym elemencie lub na wyborze elementów. Gdy żaden element nie zostanie wykryty ani wybrany, możesz użyć Quad do tworzenia elementów, wstawiania bloków, dołączania podkładań lub otwierania okien dialogowych **Ustawienia** i **Eksplorator rysunków**.

- Aby przełączać wyświetlanie **Menu Quad**, kliknij pole Quad na pasku stanu u dołu **BricsCAD®** okna aplikacji.



- Kliknij prawym przyciskiem pole **Quad**, aby zdefiniować ustawienia Quad w menu kontekstowym.

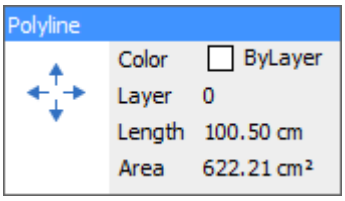


3.2.9.1 Podstawowa procedura edycji elementów za pomocą Quad

1 Przesuń kursor nad element. Element zostanie podświetlony, a Quad wyświetla ostatnio używane polecenie dla tego typu elementu.

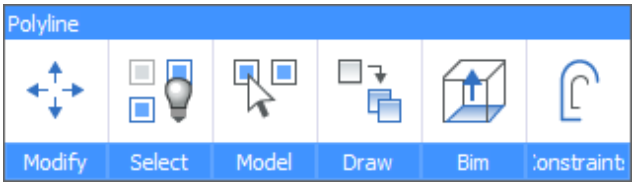


Jeśli zmienna systemowa ROLLOVERTIPS = ON, właściwości obiektów pod kursorem są wyświetlane w Quad.

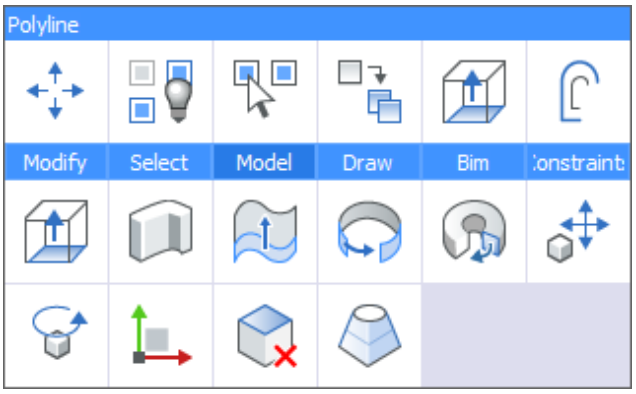


2 Przesuń kursor nad Quad, aby wyświetlić ostatnio używane polecenia z tym typem elementu.

Na dole zostanie wyświetlonych kilka kart kategorii poleceń.



3 Przesuń kursor nad zakładkami, aby rozwinąć grupę poleceń pod kursorem.



4 Kliknij ikonę, aby uruchomić polecenie.

3.2.9.2 Korzystanie z kwadratu bez wyboru

Upewnij się, że opcja **Wyświetl Quad po kliknięciu prawym przyciskiem myszy** jest ustawiona.

1 Wykonaj jedną z następujących czynności:

- Przesuń kursor do pustego obszaru w oknie rysunku i kliknij prawym przyciskiem myszy.
- Gdy kursor jest na elemencie, przytrzymaj dłużej prawy przycisk myszy.

Zostanie wyświetlony kwadracik **bez wyboru** , zawierający ikony ostatnio używanych poleceń.



2 Wykonaj jedną z następujących czynności:

- Kliknij jedno z ostatnio używanych poleceń.
- Przesuń kursor na kartę polecenia i wybierz polecenie, gdy się rozwinie.

3 Wykonaj polecenie.

3.2.10 Menu prawego przycisku myszy

Menu kontekstowe lub menu prawego przycisku myszy wyświetlają listę poleceń obok kursora po kliknięciu prawym przyciskiem myszy.

Gdy Quad jest aktywny, przytrzymaj prawy przycisk myszy nieco dłużej (= długie kliknięcie prawym przyciskiem myszy).

Zawartość menu kontekstowego zależy od bieżącego elementu lub zbioru wskazań.

Gdy Quad nie jest aktywny i nic nie jest zaznaczone, kliknięcie prawym przyciskiem myszy powtarza ostatnie polecenie. Przytrzymaj prawy przycisk myszy trochę dłużej, aby wyświetlić menu kontekstowe.

3.2.11 Pasek stanu

Pasek stanu znajduje się na dole obszaru rysunku.

- Wyświetla podpowiedzi po najechaniu kursorem na przycisk narzędzia na pasku stanu.

- Umożliwia wprowadzanie nazw poleceń i podpowiada opcje poleceń, gdy okno poleceń jest zamknięte.
- Wyświetla aktualny stan ustawień elementów: warstwa, kolor, styl linii, ... i umożliwia edycję tych ustawień.

  Uwaga: Są one domyślnie wyłączone w dowolnym obszarze roboczym.
- Wyświetla aktualny styl przyciemnienia i styl tekstu oraz umożliwia ich ustawienie.
- Wyświetla aktualny stan pomocy rysunkowych, takich jak przyciąganie elementów i umożliwia ich włączanie / wyłączanie.
- ...

Kliknij małą czarną strzałkę w dół po prawej stronie paska stanu, aby zarządzać wyświetlaniem pól.

3.2.12 Panele

Panele to dokowalne okna dialogowe.

Dostępne panele to:

- Panel Edytora Animacji
- Załączniki
- BIM BCF
- BIM Kompozycje
- BIM Profile
- Przeglądarka projektów BIM
- Menedżer BOM
- Bricsys 24/7
- Pasek poleceń
- Przeglądarka zawartości
- Detale
- Dostosowywanie rysunków
- Warstwy
- Biblioteka
- Przeglądarka Mechaniczna
- Parametry i ograniczenia
- Właściwości
- Materiały renderowane
- Raport
- Wstążka
- Zestawy Arkuszy
- Struktura
- Podpowiedzi
- Paleta Narzędzi

3.2.13 Tryb czystego ekranu

W trybie czystego ekranu można zmaksymalizować obszar rysunku, odrzucając elementy interfejsu użytkownika, takie jak pasek menu, wstążka, paski narzędzi, panele, wiersz poleceń, pasek stanu i karty dokumentów.

Nadal możesz używać Quad do uruchamiania poleceń.

Naciśnij i przytrzymaj klawisz Ctrl i naciśnij klawisz 0 (zero), aby włączyć / wyłączyć tryb czyszczenia ekranu.

Uwaga: Nie używaj klawiatury numerycznej.

Uwaga: Na klawiaturach AZERTY: nie naciskaj klawisza Shift podczas naciskania klawisza 0.

3.2.14 Widget PatrzOd (orientacja widoku)

Widżet Patrz od umożliwia sterowanie widokami rysunku. Można wybrać kilka gotowych widoków, takich jak widoki ortogonalne i widoki izometryczne.



Domyślnie widget **Patrz od** wyświetla się w prawym górnym rogu ekranu graficznego.

Zachowanie widgetu **Patrz od** może być kontrolowane przez szereg zmiennych systemowych i preferencji użytkownika.

3.3 Praca w BricsCAD®

3.3.1 Tworzenie rysunku

Istnieje kilka sposobów tworzenia nowych rysunków w BricsCAD:

- w **Początek** patka.
- Za pomocą polecenia SNOWY.
- Za pomocą polecenia NOWY.
- Korzystanie z polecenia NEWWIZ.
- Dwukrotne kliknięcie pliku *.dwt w oknie Eksploratora Windows.

Uwaga: Nowy rysunek jest zawsze tworzony jako kopia rysunku szablonu. Domyślny szablon jest zapisywany przez zmienną systemową BASEFILE.

3.3.2 Rysunki szablonów

Rysunek szablonu to plik rysunku z rozszerzeniem DWT.

Służy do tworzenia nowego rysunku, który jest kopią rysunku szablonu. Każdy rysunek z rozszerzeniem DWT jest traktowany jako rysunek szablonu. **BricsCAD®** znajdują się w folderach szablonów. Użyj polecenia TEMPLATEFOLDER, aby otworzyć folder. Jeśli zapiszesz własne rysunki szablonów w folderze szablonów, będą one dostępne na karcie Start do tworzenia nowych rysunków. Korzystanie z odpowiednio przygotowanych szablonów jest niezbędne do produktywnego korzystania z **BricsCAD®** .

Rysunek szablonu nie zawiera elementów w obszarze modelu, ale zawiera ustawienia zgodne z określonymi wymaganiami rysunku i projektu:

- Jednostki: imperialne lub metryczne oraz wyświetlanie jednostek.
- Warstwy.
- Style wymiarów i linii odniesienia.
- Style tekstu.
- Style tabel.
- Wiele układów, każdy z tabelką rysunkową, rozmiarem papieru, drukarką i stylem wydruku.
- Standardowe bloki na notatki, tabele, symbole i inne elementy.

3.3.3 Ustawienia tworzenia jednostek

Nowe elementy są tworzone na bieżącej warstwie przy użyciu aktualnych ustawień tworzenia elementów.

- Kolor
- Rodzaj linii
- Skala Rodzaju Linii
- SzerokośćLinii
- Przezroczystość

Ustawienia tworzenia elementów można zdefiniować w sekcji **Ogólne** panelu Właściwości.

Można zdefiniować wszystkie ustawienia tworzenia jednostek **JakWarstwa**. Oznacza to, że używane jest odpowiednie ustawienie bieżącej warstwy.

Właściwości warstwy można zdefiniować w sekcji Warstwa w oknie dialogowym Eksplorator rysunków.

3.3.4 Jak rysować dokładnie

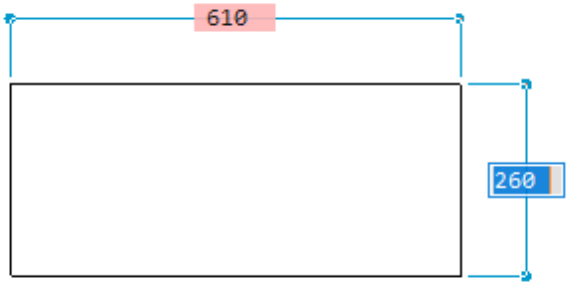
Nowe elementy są tworzone w płaszczyźnie XY bieżącego układu współrzędnych. Domyślnie jest to Globalny Układ Współrzędnych (GUW). Wszystkie wymiary w pliku DWG są wyrażone w jednostkach rysunku (DU). Zmienna systemowa INSUNITS (jednostki wstawiania) ustawia wartość 1 DU.

W przypadku nowych rysunków zmienna systemowa INSUNITS jest definiowana w szablonie rysunku wybranym do utworzenia rysunku. **BricsCAD**® udostępnia serię szablonów dla jednostek dziesiętnych i imperialnych.

Aby rysować dokładnie, **BricsCAD**® zapewnia kilka pomocy rysunkowych:

- Przyciąganie elementów 2D: możesz używać punktów charakterystycznych na istniejących elementach 2D, takich jak punkt końcowy, punkt środkowy lub środek i przecięcie. Możesz także rysować prostopadle, równolegle lub stycznie do elementu.
- Przyciąganie elementów 3D: możesz używać punktów charakterystycznych na elementach 3D, takich jak bryły.

- Wymiary dynamiczne. Kiedy dynamiczne wymiary są **włączone**, można wprowadzać wartości odległości i kątów w dynamicznych polach wymiarów. Gdy wyświetlanych jest wiele dynamicznych pól wymiarowych, na przykład podczas rysowania prostokąta, naciśnij klawisz Tab, aby przejść do następnego pola. Po wpisaniu wartości i naciśnięciu klawisza Tab pole zmienia kolor na czerwony, co oznacza, że wartość jest zblokowana.



- Śledzenie biegunowe wymusza kierunek rysowania pod określonymi kątami w odniesieniu do osi (X, Y, Z) bieżącego układu współrzędnych.
- Śledzenie punktów charakterystycznych obiektu łączy w sobie punkty charakterystyczne i śledzenie biegunowe.
- Wprowadzanie współrzędnych umożliwia wprowadzenie współrzędnych XYZ w celu określenia punktów.
- Przyciąganie i siatka. Siatka to zestaw równomiernie rozmieszczonych linii lub kropek, które służą jako wizualne odniesienie odległości. Kursor jest przyciągany do punktów siatki, co umożliwia szybkie rysowanie w stałych odległościach między punktami.
- Ograniczenia rysunków pozwalają ograniczyć obszar rysunku, co zapobiega tworzeniu elementów poza granicami rysunku.
- Tryb Orto wymusza przesuwanie kursora równolegle do osi X lub Y bieżącego układu współrzędnych.
- Układ współrzędnych użytkownika Można zdefiniować nazwane układy współrzędnych użytkownika (LUW) znajdujące się w dowolnym miejscu w przestrzeni trójwymiarowej. Nazwany LUW można przywrócić w dowolnym momencie.
- Bezpośrednie wprowadzanie odległości umożliwia wprowadzanie długości, promieni, średnic i wektorów ruchu.
- Najbliższa odległość umożliwia dynamiczne wyświetlanie i modyfikowanie odległości między dwoma wybranymi elementami lub podelementami.
- Manipulator to wszechstronne narzędzie do chwytania, które umożliwia szybkie przesuwanie , obracanie , skalowanie i tworzenie lustrzanego odbicia wybranych elementów.

3.3.5 Wykonywanie poleceń

W **BricsCAD**® możesz uruchamiać polecenia w następujący sposób:

- Wpisz polecenie w wierszu poleceń.

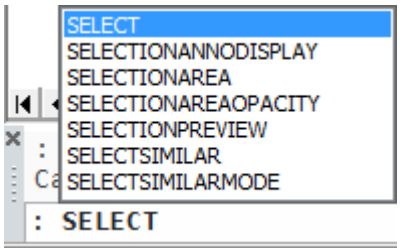
- Wybierz polecenie w menu.
- Kliknij przycisk narzędzia polecenia na pasku narzędzi.
- Kliknij ikonę polecenia na Wstążce.
- Wybierz polecenie w Quad.

3.3.5.1 Wpisywanie polecenia w wierszu poleceń

Jeśli znasz nazwę polecenia, możesz wpisać ją w wierszu poleceń. Gdy wiersz poleceń jest zamknięty, wpis z klawiatury i monity poleceń są wyświetlane na pasku stanu .

Jeśli flaga **Włącz** w zmiennej systemowej AUTOCOMPLETEMODE jest ustawiona, nazwy poleceń są automatycznie uzupełniane podczas wpisywania.

Wszystkie polecenia i / lub zmienne systemowe i / lub preferencje użytkownika, które zawierają bieżący ciąg znaków, są wyświetlane na liście sugestii.



Wykonaj jedną z następujących czynności:

- Naciśnij klawisz Enter, jeśli zostanie wyświetlone aktualnie podświetlone polecenie.
- Kliknij polecenie na liście sugestii.

Niektóre polecenia można uruchamiać za pomocą aliasów poleceń . Na przykład możesz uruchomić polecenie LINIA , wpisując L w wierszu polecenia.

3.3.5.2 Wybór opcji polecenia

Wiele **BricsCAD**® udostępnia wiele opcji. Opcje te pojawiają się w wierszu poleceń lub na pasku stanu (gdy wiersz poleceń jest zamknięty), a także w menu kontekstowym .

Wykonaj jedną z następujących czynności:

- Naciśnij klawisz Enter, aby wybrać opcję domyślną. Opcja domyślna znajduje się na końcu wiersza polecenia, między nawiasami strzałek (<>).
- Kliknij opcję w wierszu poleceń.
- Kliknij opcję w menu kontekstowym .
- Wpisz wielką literę opcji, naciśnij klawisz Enter lub kliknij prawym przyciskiem myszy.

3.3.6 Kontrola widoku

W **BricsCAD**® możesz sterować wyświetlaniem swojego rysunku na różne sposoby:

- Za pomocą myszy i klawiatury
- Używając poleceń Widoku

- Korzystając z widżetu Patrz od

Najczęściej używane manipulacje widokiem to:

Działanie	Mysz	Polecenie	PatrzOd
Zakresy widoków	Kliknij dwukrotnie kółko przewijania	**Powiększ do Zakres**	Kliknięcie strefy wyzwalania powoduje obrócenie widoku i powiększenie do zakresu rysunku.
Zoom	Kółko przewijania	**PrzybliżOddal**	
Przesuń	Przytrzymaj kółko przewijania ***Uwaga:*** Gdy zmienna systemowa MBUT-TONPAN = ON	Przesuń	
Przywróć widok planu (widok z góry)		**Plan** > **Aktualny LUW**	Kliknij środkową strefę wyzwalania

3.3.7 Wskazywanie elementów

Polecenia manipulacyjne, takie jak PRZESUŃ , KOPIUJ i LUSTRO, monitują o wybranie jednego lub więcej elementów. Jeśli zmienna PICKFIRST jest włączona, możesz również utworzyć zbiór wskazań przed uruchomieniem polecenia.

BricsCAD® udostępnia kilka metod selekcji. Możesz łączyć metody selekcji, aby utworzyć zbiór wskazań.

3.3.7.1 Wybieranie jednostek jeden po drugim

1 Przesuń kursor nad element.

Result: Jednostka wyróżnia się.

Pojawia się Quad i wyświetla ostatnio używane polecenie oraz wybrane właściwości elementu.

2 Wykonaj jedną z następujących czynności:
- Kliknij, aby wybrać jednostkę.
- Kilkakrotnie naciśnij klawisz Tab, aby podświetlić zasłonięte elementy i kliknij, gdy żądany element zostanie podświetlony.

3 Powtórz poprzednie kroki, aby wybrać więcej elementów lub uruchomić polecenie .

3.3.7.2 Wybieranie wielu podmiotów

BricsCAD® używa metod selekcji oknem, w celu wyboru wielu elementów

1 Kliknij, aby zdefiniować narożnik prostokąta wyboru.

2 Wykonaj jedną z następujących czynności:
   - Przesuń kursor w prawo. Zostanie wyświetlony niebieski prostokąt. Okrąg **wewnątrz ikony prostokąta** wskazuje, że zostaną wybrane wszystkie elementy, które znajdują się w całości w obrębie prostokąta wyboru.
   - Przesuń kursor w lewo. Zostanie wyświetlony zielony prostokąt. Okrąg **nakładający się na ikonę prostokąta** oznacza, że zostaną wybrane wszystkie elementy, które leżą w obrębie prostokąta wyboru lub nakładają się na niego.
3 Kliknij, aby zdefiniować przeciwległy narożnik prostokąta wyboru.

3.3.7.3 Usuwanie elementów ze zbioru wskazań

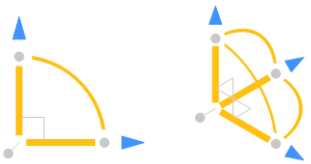
Przytrzymaj klawisz Shift podczas wykonywania metody wyboru, aby usunąć elementy z bieżącego zbioru wskazań.

Naciśnij klawisz Esc, aby wyczyścić bieżący zbiór wskazań.

3.3.8 Korzystanie z manipulatora

Manipulator to wszechstronne narzędzie do chwytania, które umożliwia szybkie przesuwanie , obracanie , skalowanie i tworzenie lustrzanego odbicia wybranych elementów.

Jeśli przytrzymasz klawisz Ctrl podczas tych czynności, wybrane elementy zostaną skopiowane.

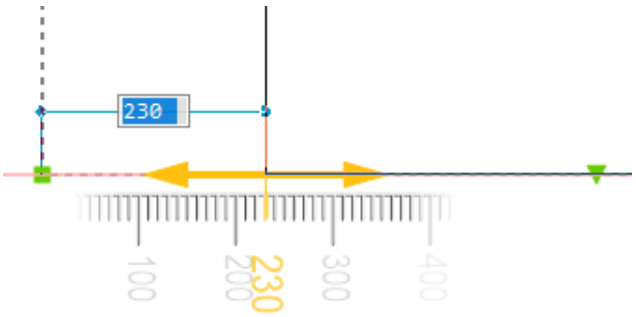


Domyślnie Manipulator wyświetla się po wybraniu elementu długim kliknięciem (przytrzymując nieco dłużej lewy przycisk myszy).

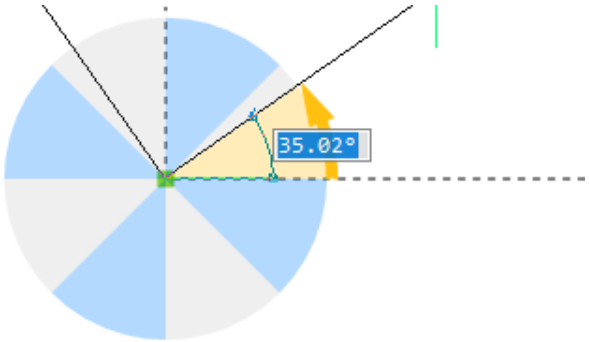
Punkt zakotwiczenia manipulatora znajduje się w najbliższym punkcie uchwytu względem punktu wyboru.

Umieść kursor na jednej ze stref wyzwalania, aby rozpocząć akcję:

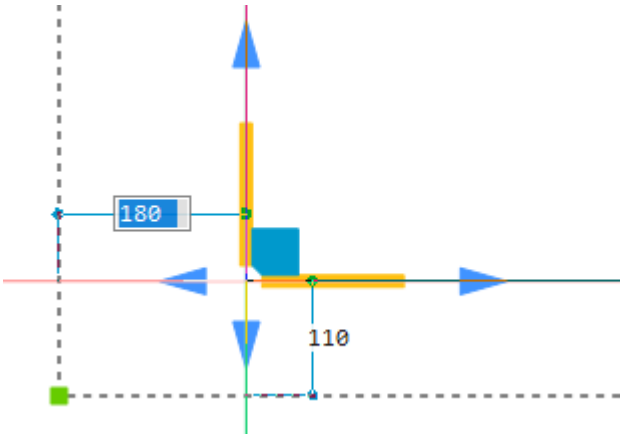
- Osie, aby poruszać się wzdłuż osi:



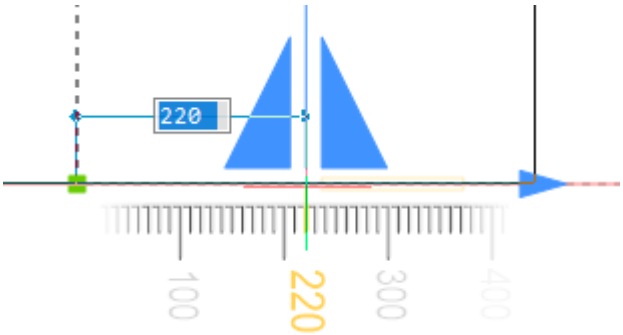
- Łuki do obracania:



- Próbki płaszczyzny (= kwadraty w punkcie kotwiczenia), aby swobodnie poruszać się po wybranej płaszczyźnie (XY, XZ lub YZ):



- Groty strzałek do lustra:



Uwaga: Wielkość kroku linijki zależy od współczynnika powiększenia. Przybliż, aby zmniejszyć rozmiar kroku, oddal, aby zwiększyć.

3.3.9 Praca z Panelem struktury

Panel Struktura wyświetla uporządkowany widok drzewa zawartości rysunku.

BricsCAD® zapewnia 3 definicje drzewa konfiguracji (CST):

- **Domyślnie** wyświetla strukturę opartą na warstwach.
- **BIM** wyświetla strukturę opartą na jednostkach BIM.

- **Mechanical** wyświetla strukturę opartą na elementach.

Kliknij menu hamburgera (☰), aby wybrać definicję.

W panelu Struktura możesz:

- Kliknij ikonę rozwijania (**+**), aby rozwinąć węzeł.
- Kliknij ikonę zwijania (**-**), aby zwinąć węzeł.
- Wybierz węzeł, kliknij prawym przyciskiem myszy i wybierz opcję **Rozwiń wszystko**, aby rozwinąć wszystkie węzły podrzędne wybranego węzła.

  Wybierz nazwę rysunku i użyj tej opcji, aby rozwinąć wszystkie węzły w całym drzewie.
- Wybierz węzeł, kliknij prawym przyciskiem myszy i wybierz opcję **Zwiń wszystko**, aby zwinąć wszystkie węzły podrzędne wybranego węzła.

  Wybierz nazwę rysunku i użyj tej opcji, aby zwinąć całe drzewo.
- Kliknij węzeł lub element w panelu Struktura, aby wybrać go na rysunku.

  Uwaga: Zależy to od ustawień w **Drzewie struktury**. Aby skonfigurować Drzewo struktury, kliknij prawym przyciskiem myszy **Panel struktury** > **Konfiguruj** > **Opcje**.

  Przytrzymaj klawisz Ctrl, aby utworzyć zbiór wskazań.
- Kliknij prawym przyciskiem myszy, aby wyświetlić menu kontekstowe, które umożliwia:
  - **Pokaż** : wyświetla zaznaczenie, gdy zostało wcześniej ukryte.
  - **Ukryj** : ukrywa zaznaczenie.
  - **Izoluj** : izoluje zaznaczenie, ukrywa całą resztę.
  - **Zoom** : powiększa zaznaczenie.

Uwaga: Uwaga : Wybierz nazwę rysunku. Następnie kliknij prawym przyciskiem myszy i wybierz **Pokaż** z menu kontekstowego, aby wyświetlić cały rysunek, lub wybierz **Pokaż elementy** na karcie **Ogólne** w Kwadracie bez wyboru.

4. Interfejs użytkownika

4.1 The BricsCAD application window

The components of the BricsCAD application window are:

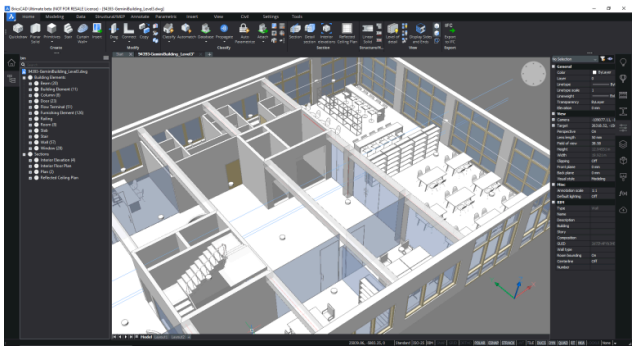
- Menu bar
- Toolbars
- Ribbon Document tabs
- Model space viewports
- Paper space viewports
- Model and Layout tabs
- Dockable panels
- Command line
- Status Bar

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

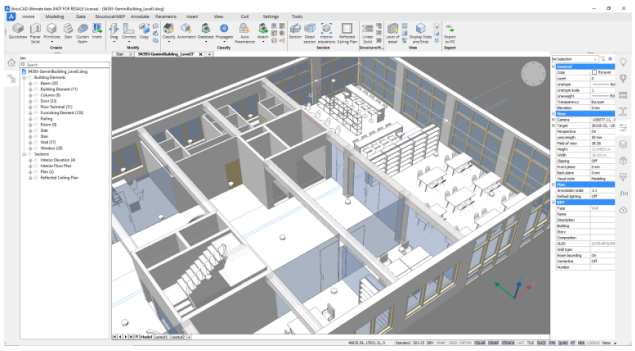
4.1.1 The color theme

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.

Uwaga:

- 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**.

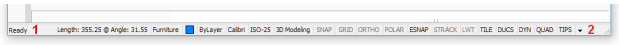
4.3 The Status bar

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.

Uwaga:

- 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**.

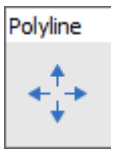
4.6.3 Editing the Quad settings

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.

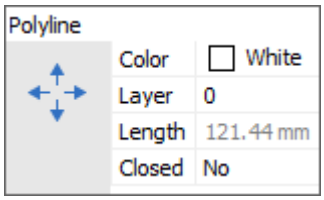
4.6.4 Using the Quad

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.

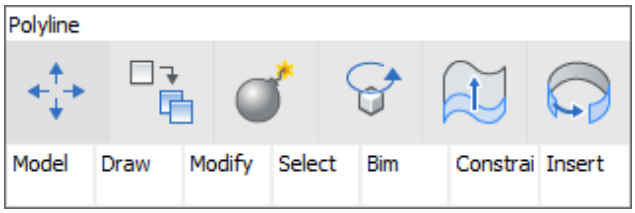


Launching the default command

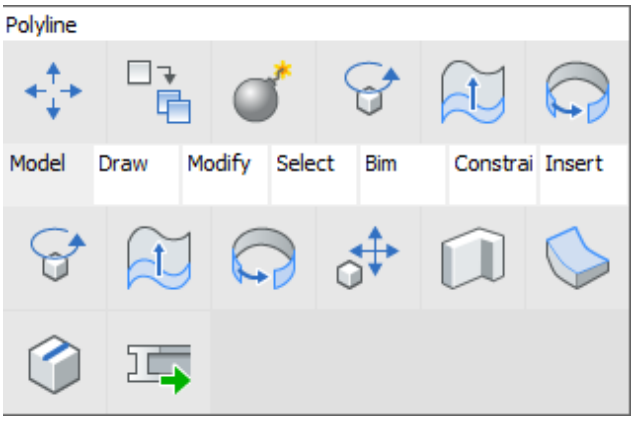
- If QUADCOMMANDLAUNCH = 1, right-click without entering the Quad to launch the command.
- If QUADCOMMANDLAUNCH = 0, click the command button.

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.

Uwaga: 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.

Uwaga:

- 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.

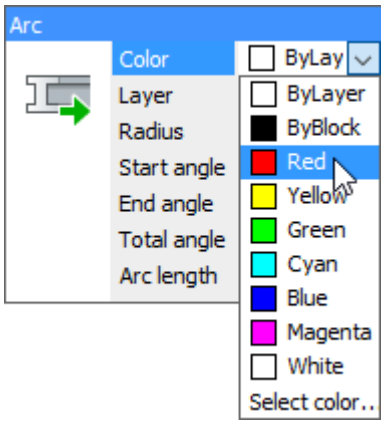
The availability of Rollover Tips is controlled through the ROLLOVERTIPS system variable.

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.

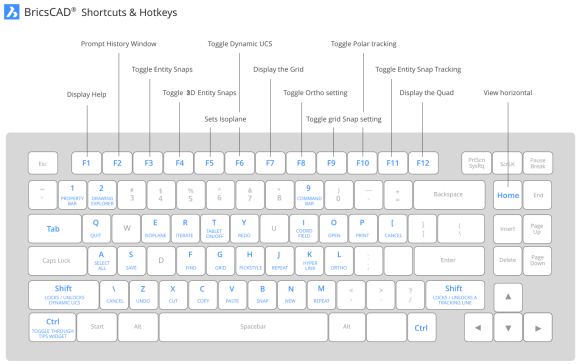
4.8.1 Controlling the display of prompt menus

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.

Uwaga: 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	***Uwaga:*** 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	***Uwaga:*** 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	***Uwaga:*** 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	***Uwaga:*** 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.

Uwaga:

- 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.

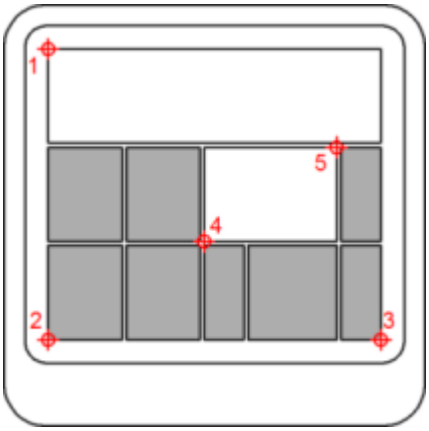
2 Choose the **Configure** option.

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:

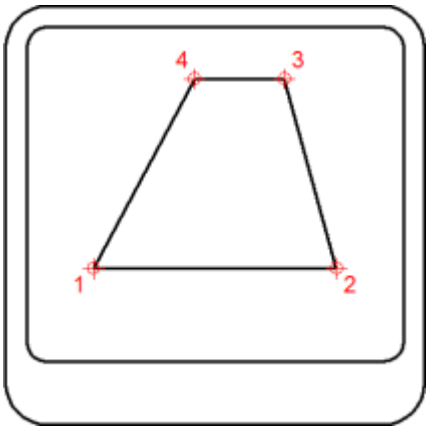
4 Do one of the following:
- Click a point in the BricsCAD drawing window.
- Enter the coordinates (X,Y) in the Command line.

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 stronie 166	
	Attachments panel na stronie 168	
	BIM BCF panel na stronie 170	
	BIM compositions panel na stronie 171	

Icon	Title	Command
	BIM profiles panel na stronie 173	
	BIM project browser panel na stronie 175	
	BOM manager panel na stronie 178	
	Bricsys 24/7 panel na stronie 185	
	Command line panel na stronie 187	
	Content browser panel na stronie 188	
	Details panel na stronie 190	
	Drawing customizations panel na stronie 192	
	Layers panel na stronie 199	
	Library panel na stronie 205	
	Mechanical browser panel na stronie 208	
	Parameters and constraints panel na stronie 212	
	Properties panel na stronie 214	
	Render materials panel na stronie 221	
	Report panel na stronie 223	
	Ribbon	

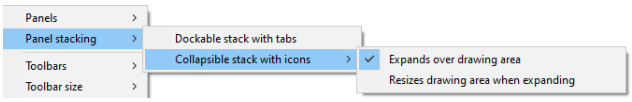
Icon	Title	Command
	Sheet sets panel na stronie 225	
	Structure panel na stronie 228	
	Tips panel na stronie 230	
	Tool palettes panel na stronie 231	Tool palettes na stronie 852

4.11.3 Setting the stack type

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

A context menu displays.

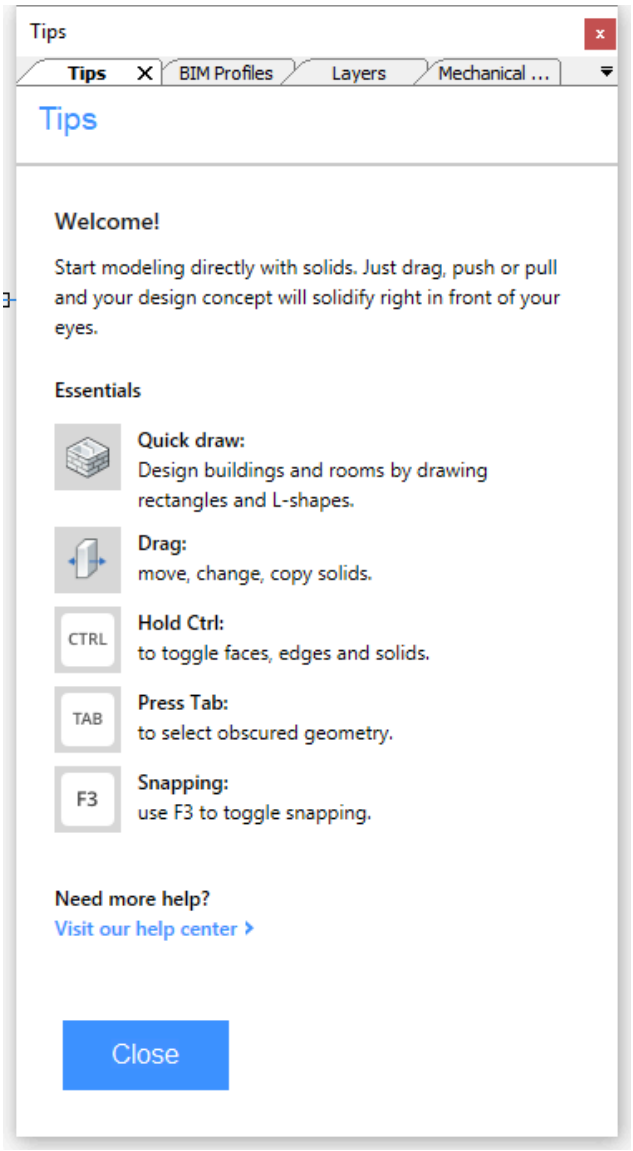
2 Choose **Panel stacking**.



3 Select the desired stack type.

Dockable stack with tabs

Panels are docked in a tabbed panel.

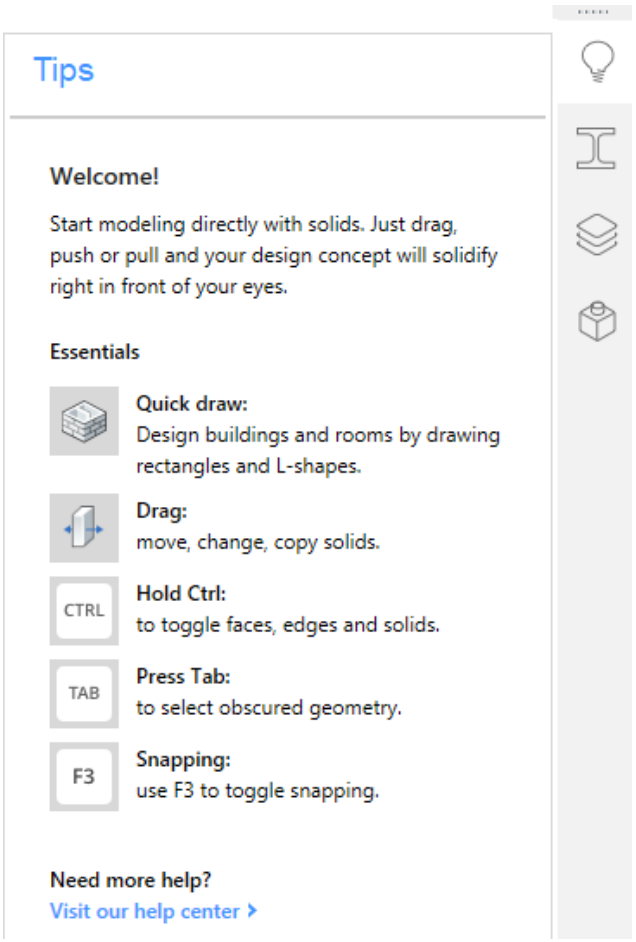


- Click a tab to display the panel.
- Press and hold the left mouse button to drag a tab to re-arrange the tab order.
- Double click a tab to make the panel floating.
- Double click the title bar of floating panel to dock it at its previous position.

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.

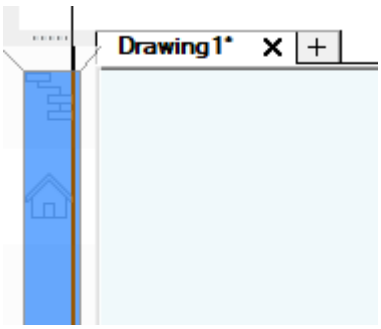


- Click an icon to expand a panel.
- To collapse an expanded panel click its icon again.
- Press and hold the left mouse button to drag an icon to re-arrange the icon order.
- Double click the icon to make the panel floating.
- Double click the title bar of a floating panel to dock it at its previous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

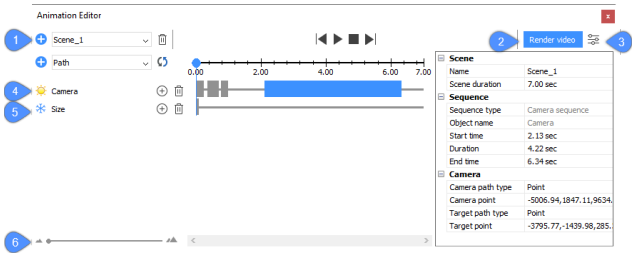


2 Move the cursor over the center docking location then release the left mouse button.

Uwaga: 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

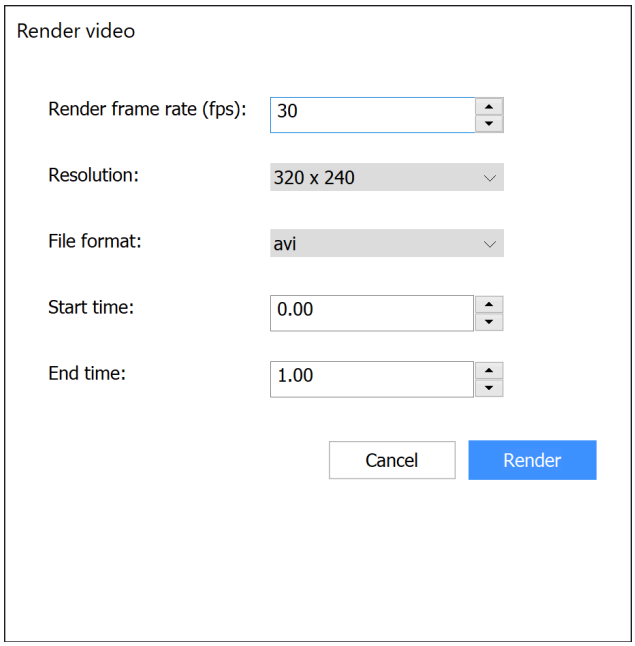
4.11.5.1 Add scene

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

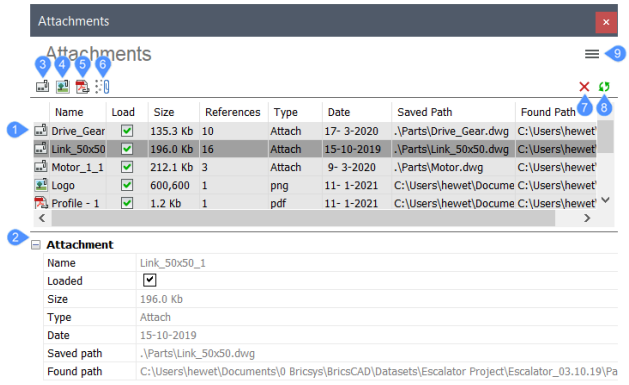
Enable timeline effects.

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.

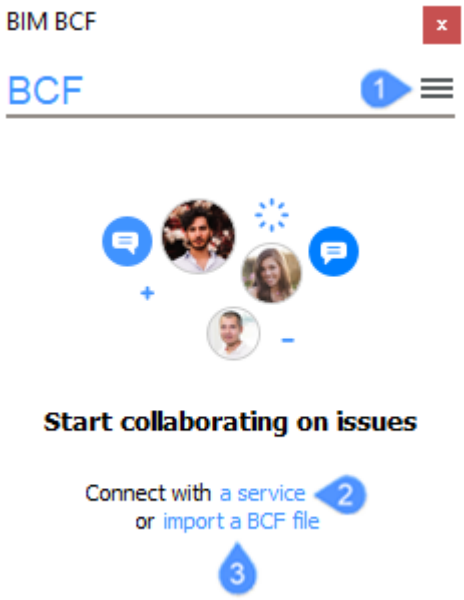
4.11.6.17 Menu

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

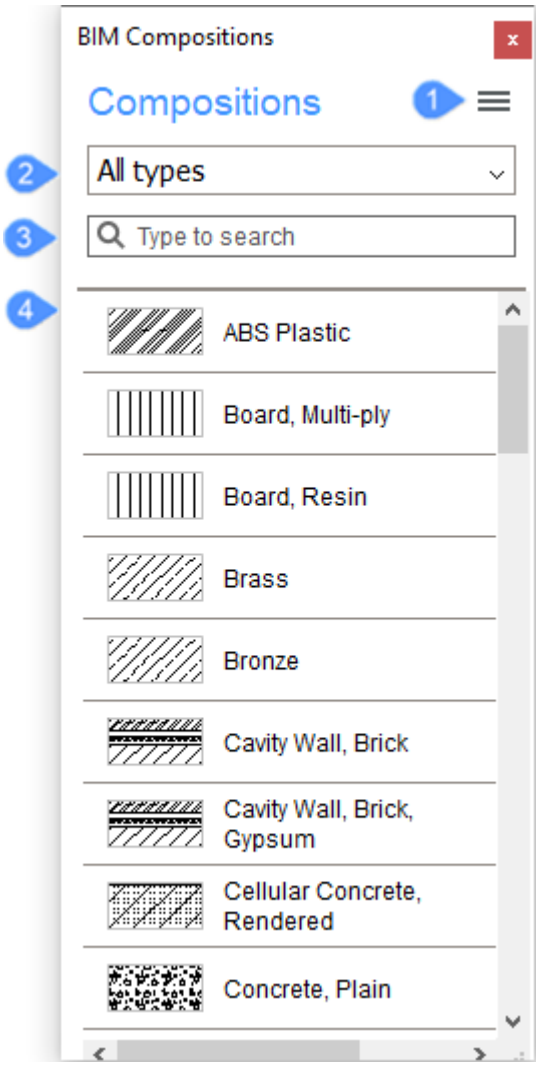
Opens the Custom log in dialog box.

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.

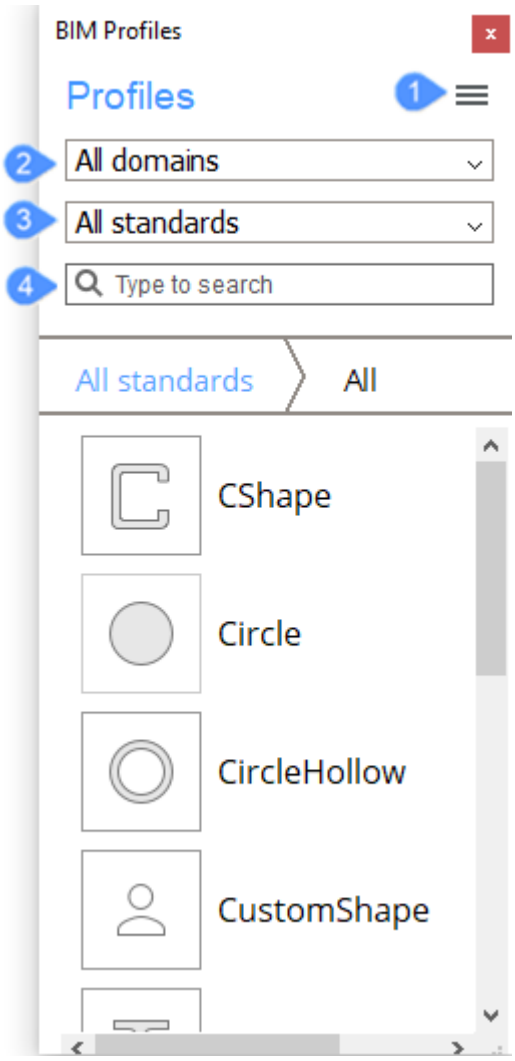
4.11.8.4 Available compositions

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

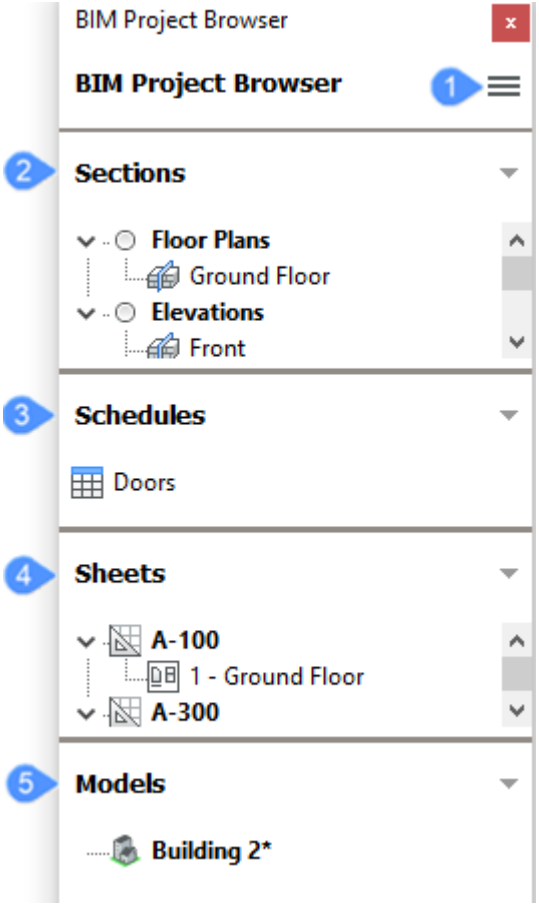
Filter profiles based on the country standard.

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

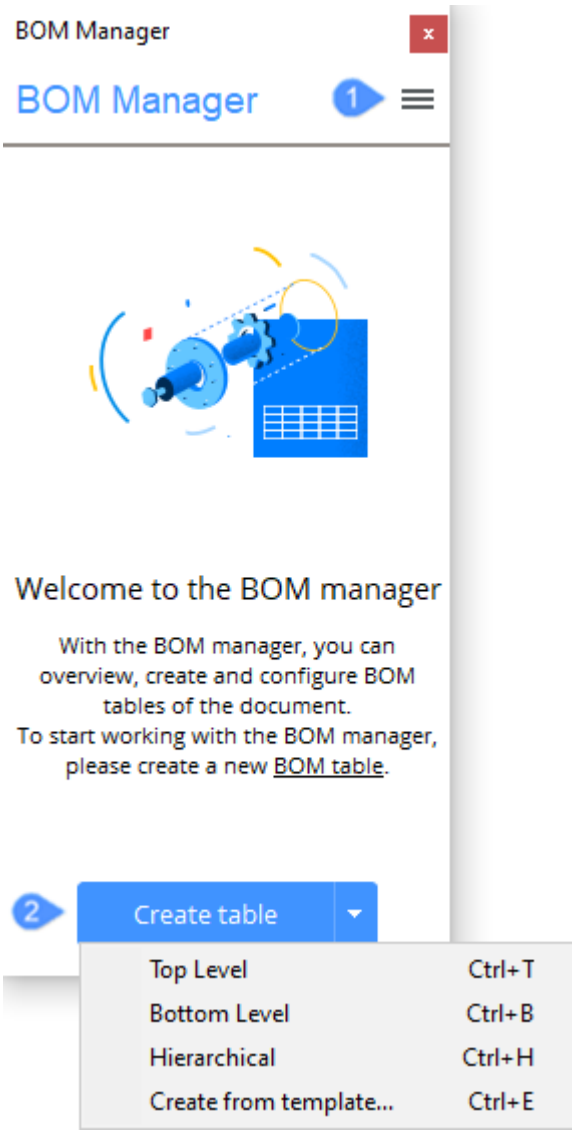
The BOM Manager panel allows you to browse, create and edit bills of materials (BOMs) in the current drawing.

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.

Uwaga: 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

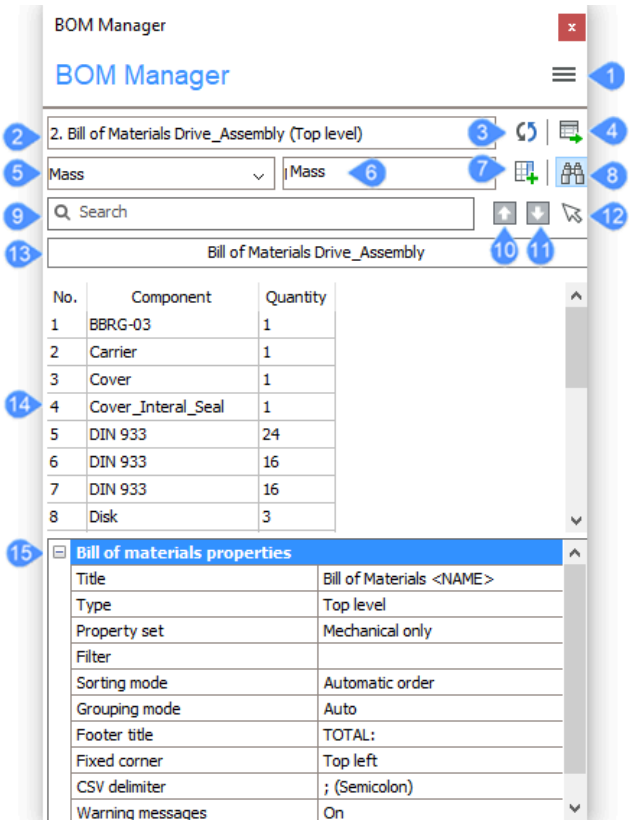
Top level
Creates a new Top level BOM.

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.

  Uwaga: 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.

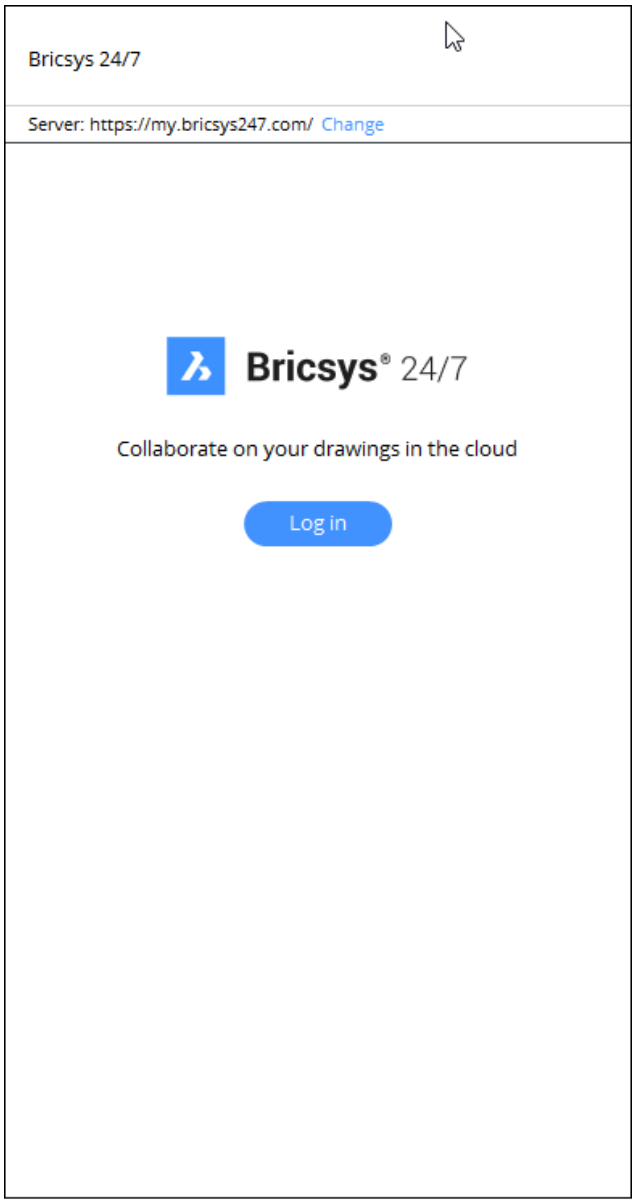
BOM Properties
Displays the BOM and columns Properties. The columns properties will appear after you insert a column on the BOM table.

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.



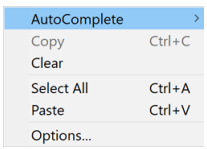
1 Command line

2 Command history

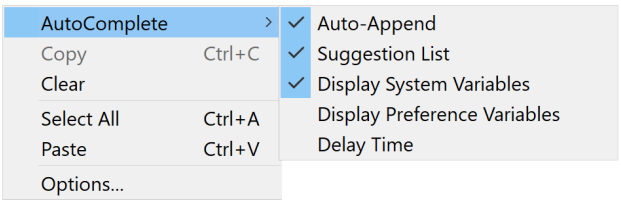
4.11.13.1 Command line

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.

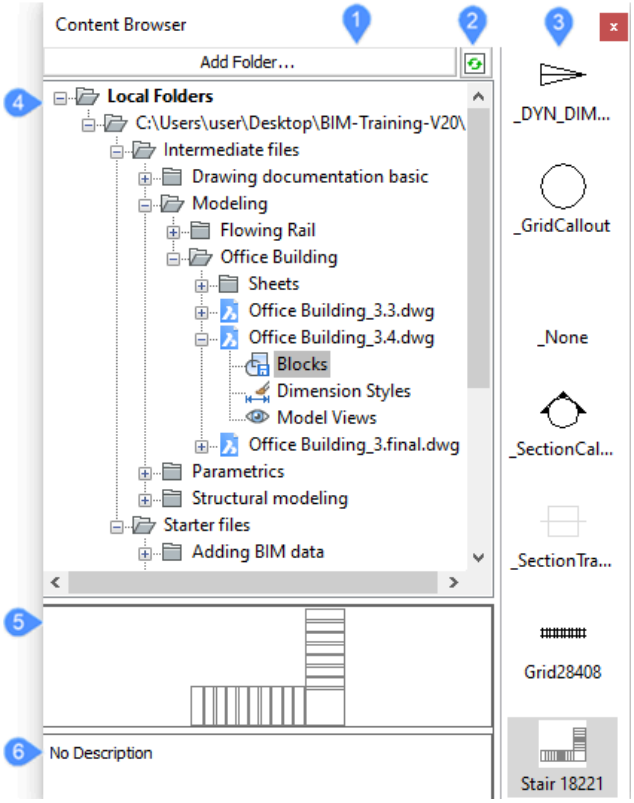
Options
This option displays the Settings dialog box at the Command line section.

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

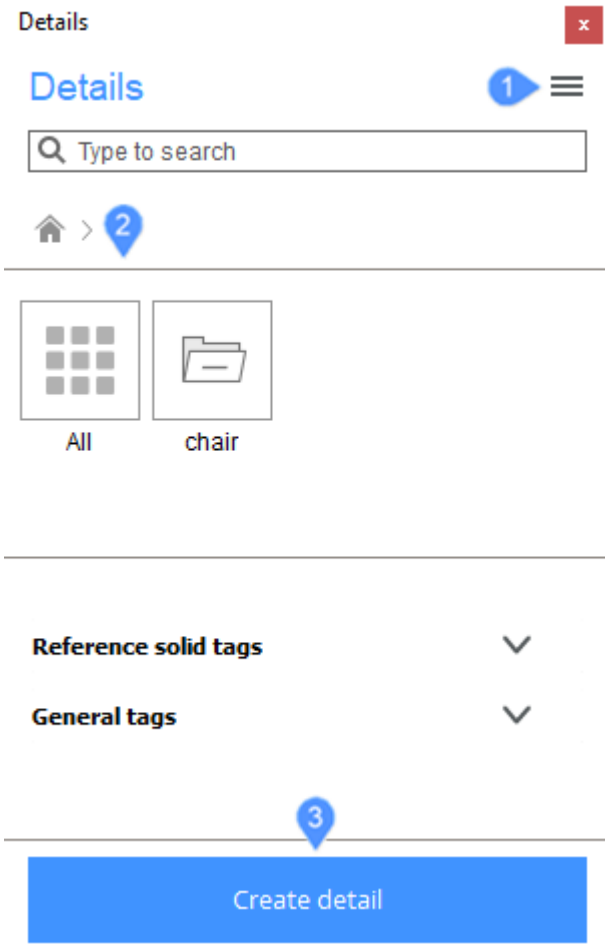
Shows a preview of your drawing/block/dimension style.

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.

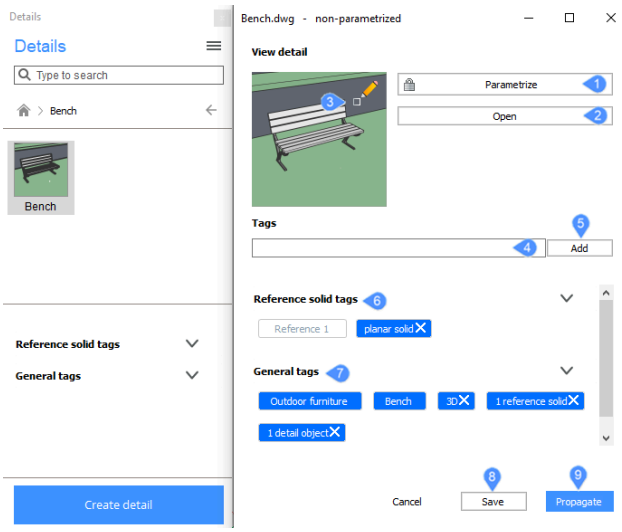
Generate thumbnails
Loads the new details in your library panel.

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.



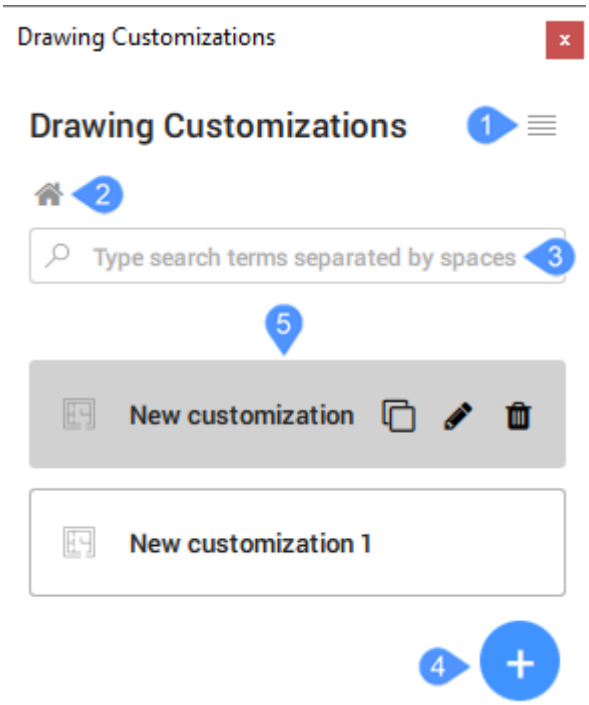
1 Parametrize
2 Open
3 Edit detail
4 Tags
5 Add tag
6 Reference solid tags
7 General tags
8 Save
9 Propagate

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.

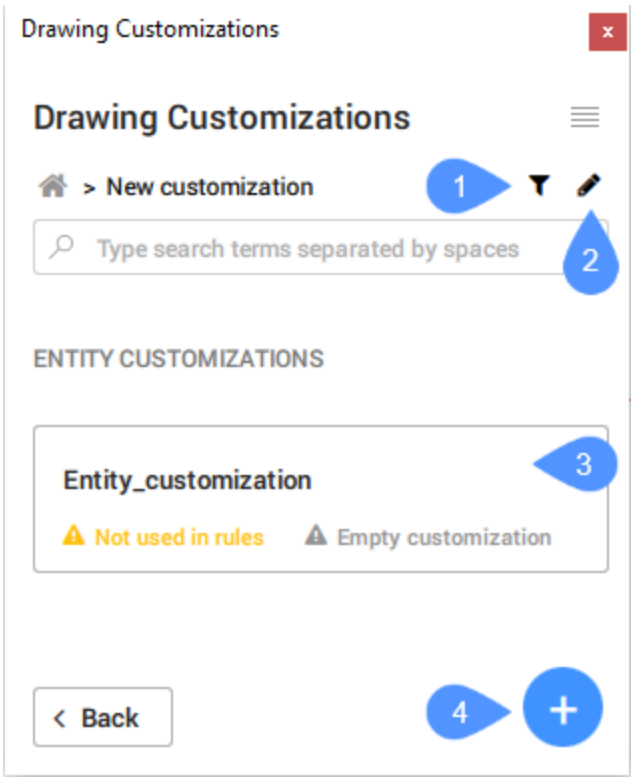
Duplicate
Creates a copy of your selected drawing customization.

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.

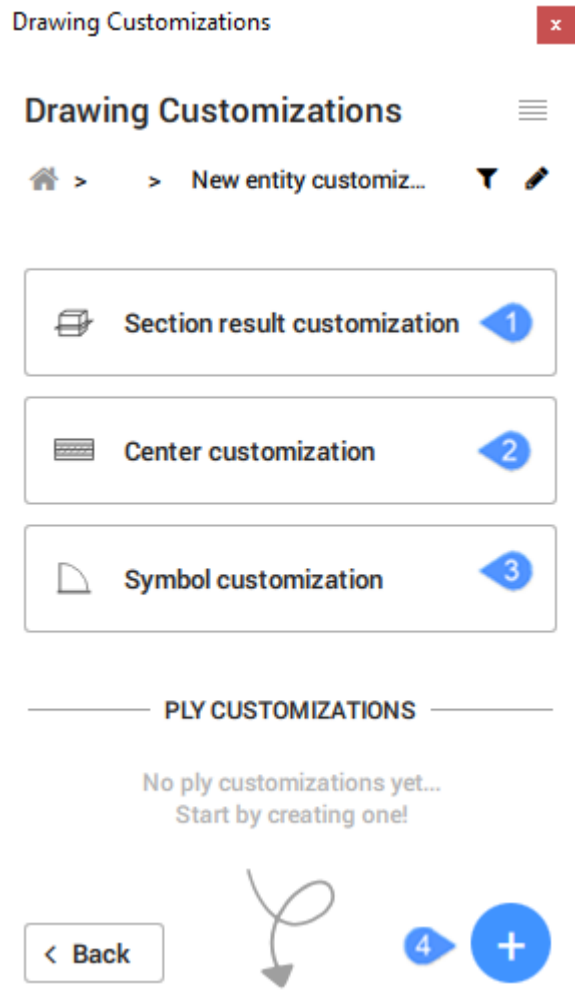
To style definitions

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

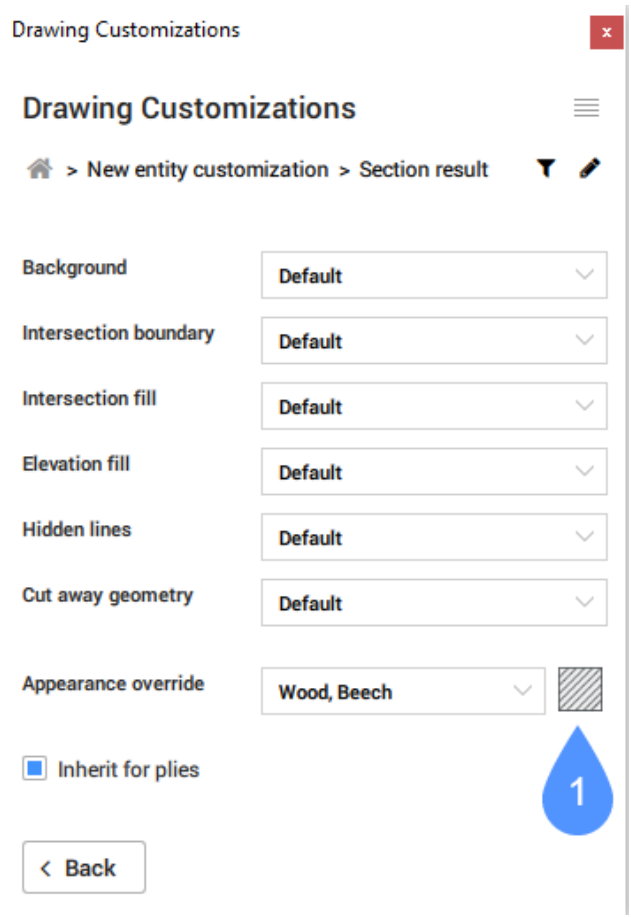
2 Center customization

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.

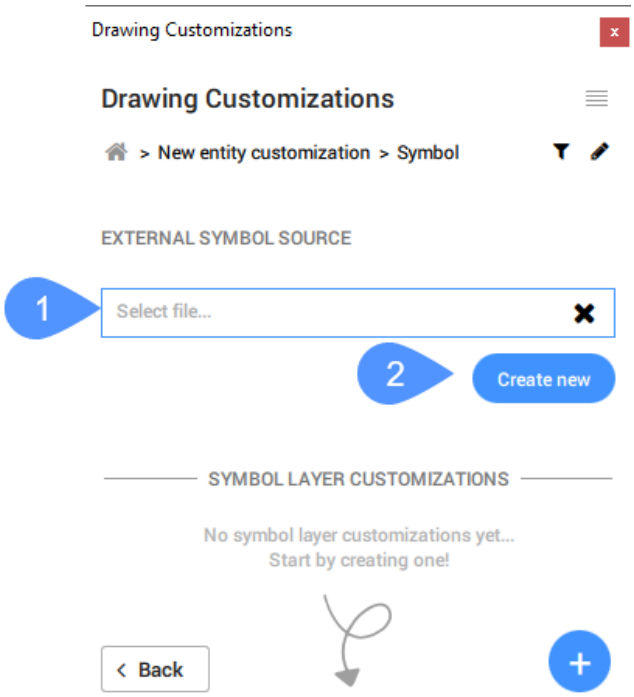


1 Physical Materials

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.

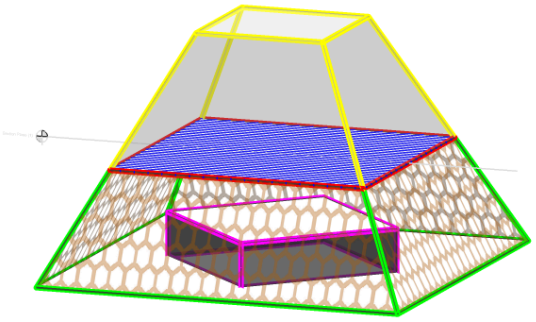


1 Select file...

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.

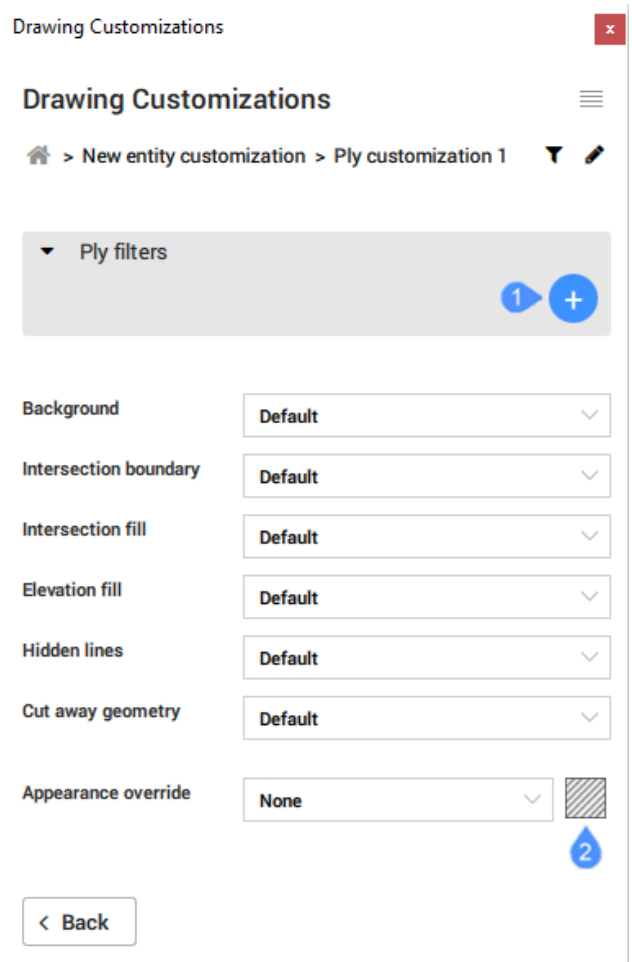
- Green outline represents**Background**, which is essentially the elevation outline of the cut object.
- Red outline represents**Intersection Boundary**, which traces over the cut area.

- 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

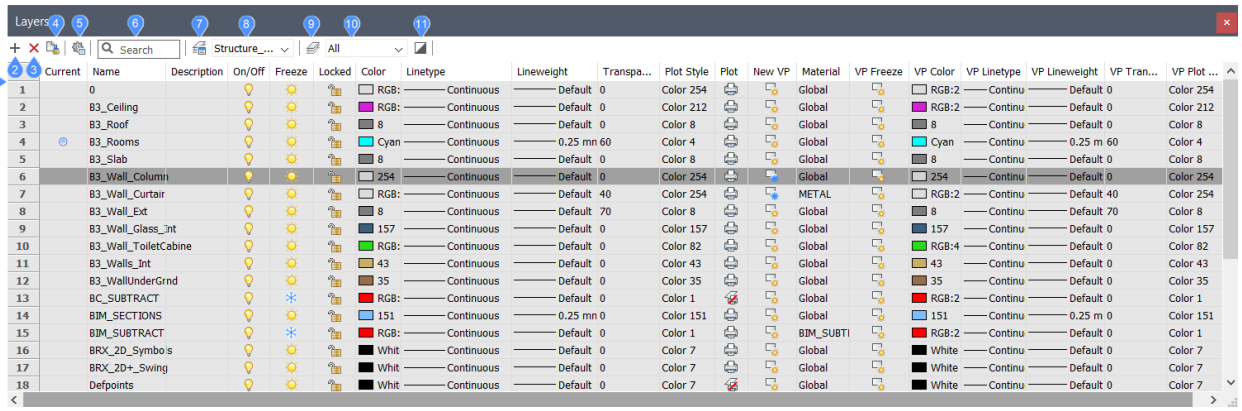
Add default function

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.

Uwaga: 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.

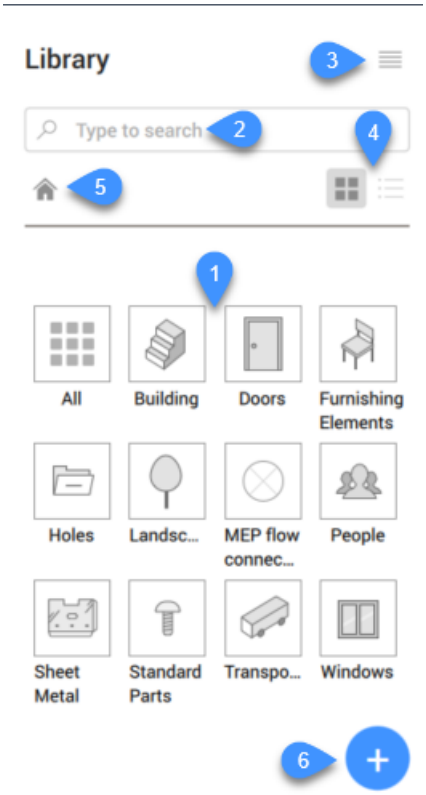
4.11.17.3 Search

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

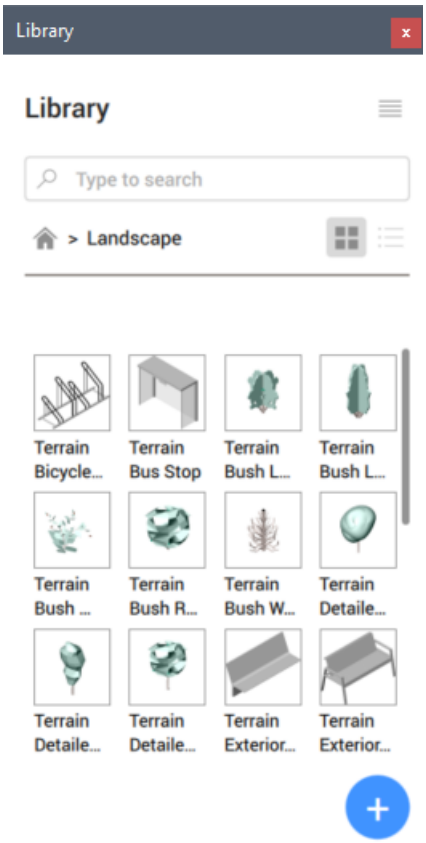
4 View

5 Home

6 Add

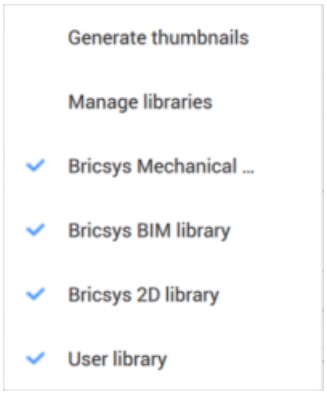
4.11.18.1 Block library

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

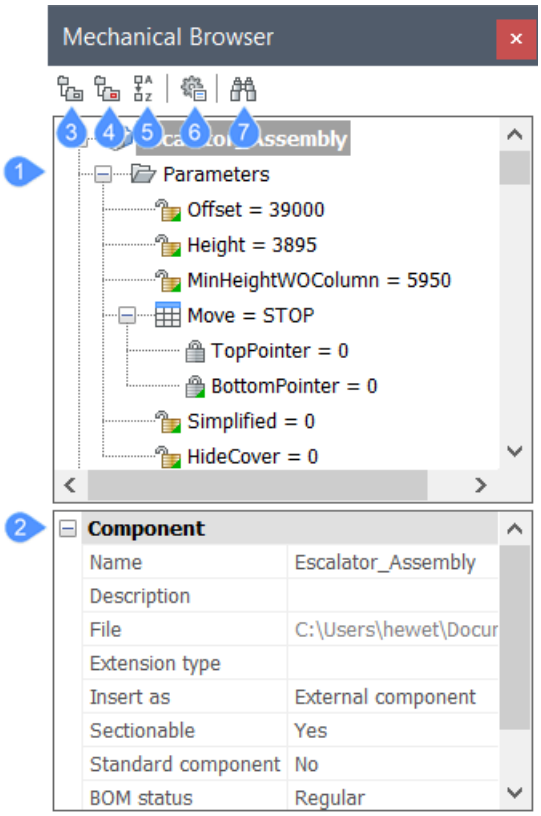
Returns the block library to the home screen.

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.

Uwaga: 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.

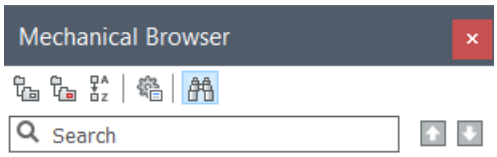
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

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

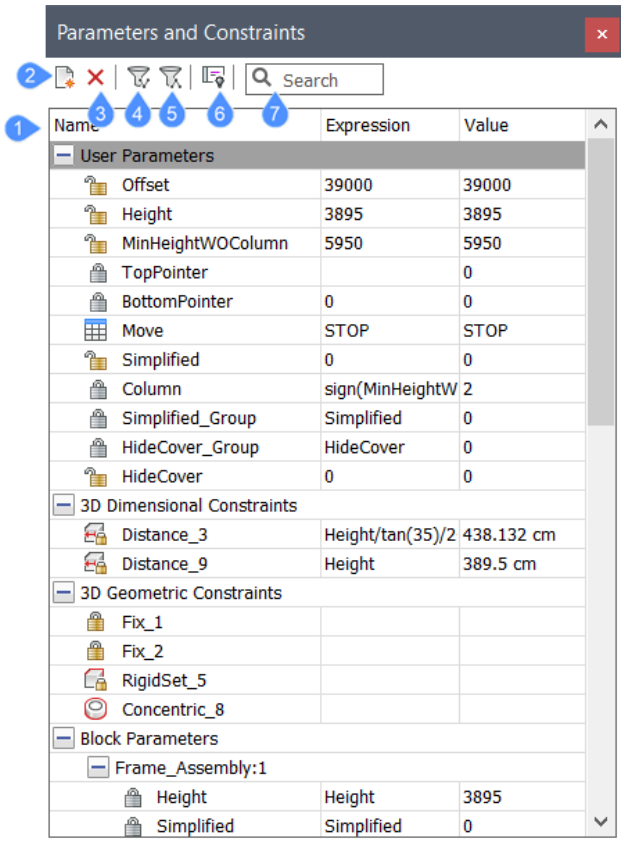
Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

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

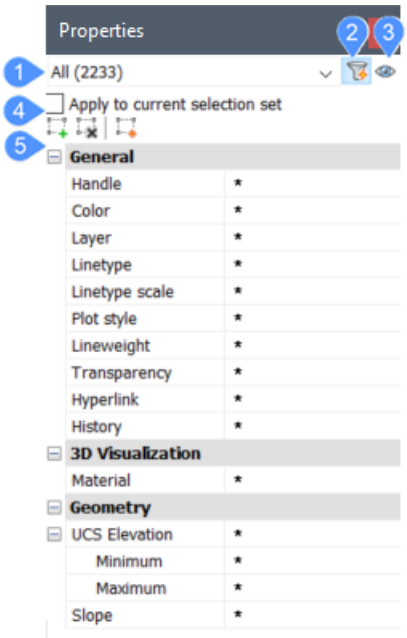
0

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.

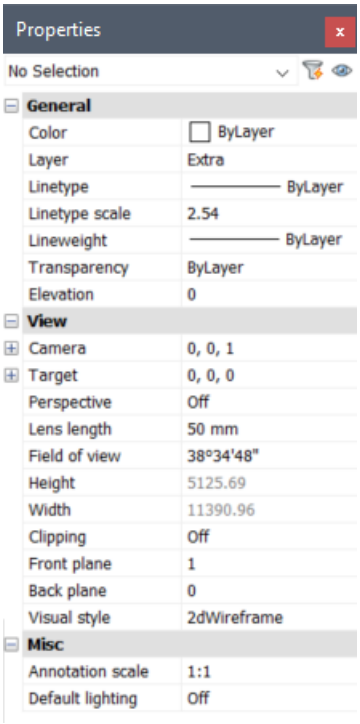
Entities list

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.

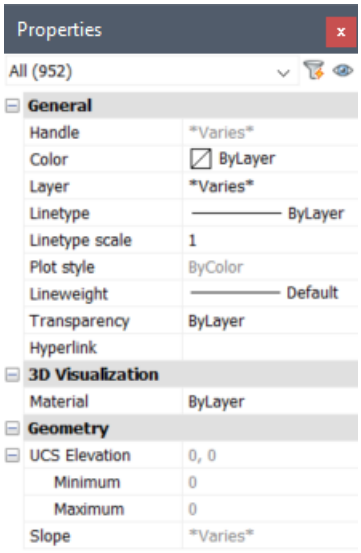
Default Lighting

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.

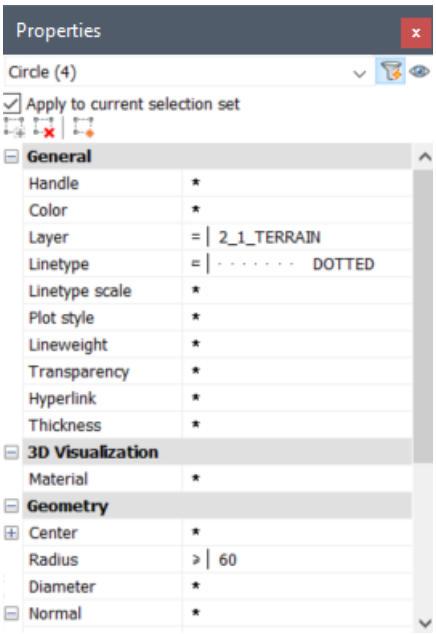
3D Visualization Properties
3D Visualization properties describe the materials assigned to the selected entities.

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

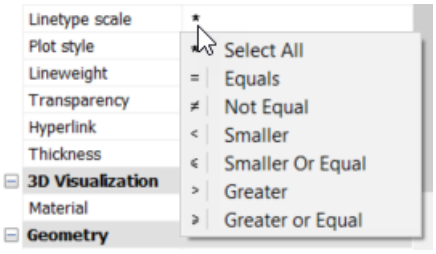
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is disabled since entities can only be removed from an existing selection set.

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.

Uwaga: 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.

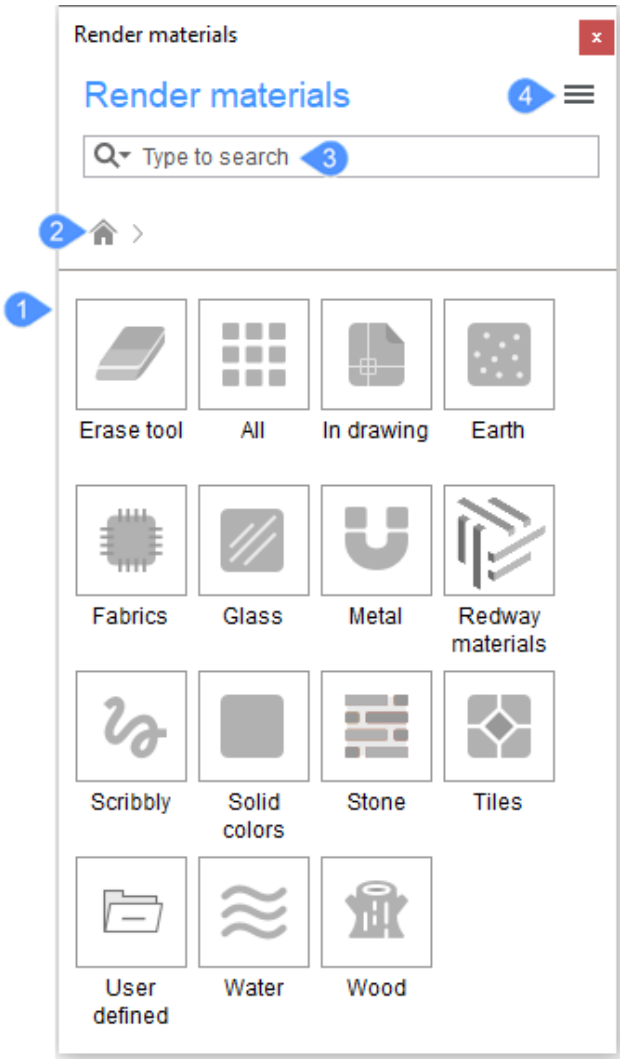
Material
Enables you to filter entities based on the material of the entity. You can specify a material from the drop-list.

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.

Uwaga: 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.

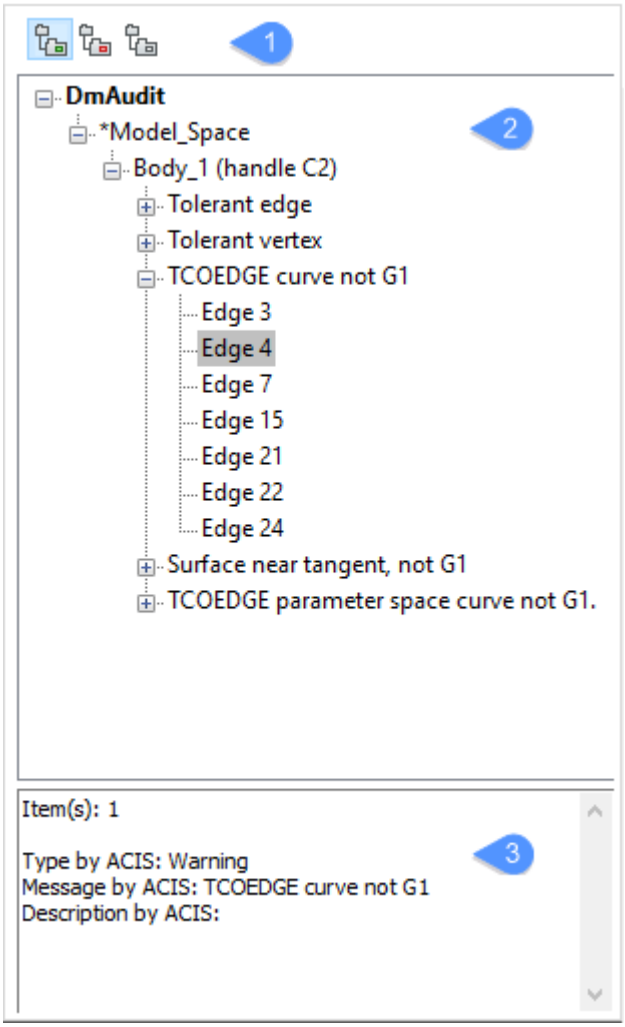
Manage libraries
Opens the Settings dialog box to change the render materials directory path.

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

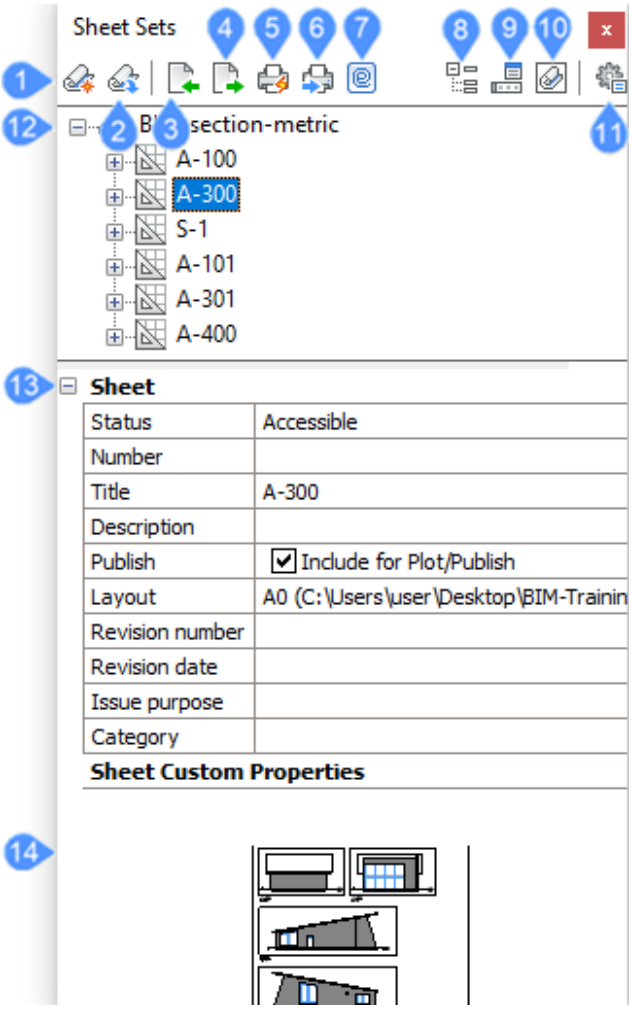
Displays command output.

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...

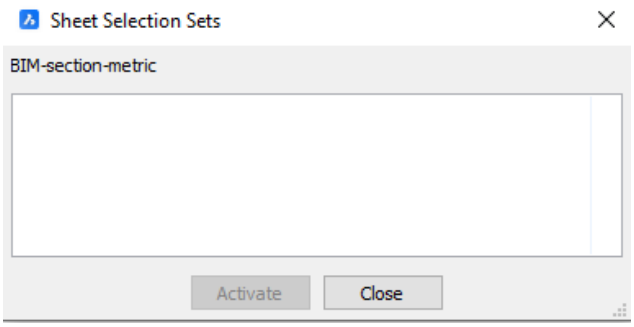
Publishes the currently selected sheet set(s), subset(s) or sheet(s). Opens the Publish dialog box.

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

Uwaga: 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.

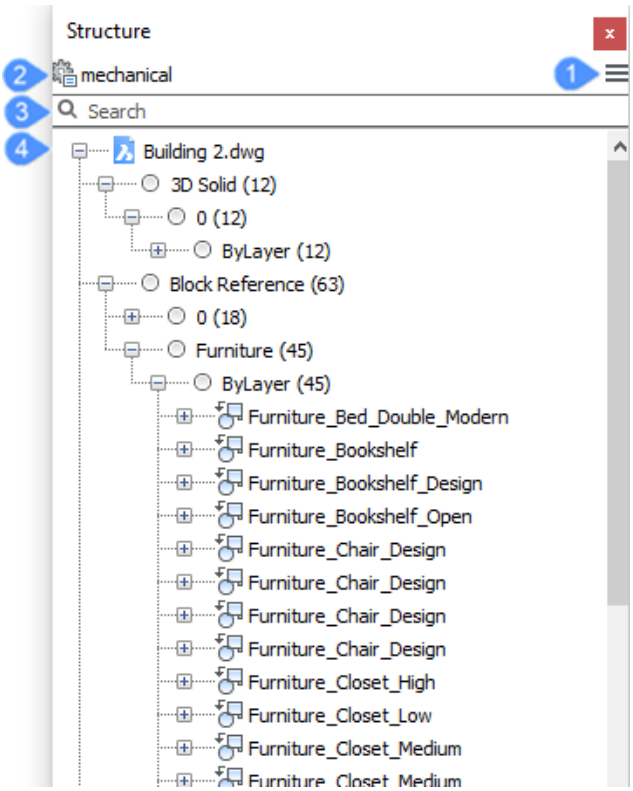
4.11.24.1 Sheet preview

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.

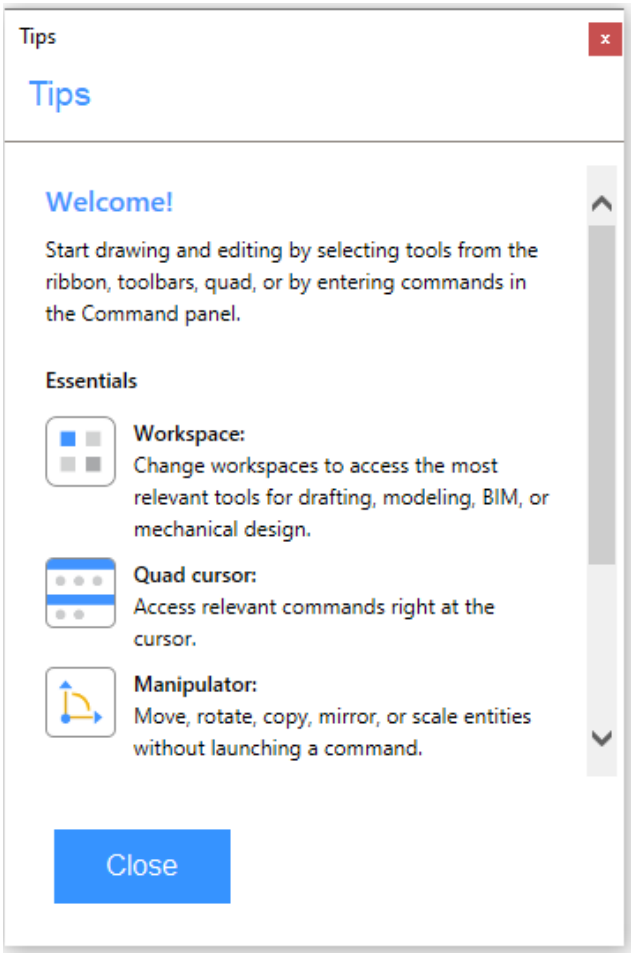
4.11.25.4 Content tree

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.

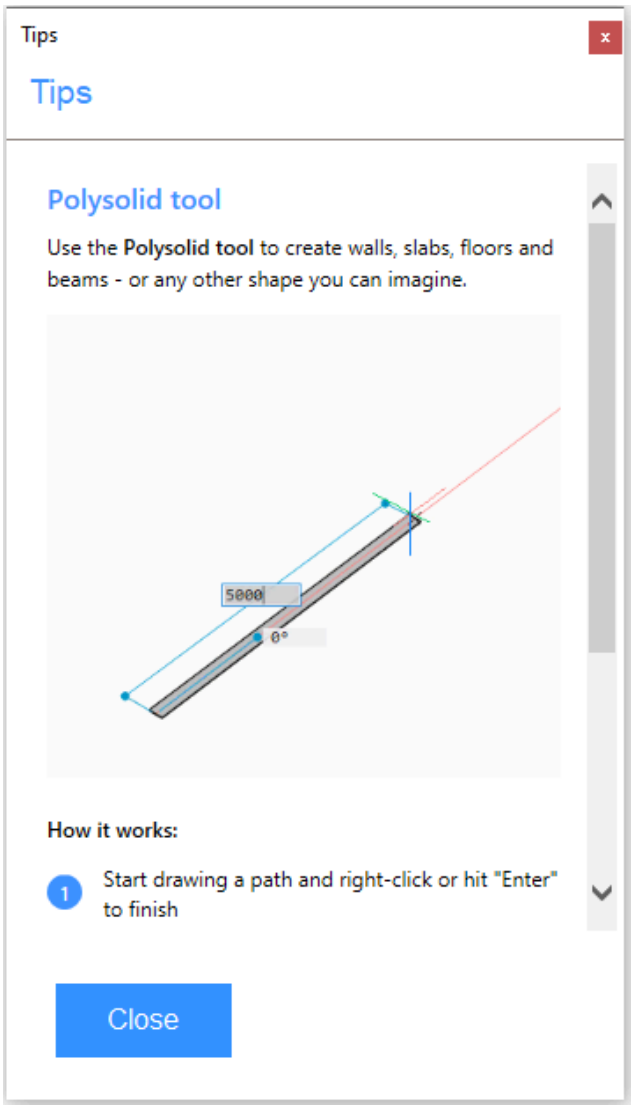
A right-click menu on a tree node or on an entity offers relevant tools.

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.

> ***Uwaga:*** 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.

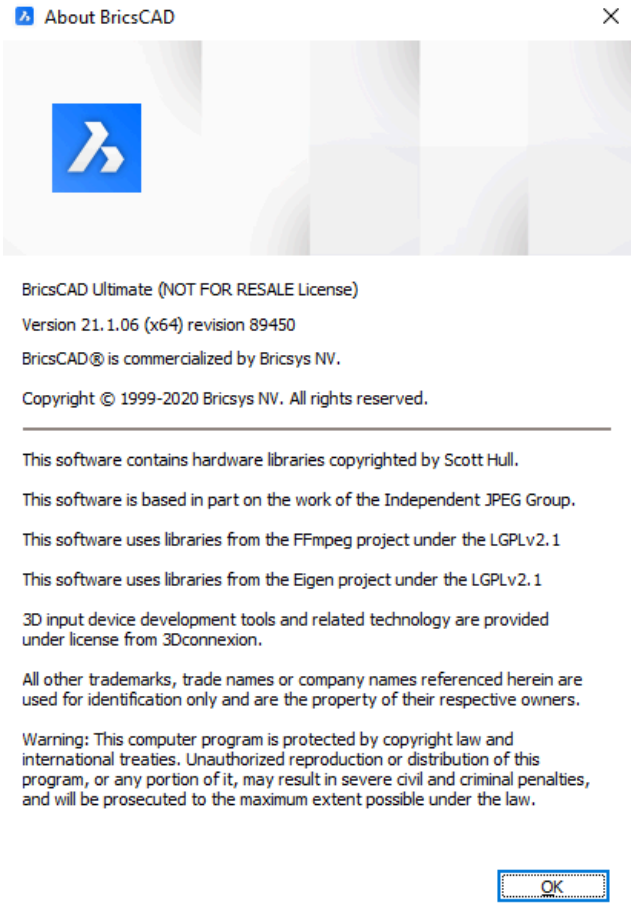
Add Tool…:
Opens the Customize dialog box, which allows you to add a new tool.

4.12 Dialog boxes

4.12.1 About BricsCAD dialog box

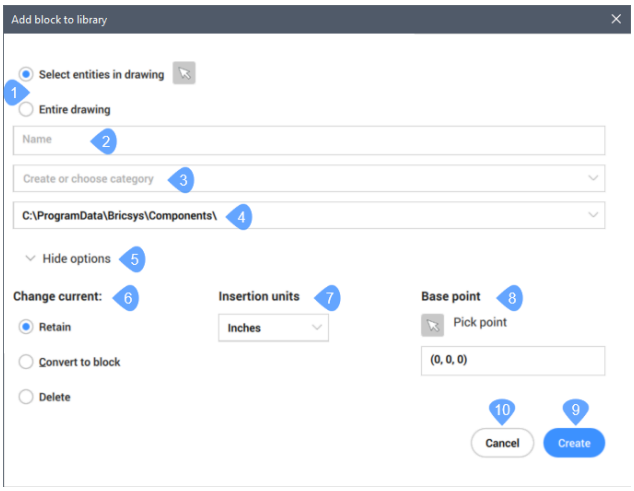
The About BricsCAD dialog box displays copyright and product information.

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

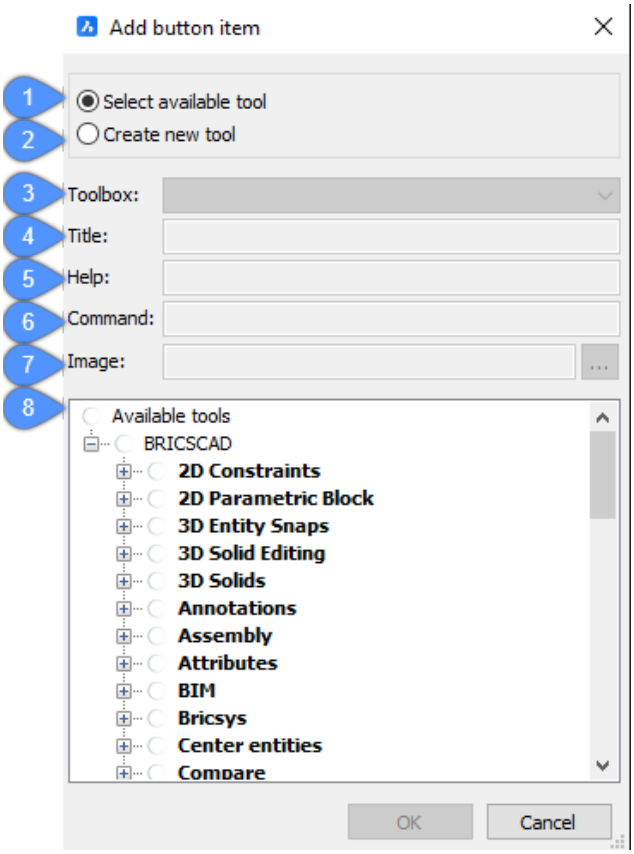
Creates a DWG file for the block definition in the specified location, adds the block to the library panel under the specified category, and closes the dialog box.

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

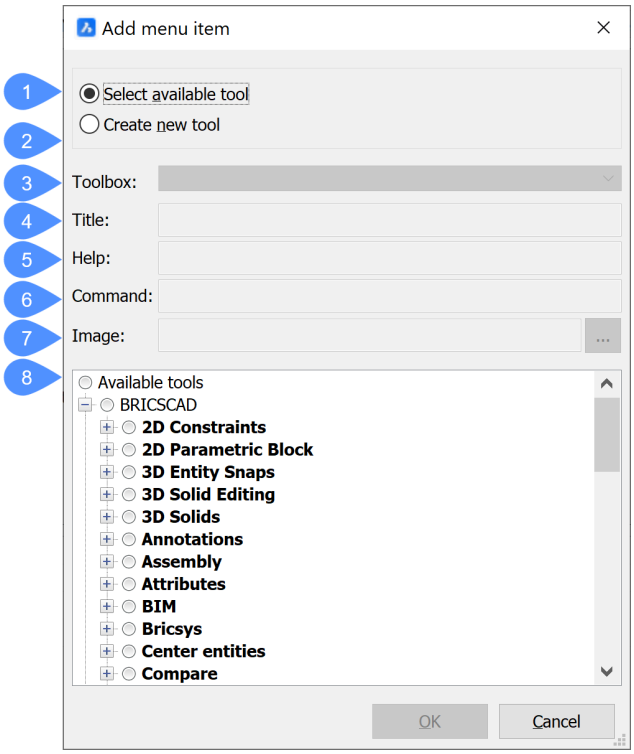
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

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

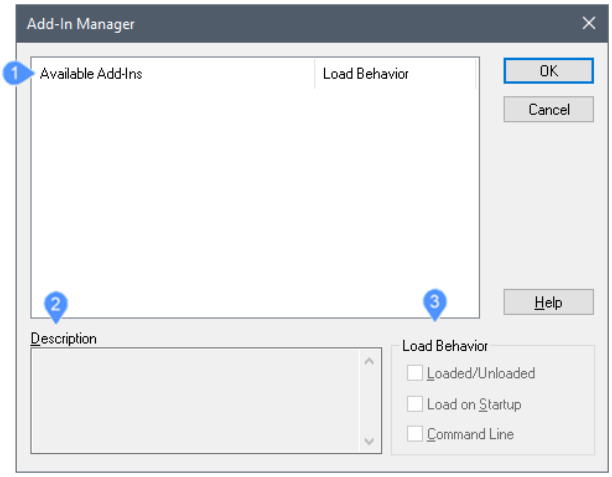
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

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.

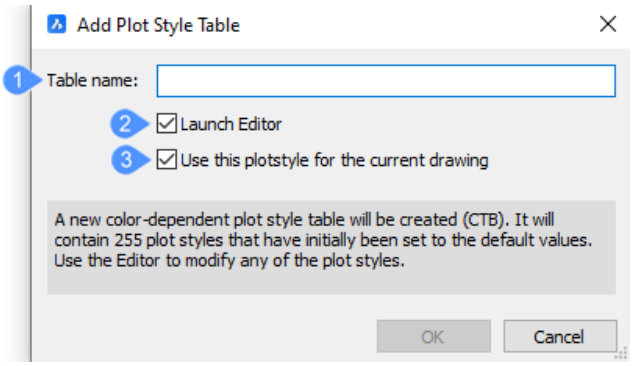
Load on Startup
Loads modules when BricsCAD starts up.

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

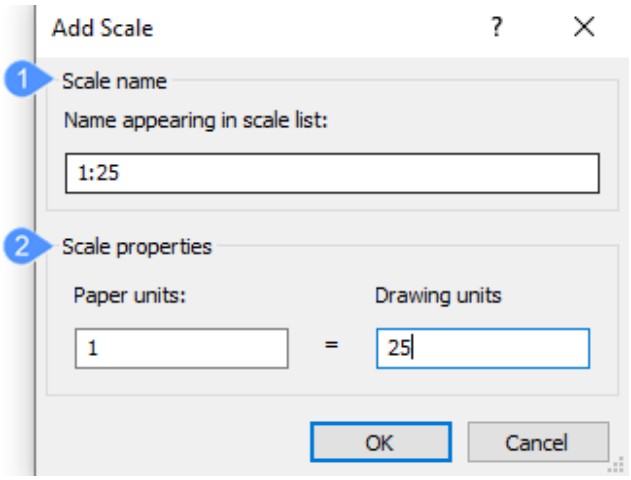
Opens the Plot style table editor dialog box after you enter the name and press the OK button.

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

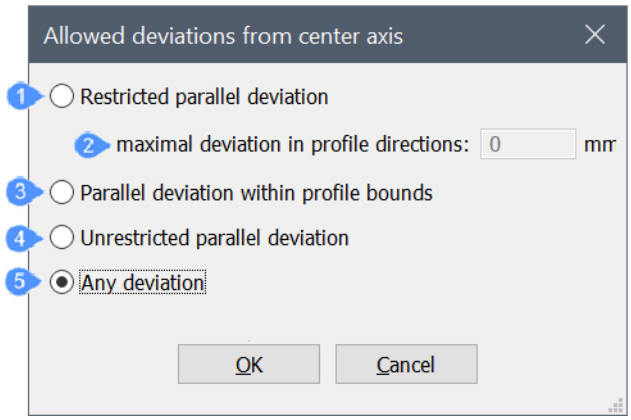
Specifies the scale name as it will appear in the scale list.

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.

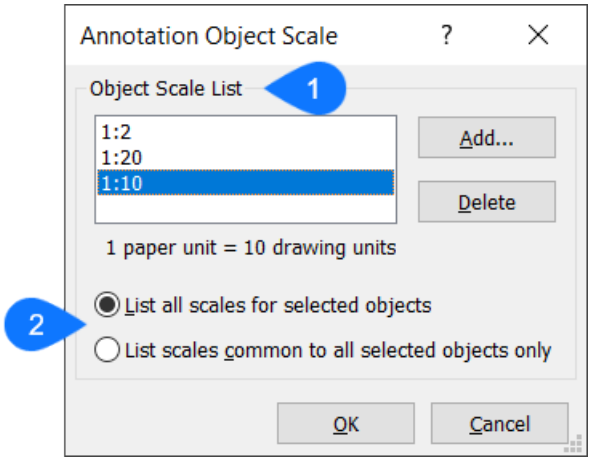
4.12.8.5 Any deviation

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.

Uwaga: This option does not work if there is just one scale factor attached to the entities.

4.12.9.2 Options

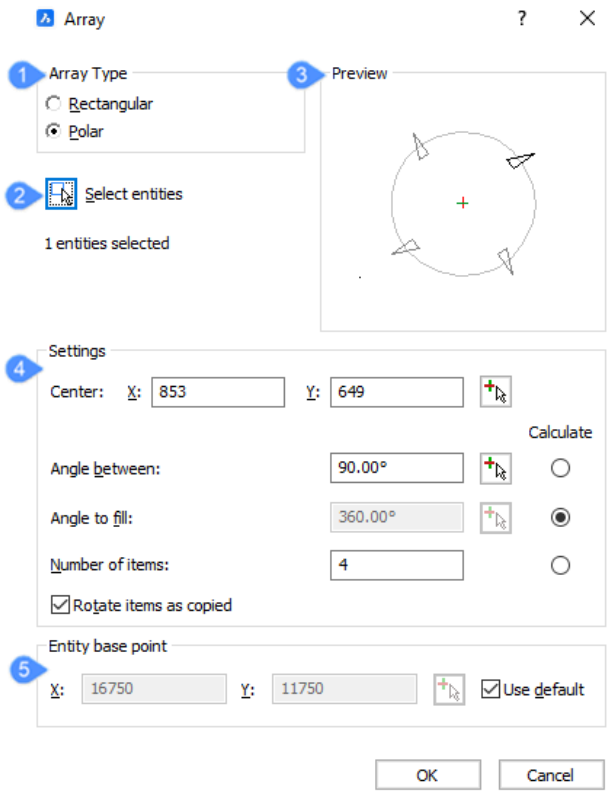
List all scales for selected objects
Lists all the scale factors.

List scales common to all selected objects only
Lists only the scale factor shared by the selected entities.

Uwaga: 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.

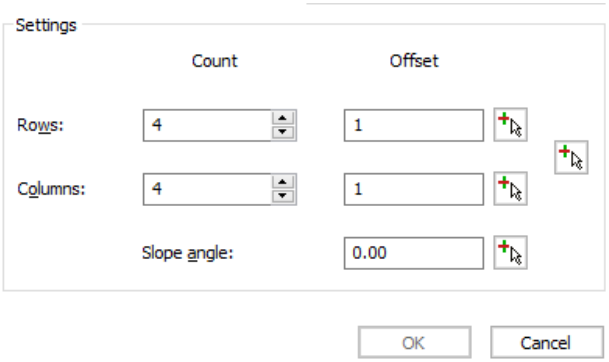
4.12.10.3 Preview

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

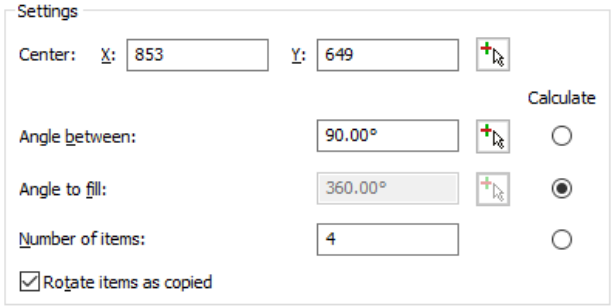
Enables you to type an angle for the slope.

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.

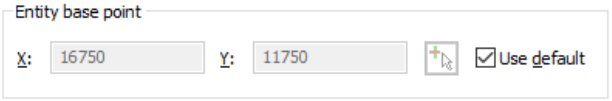
Calculate
Enables you to specify the number of items based on the angle between items and the fill angle. Choosing the Calculate radio button disables the Number of items options and enables the Angle Between and Angle to Fill options.

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.

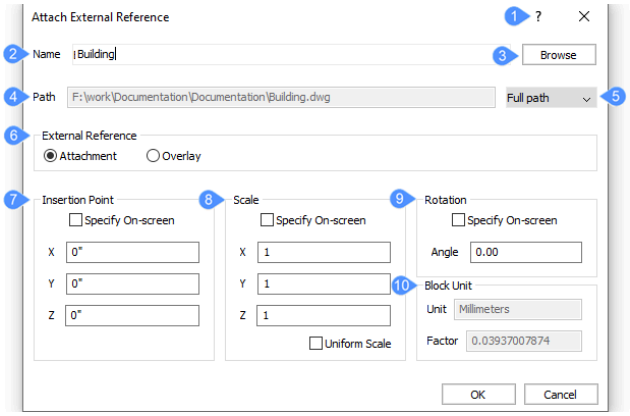
Y field
Enables you to type a coordinate to specify the basepoint of array items along the Y axis.

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

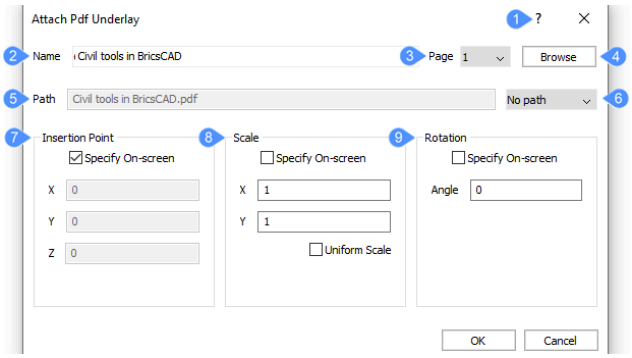
Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm:

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:

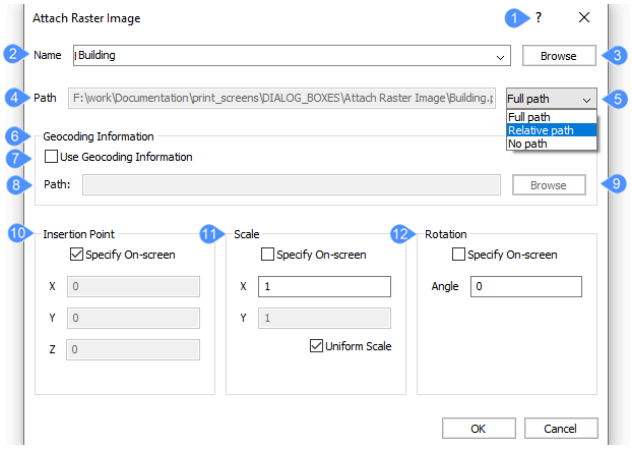
Specify On-screen
Determines how the rotation angle is specified:

- 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.

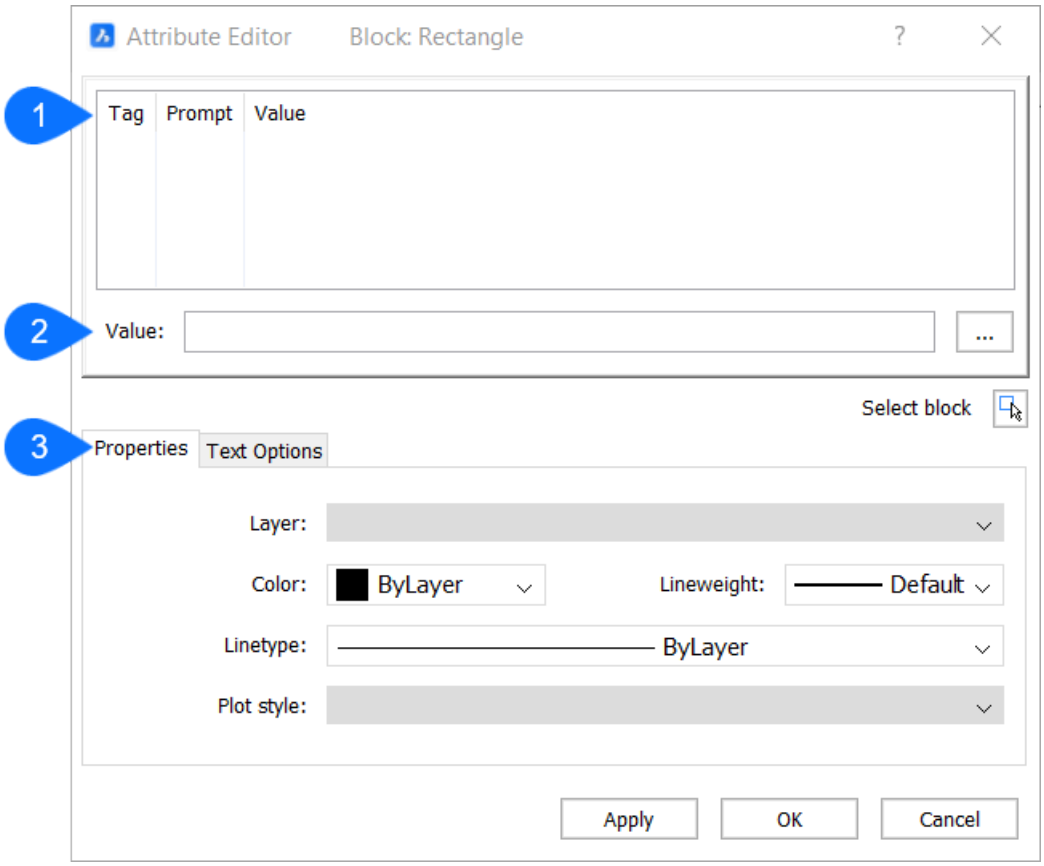
- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

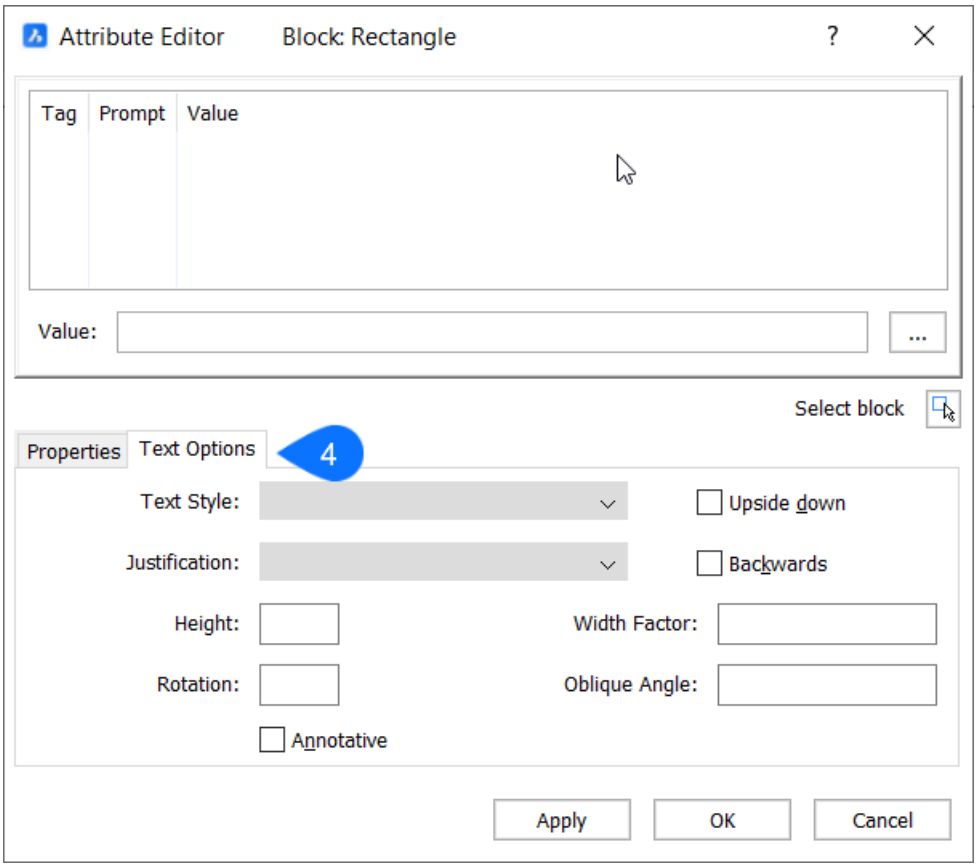
4.12.13.1 Angle

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.

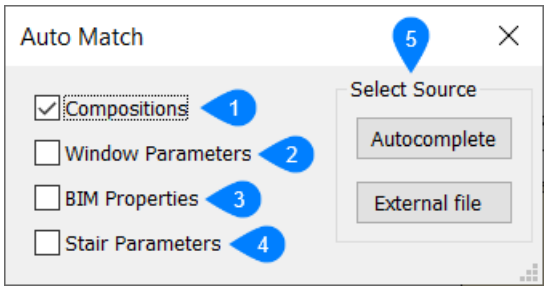
Rotation
Specifies the rotation angle of the text in the selected attribute.

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.

Uwaga: 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.

Uwaga: 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.

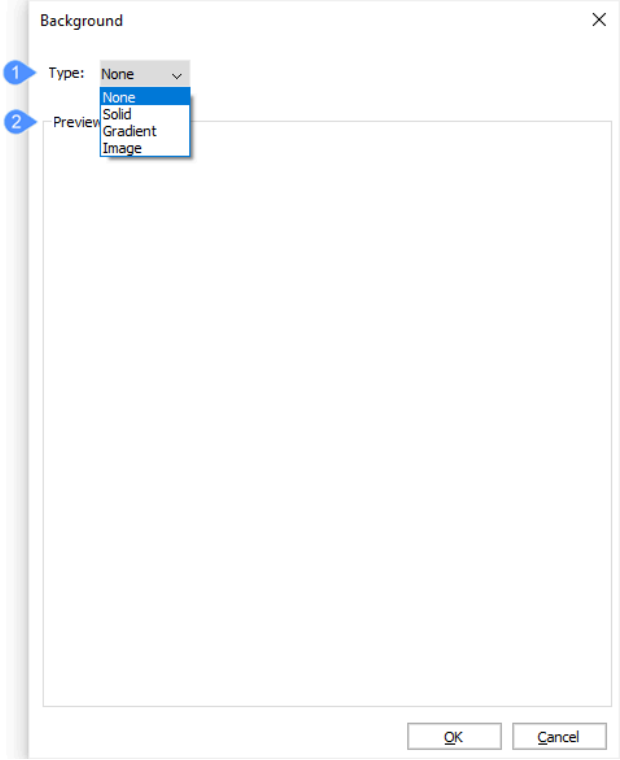
4.12.16 Background dialog box

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.

Uwaga: 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.

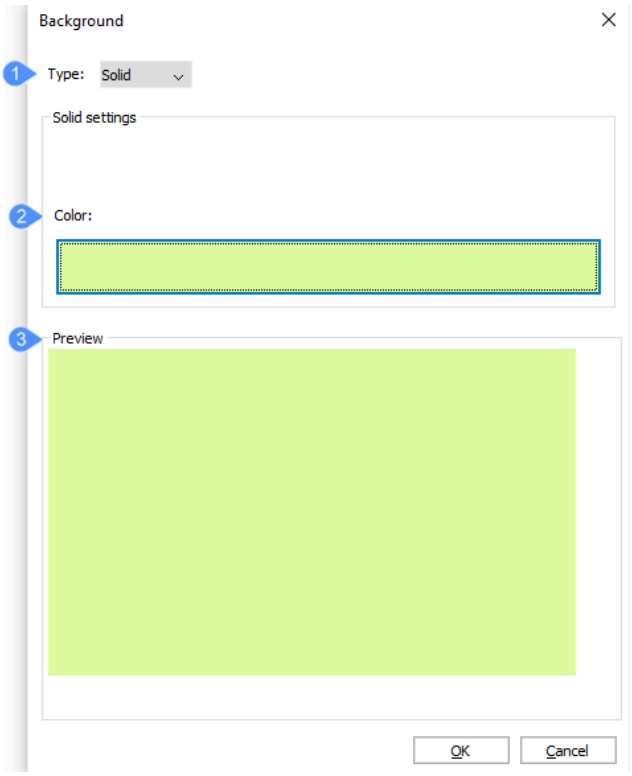
Preview
Displays a preview of the background. When set to None, the background depends on the PERSPECTIVE variable:

- **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.

Uwaga: 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

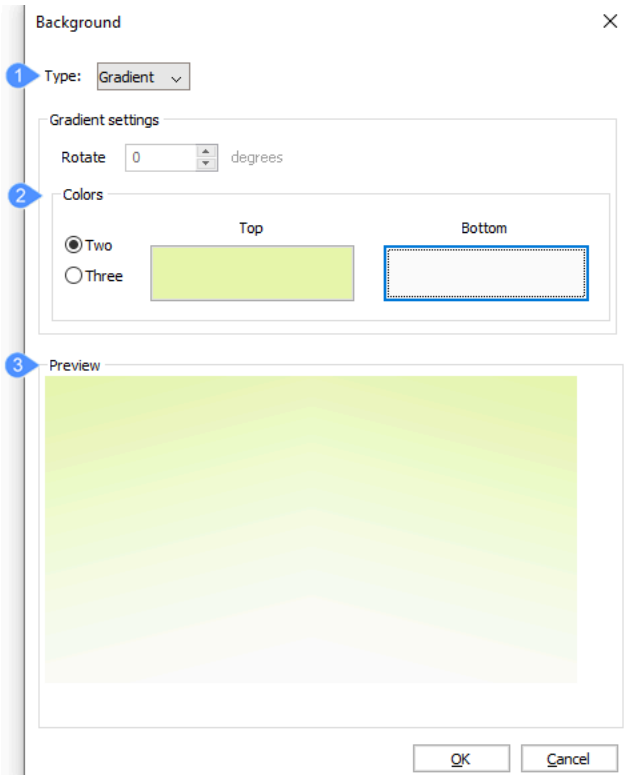
Type
Specifies the background type for the current viewport. When the background type is Solid, a single color is applied to the current viewport.

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.

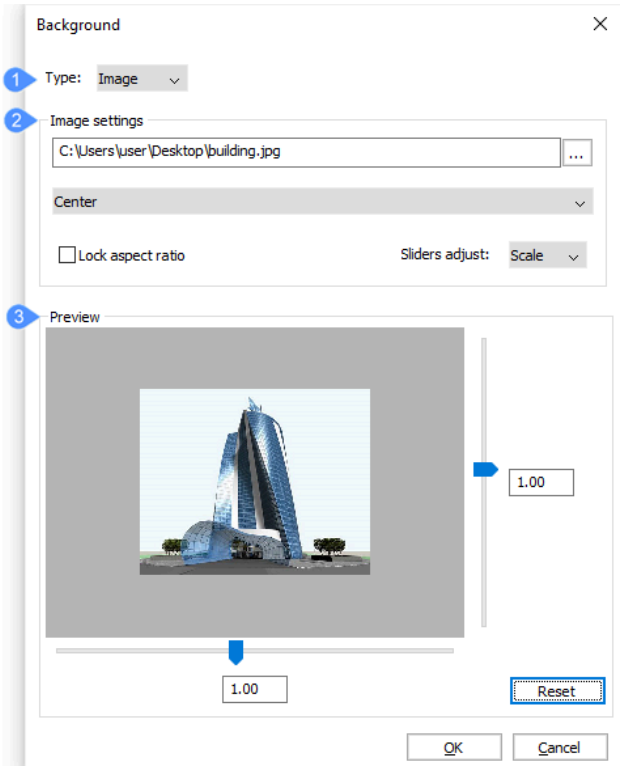
Two
Applies a two-color gradient to the background. When you select this option, the dialog box displays two color swatches. One is for the top color and one is for the bottom color. You can click on the color swatch to display the Select Color dialog box and specify a different color.

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.

> ***Uwaga:*** 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:

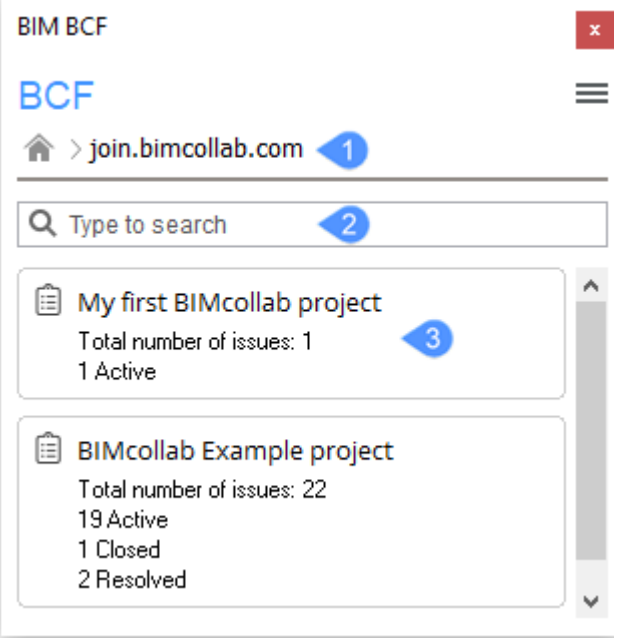
Cloud Service	Source URL
BIMcollab	<space name>.bimcollab.com *
BimSync	bcf.bimsync.com
BIMtrack	Bcfrestapi.bimtrackapp.co

Uwaga: * 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

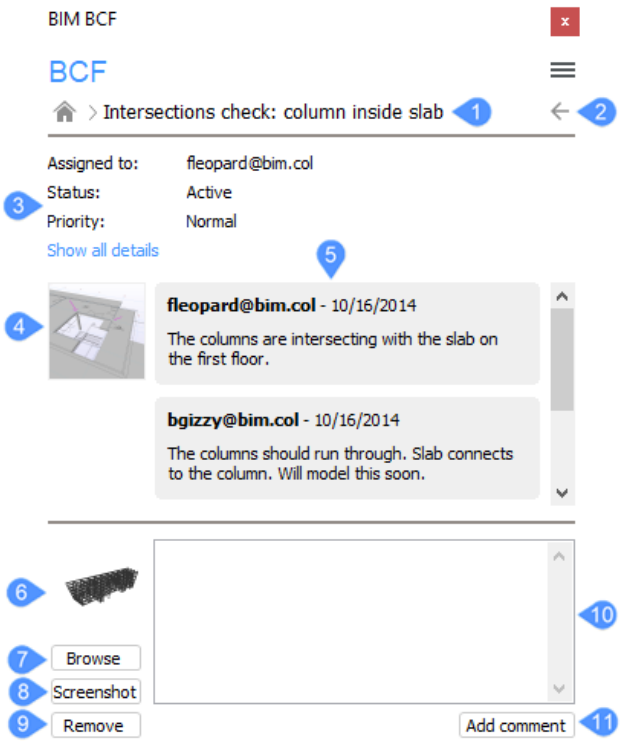
Source URL
Displays the source URL.

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).

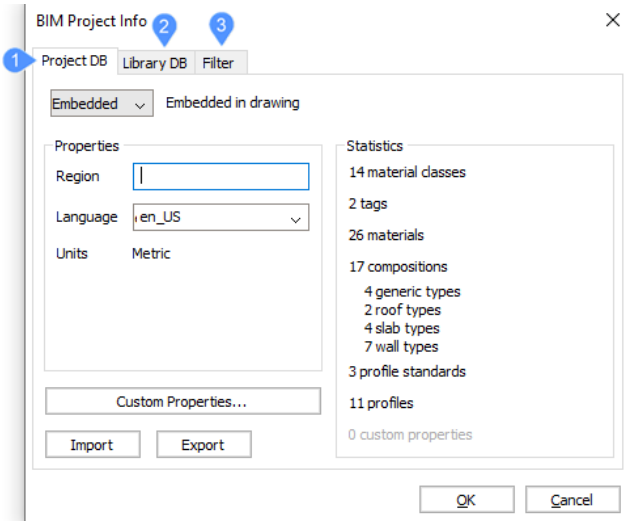
Remove
Removes the screenshot preview.

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.



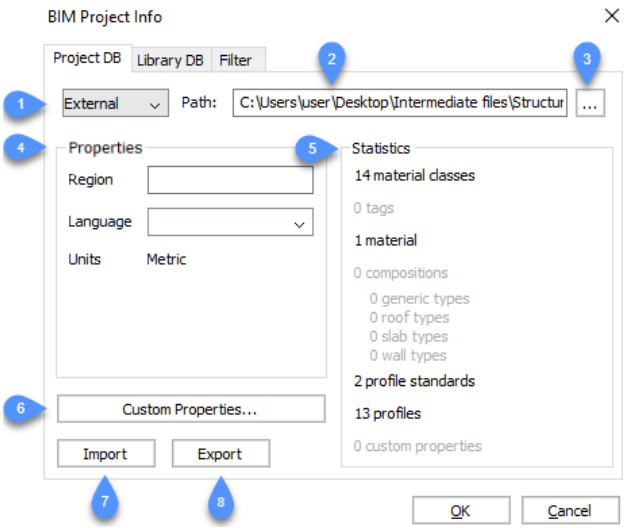
1 Project DB

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).

Uwaga: 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).

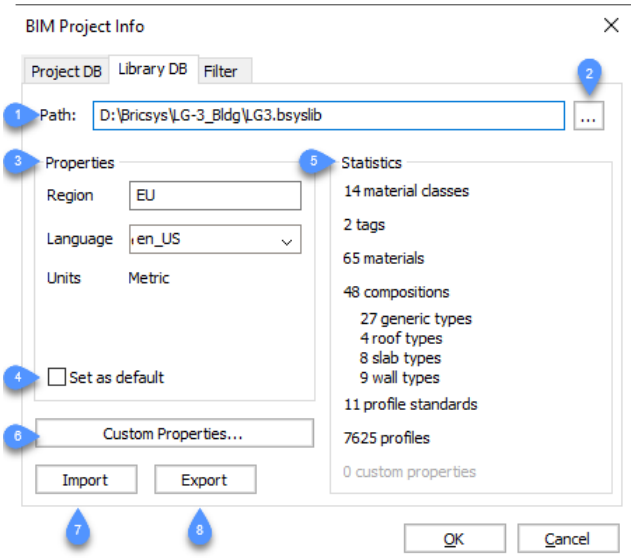
Statistics
Shows the location, properties, and the content of the current project library of the BIM model.

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.

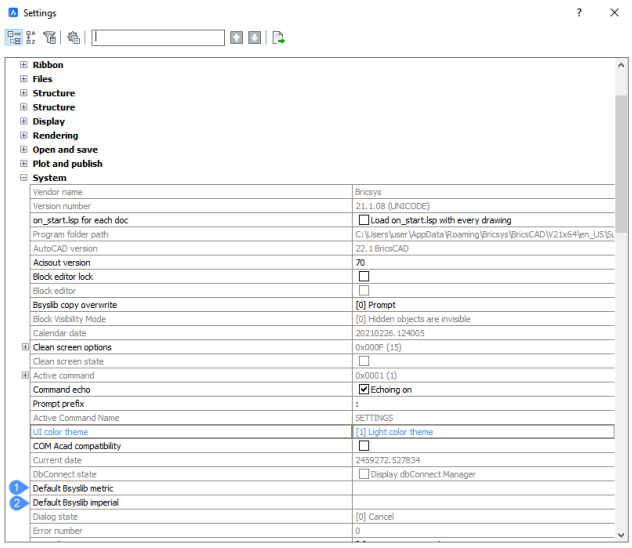
Region
Fill out the region field.

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.

Uwaga: 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.

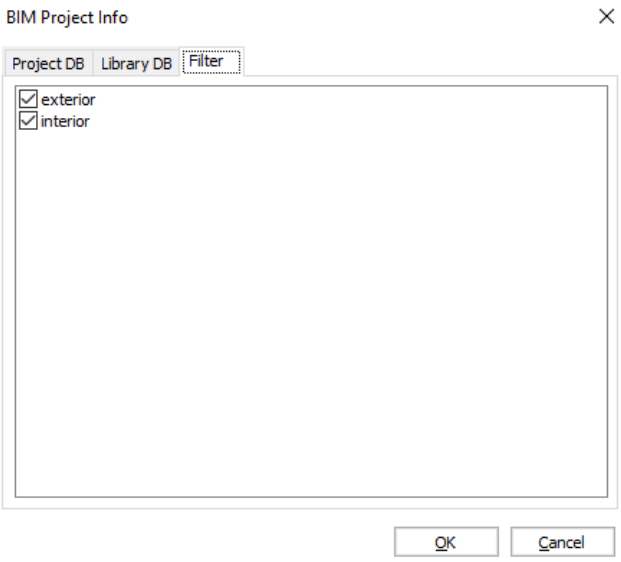
Custom Properties…
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

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.

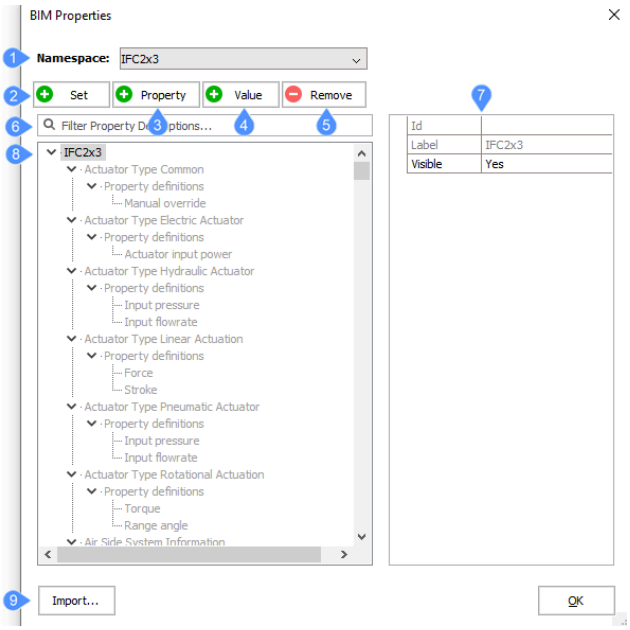
4.12.18.3 Filter

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

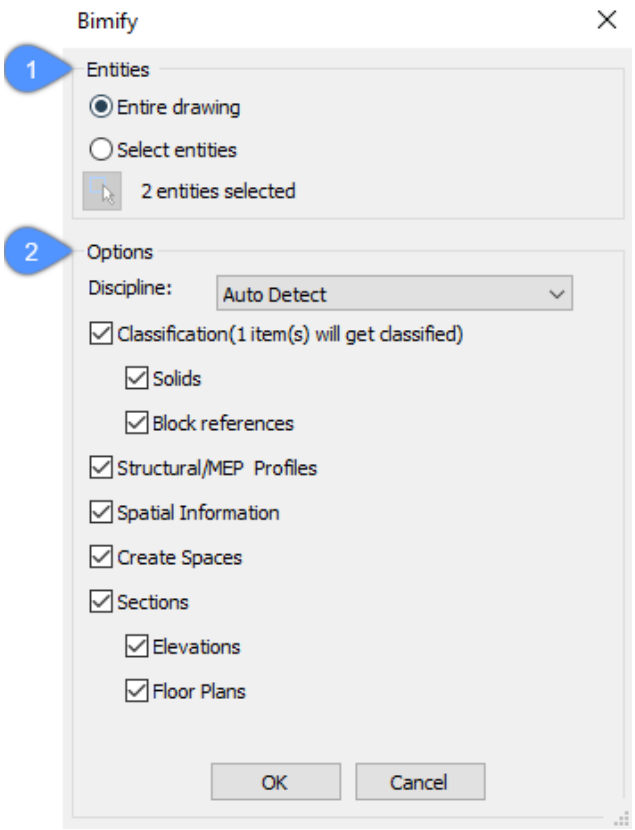
Displays the property sets, property definitions and value definitions.

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.

> ***Uwaga:*** 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).

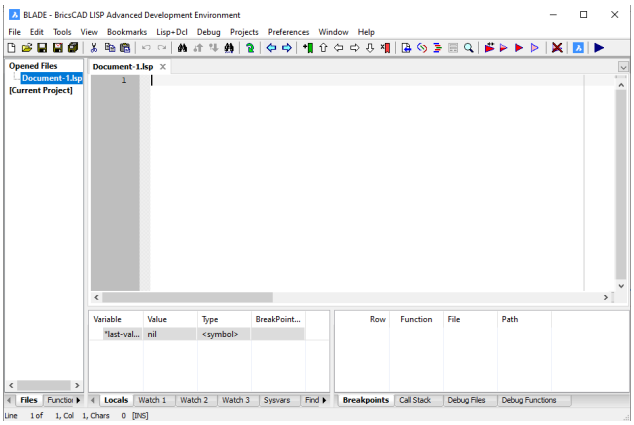
Create Spaces
Detects external walls and sets the property Wall Common/is External = ON.

Sections
Creates BIM Section entities.

- **Elevations**: creates 4 elevations (Front, Back, Left and Right).
- **Floor Plans**: creates a plan section for each floor.

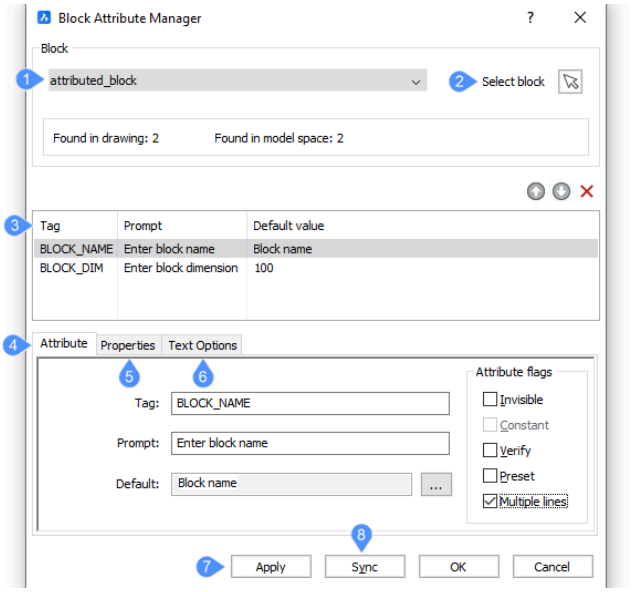
4.12.21 Blade - BricsCAD LISP advanced development environment dialog box

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

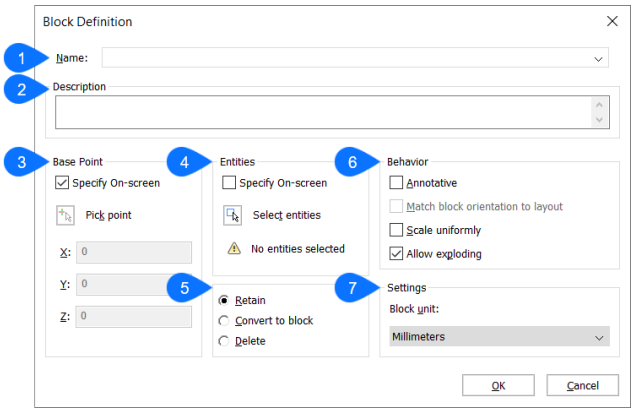
Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.

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.

Uwaga: 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.

Uwaga: 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.

Wskazówka: To edit the entities of an unexploded block, use the *BEDIT* command.

- **On** - block can be exploded after being inserted, with the Explode command

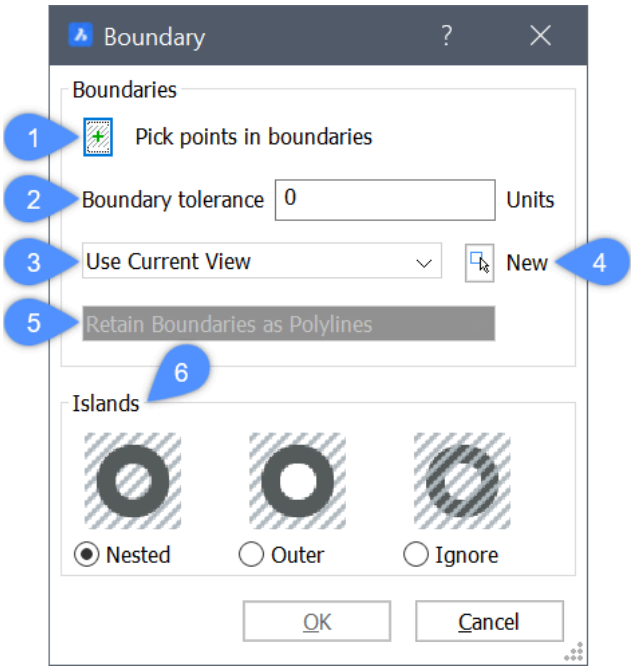
- **Off** - blocks cannot be exploded; you can change this property with the *EXPBLOCKS* command, Blocks section.

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.

Uwaga: 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.

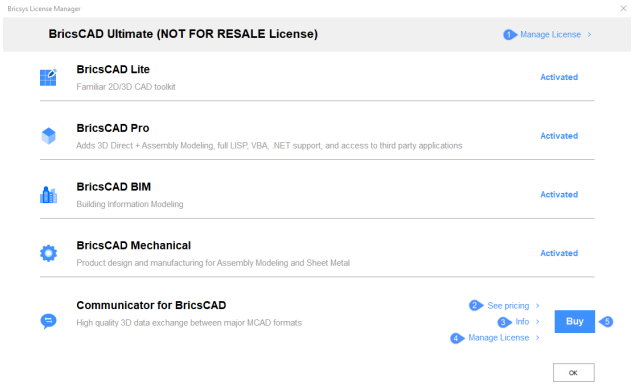
Nested
Treats every island as a boundary.

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

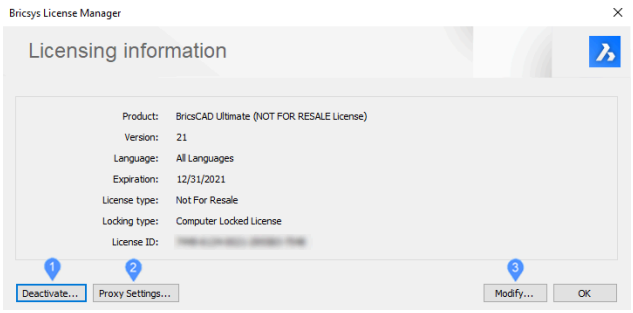
3 Info

4 Manage License for Communicator

5 Buy

4.12.25.1 Manage License

Opens the Bricsys License Manager dialog box.



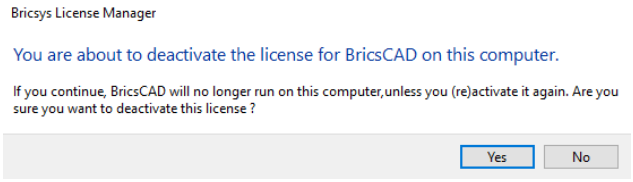
1 Deactivate

2 Proxy Settings...

3 Modify...

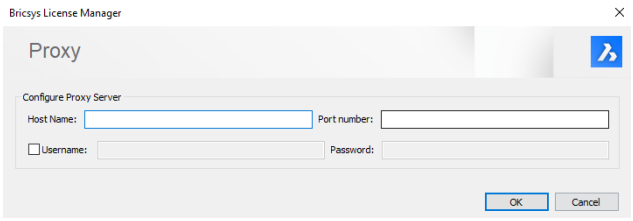
Deactivate

Opens the Bricsys License Manager dialog box for confirming deactivation.



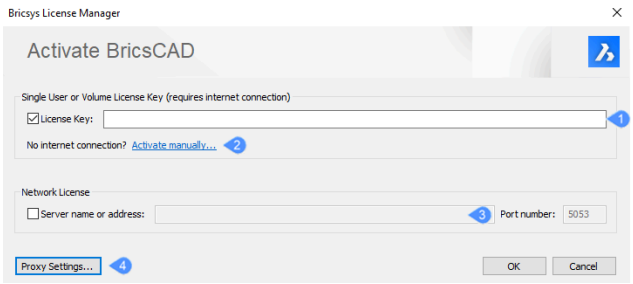
Proxy Settings...

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

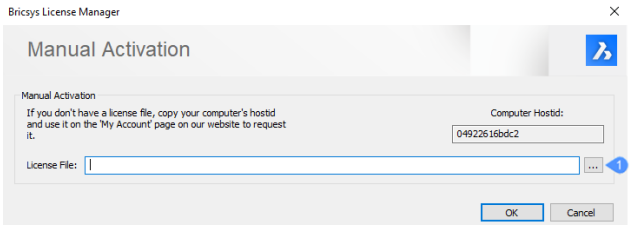
2 Activate manually...

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.

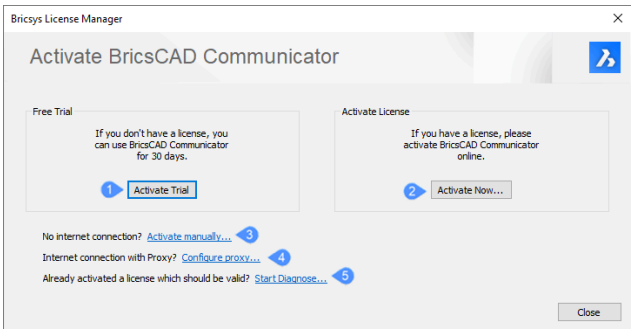
4.12.25.2 See pricing

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

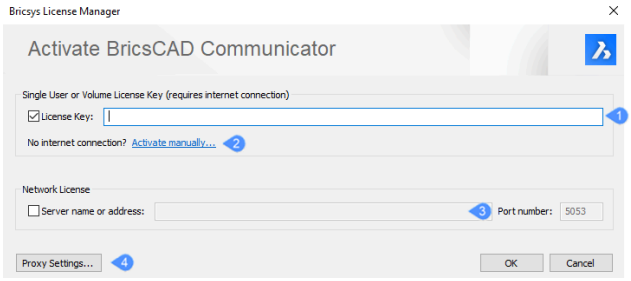
3 Activate manually...

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

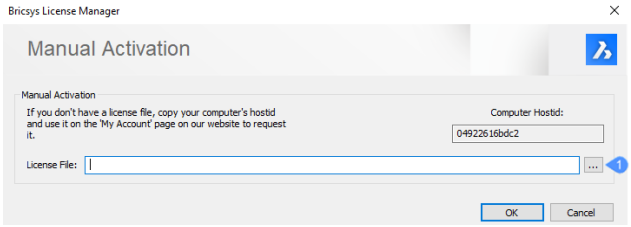
2 Activate manually

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



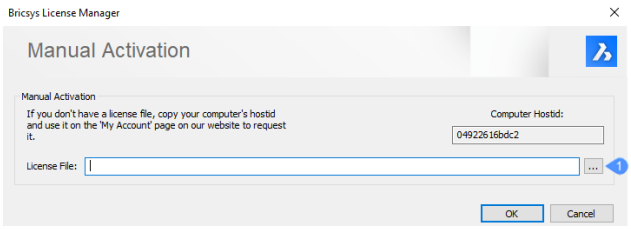
1 Select license file

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.



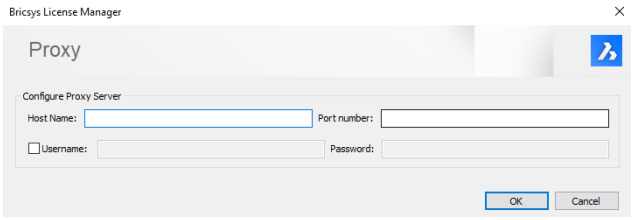
1 Select license file

Select license file

Opens the Select license file dialog box.

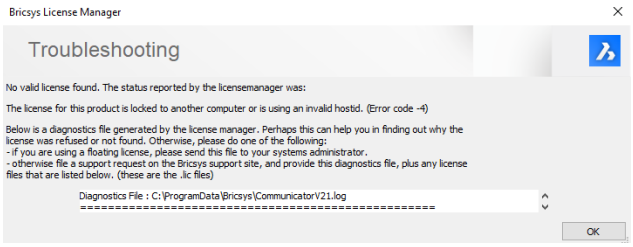
Configure proxy…

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.



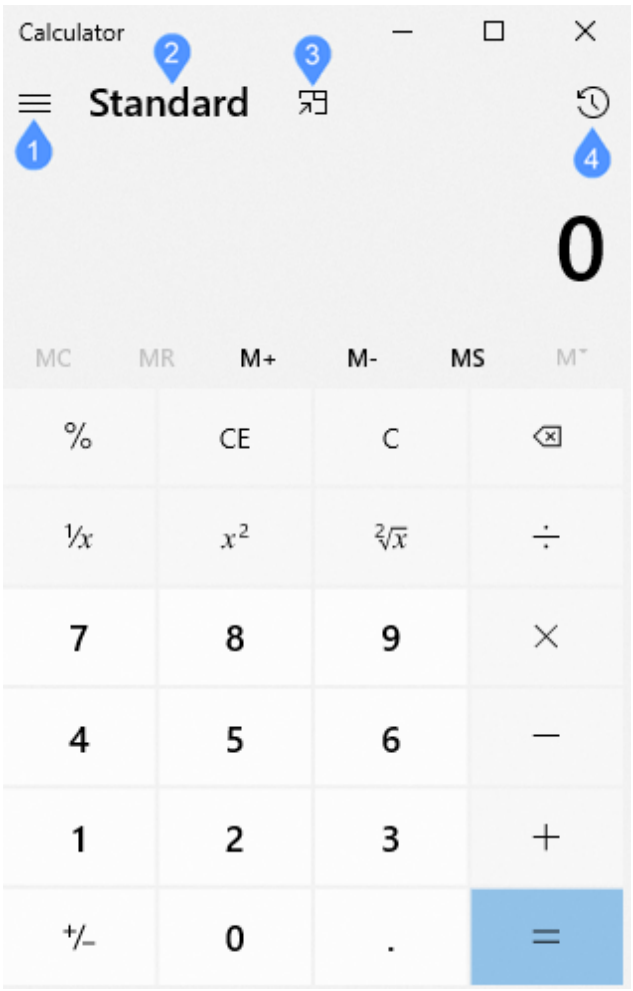
4.12.25.5 Buy

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

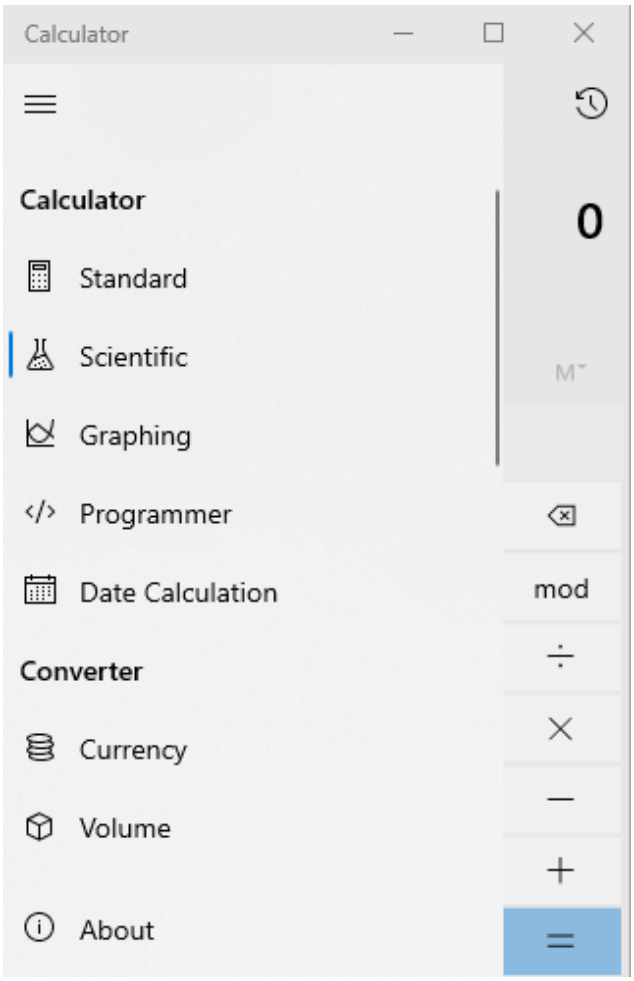
3 Keep on top

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.

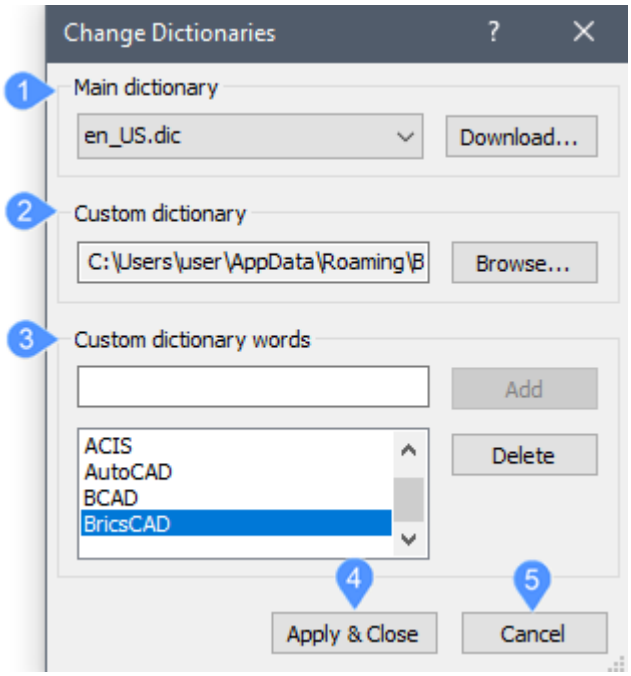
4.12.26.4 History

Displays the history of your calculations done in the current session.

Uwaga: 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

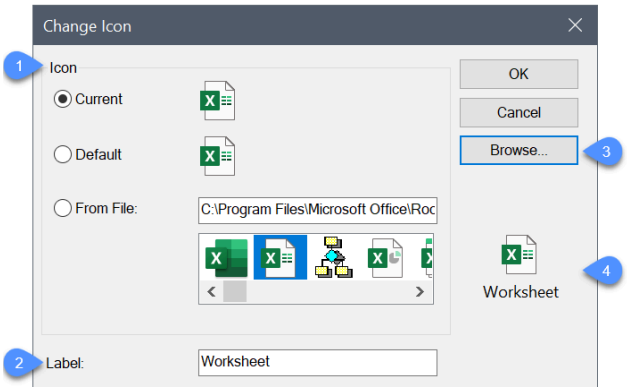
Apply your changes and close the dialog box. The custom dictionary file is only updated with added or deleted words after you choose Apply and Close.

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

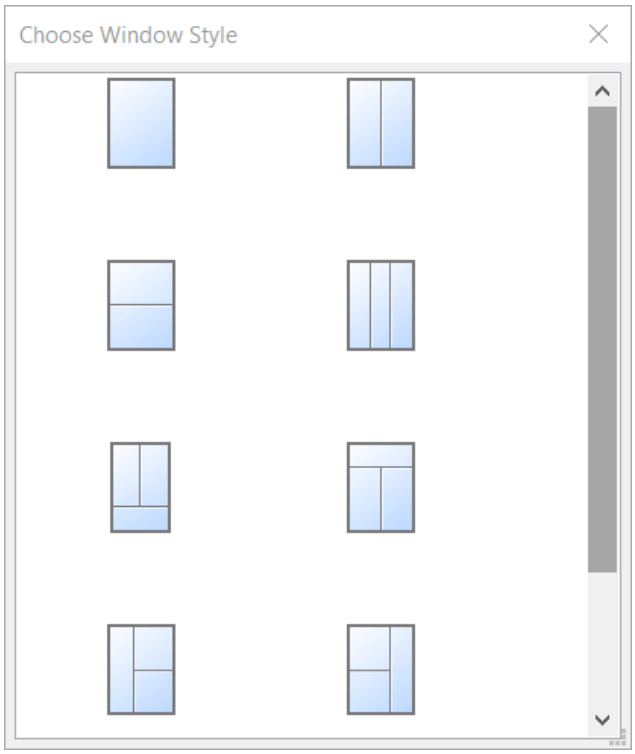
Displays the icon preview.

4.12.29 Choose window style dialog box

The Choose Window Style dialog box allows you to specify the style of a window.

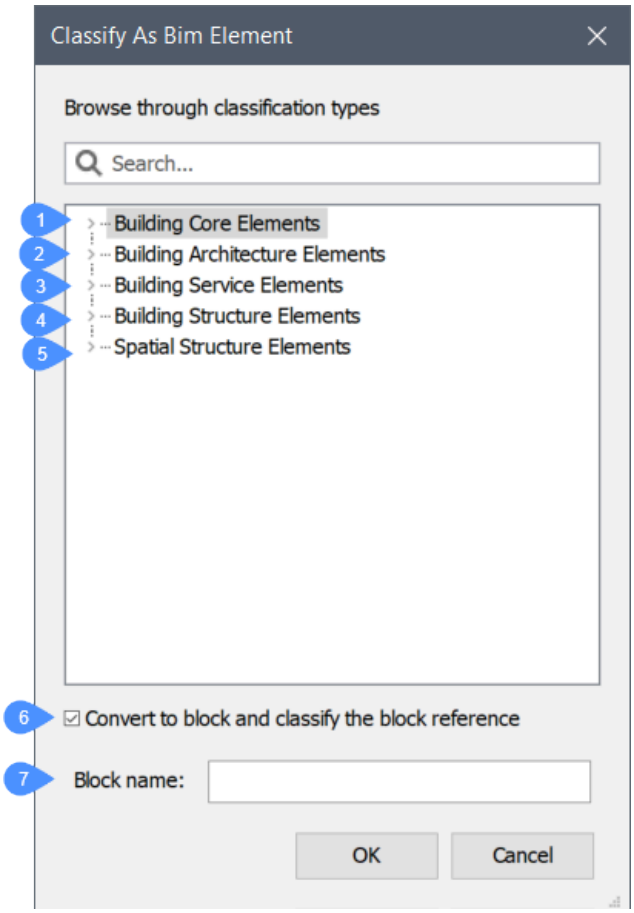
A series of parameters is created, which can be edited in the Parameters section of the Properties panel.

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

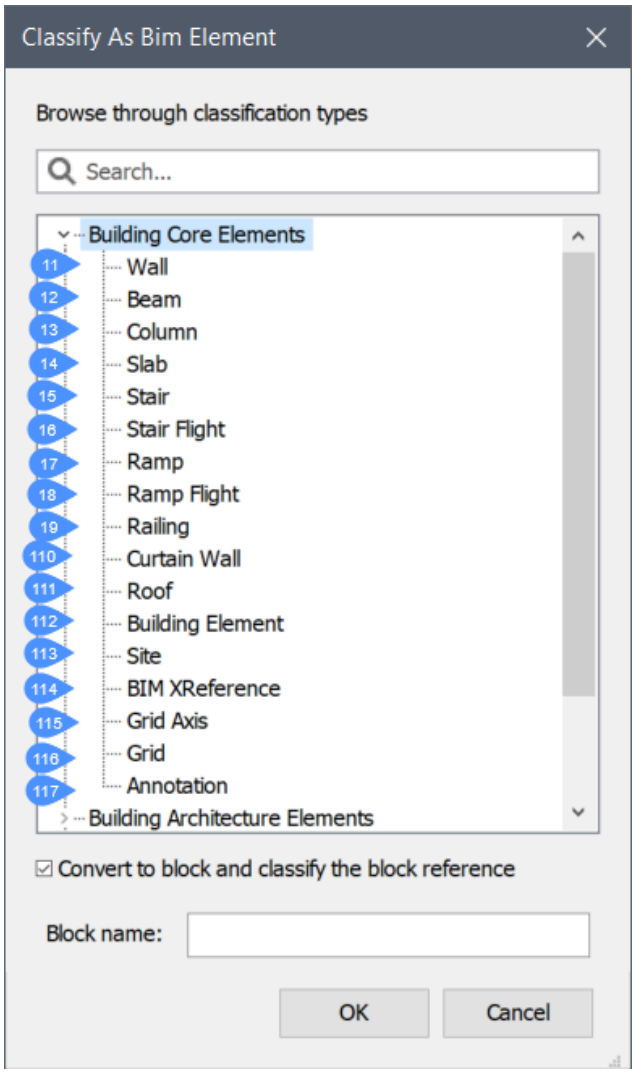
4 Building Structure Elements

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.

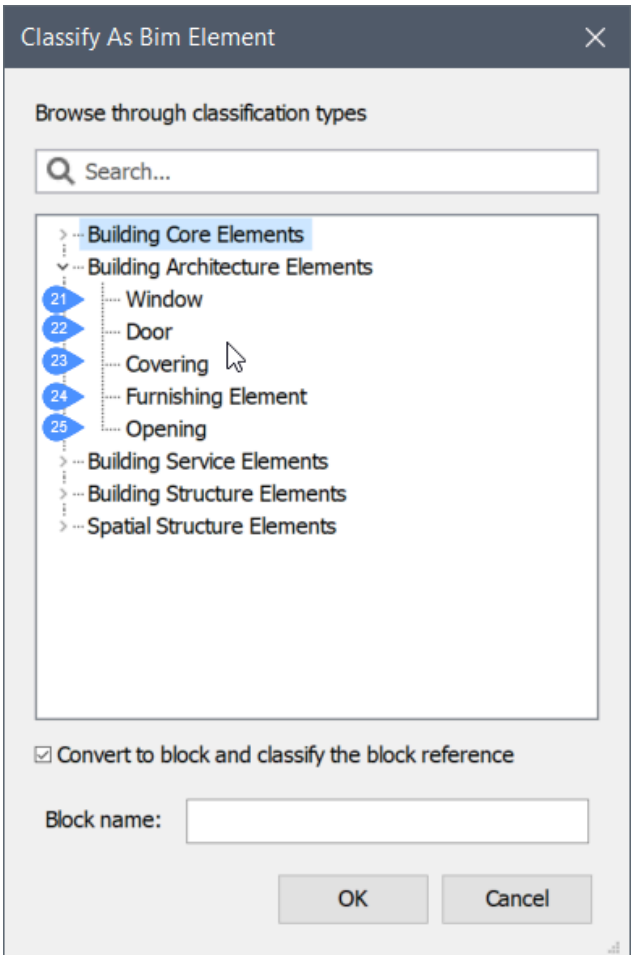
BIM XReference
The identification of information that is not explicitly represented in the current model or in the project database. Such information may be contained in classifications, documents or libraries within the external source.

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

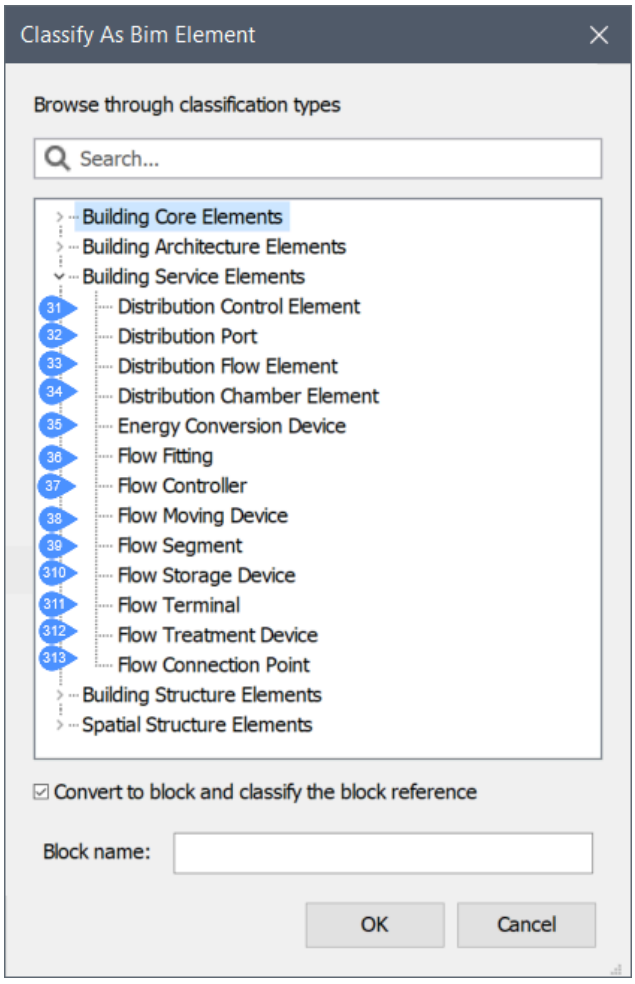
A covering is an element which covers some part of another element and is fully dependent on that other element.

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.

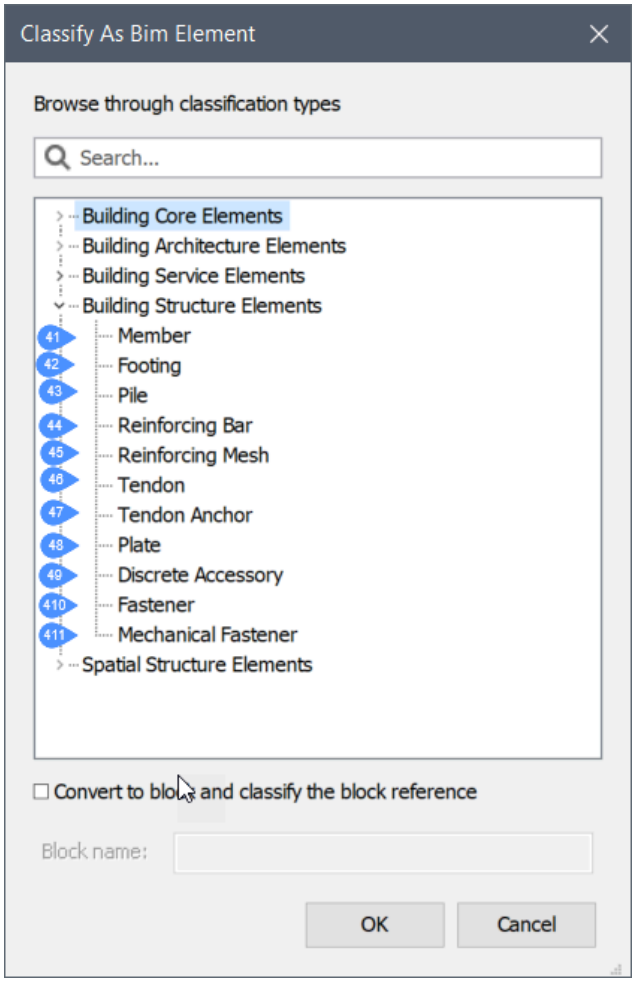
Flow Storage Device
Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow.
Example: tank, battery

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

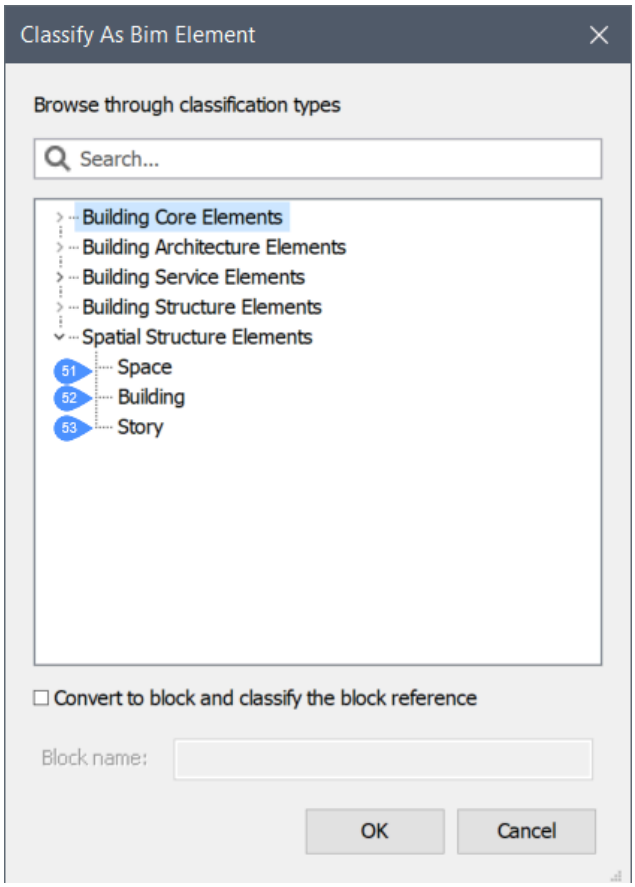
A Plate is a planar and often flat part with constant thickness. A plate can be a structural part carrying loads between or beyond points of support, however it is not required to be load bearing. The location of the plate (horizontal, vertical or sloped) is not relevant to its definition.

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

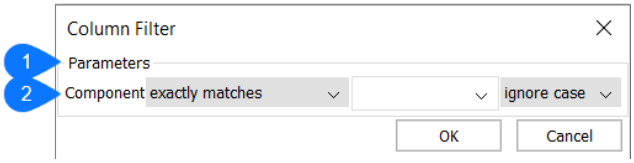
Space
BIM Space entities from enclosed boundaries (3D solids or linear 2D entities).

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.

Uwaga: The list of possible filters depends on the column content type.

Uwaga: 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.

Uwaga: Appears with numerical columns.

Not equal to
Value should be not equal to the target one.

Uwaga: Appears with numerical columns.

Less than
Value should be less than the target one.

Uwaga: Appears with numerical columns.

Greater than
Value should be greater than the target one.

Uwaga: Appears with numerical columns.

Less than or equal to
Value should be less than or equal to the target one.

Uwaga: Appears with numerical columns.

Between
Value should be in the given range, both ends excluded.

Uwaga: Appears with numerical columns.

Not in range
Value should not be in the given range, both ends excluded.

Uwaga: Appears with numerical columns.

Exactly matches
The content of the column should exactly match the target one.

Uwaga: Appears with columns containing text.

Does not match
The content of the column should be different from the target one.

Uwaga: Appears with columns containing text.

Starts with
The content of the column should start with the given text.

Uwaga: Appears with columns containing text.

Does not start with
The content of the column should not start with the given text.

Uwaga: Appears with columns containing text.

Ends with
The content of the column should end with the given text.

Uwaga: Appears with columns containing text.

Does not end with
The content of the column should not end with the given text.

Uwaga: Appears with columns containing text.

Contains
The content of the column should contain the given text.

Uwaga: Appears with columns containing text.

Does not contain
The content of the column should not contain the given text.

Uwaga: Appears with columns containing text.

Matches pattern
The content of the column should match the given wildcard template.

Uwaga: Appears with columns containing text.

Does not match pattern
The content of the column should not match the given wildcard template.

Uwaga: Appears with columns containing text.

Included in
The content of the column should be in the given set of values.

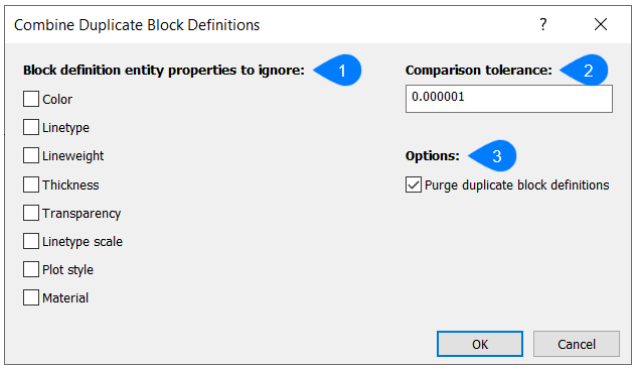
Uwaga: Appears with columns containing text and numerical columns.

Not included in
The content of the column should not be in the given set of values.

Uwaga: 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

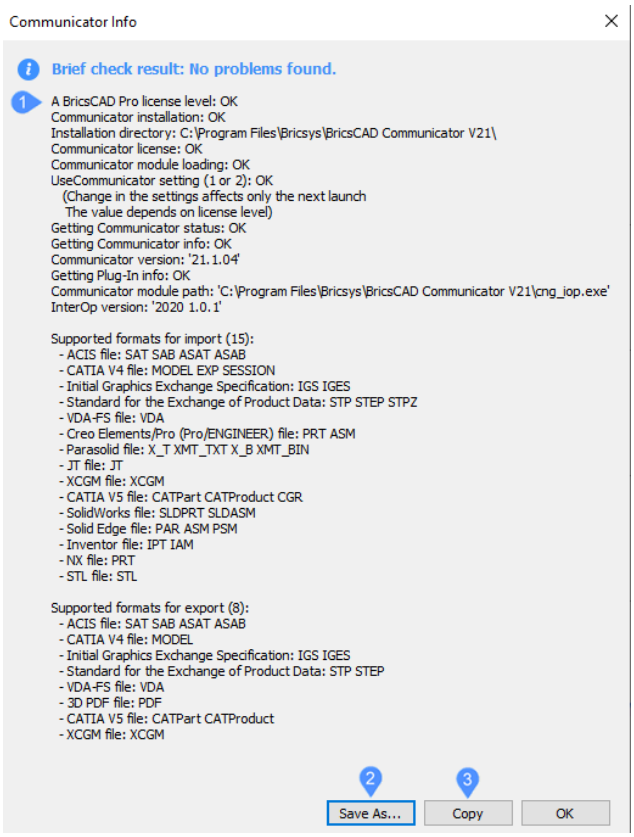
Purge duplicate block definitions

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.

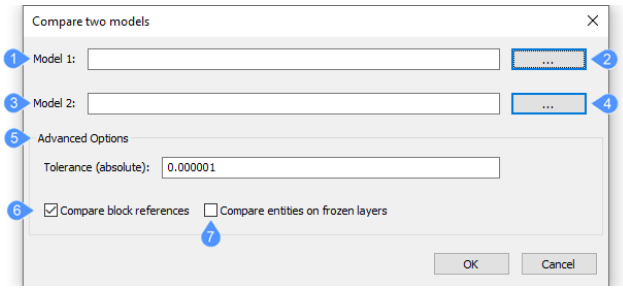
4.12.33.3 Copy

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

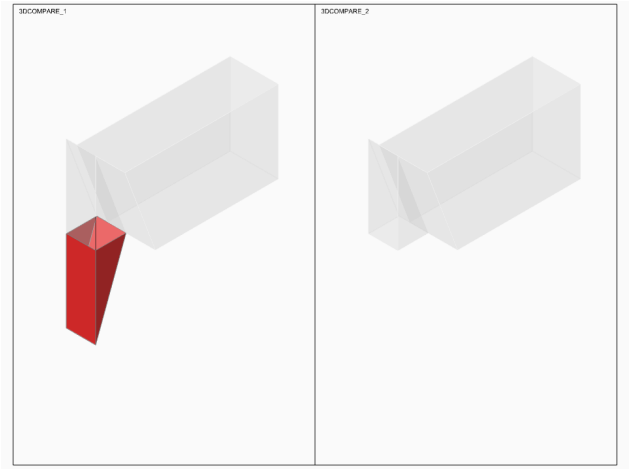
Determines if entities on frozen layers should be compared:

- **Yes** - solids and surfaces on frozen layers are compared.
- **No** - solids and surfaces on frozen layers are ignored.

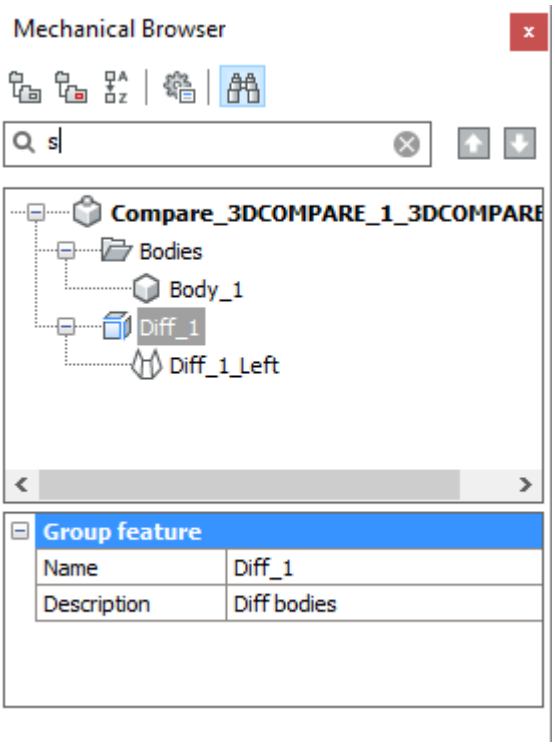
The value of the *3DCOMPAREMODE* system variable controls the display in the viewports of the Comparison layout; in the Settings dialog box, search for "3dcomparemode".

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.

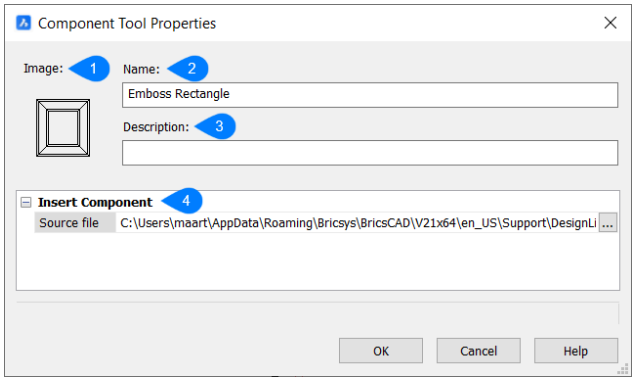


At the same time, the Mechanical Browser panel opens automatically and lists the differences between the 2 drawings.



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

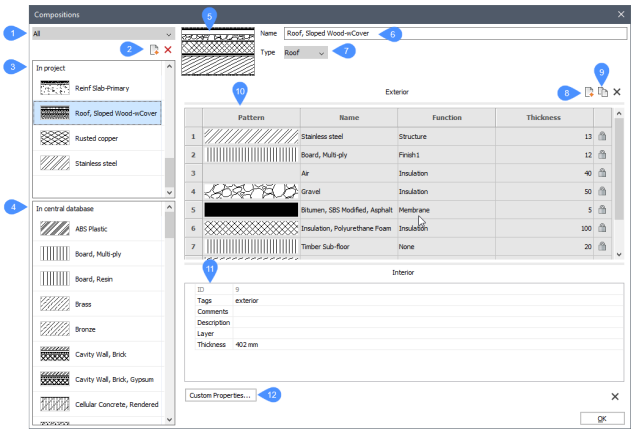
Provide an optional description of the component.

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.

Uwaga: 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

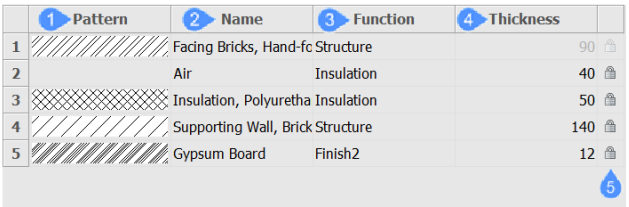
Inserts a copy of the selected ply.

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

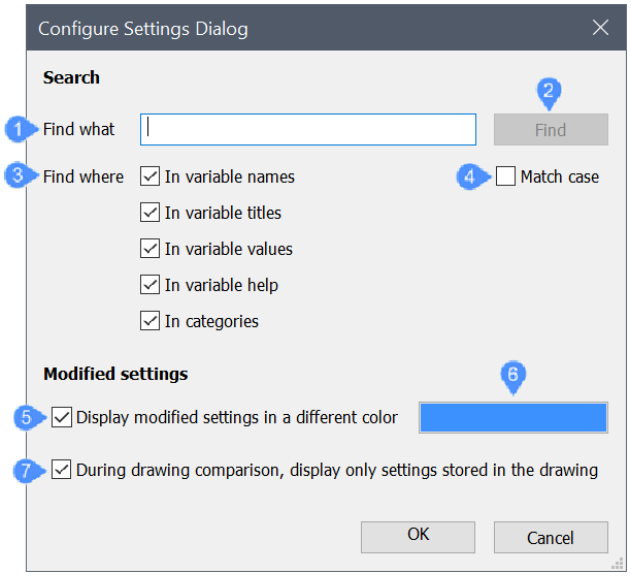
Thickness
Displays the overall thickness of a composition, which is equal to the sum of the thicknesses of the composing plies.

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

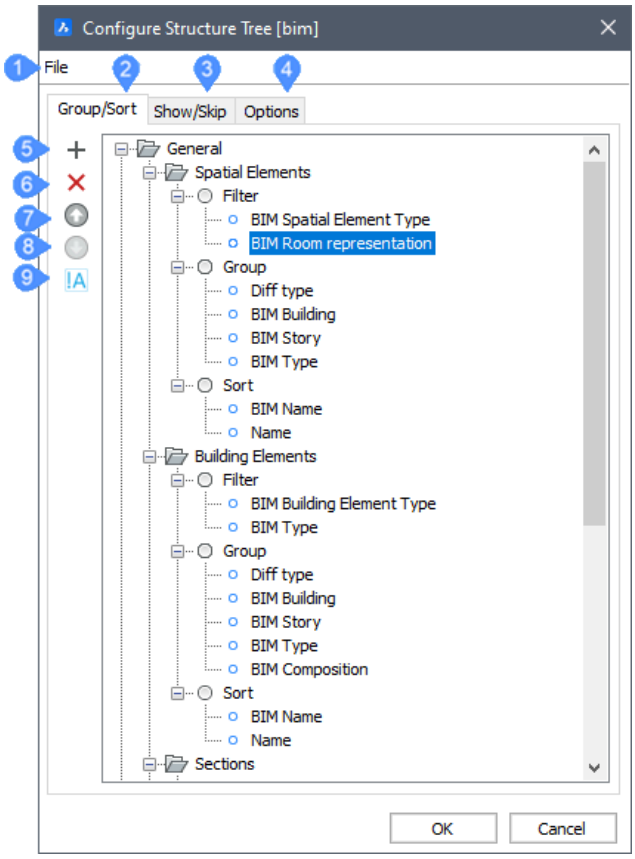
Click this button to select another color.

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.

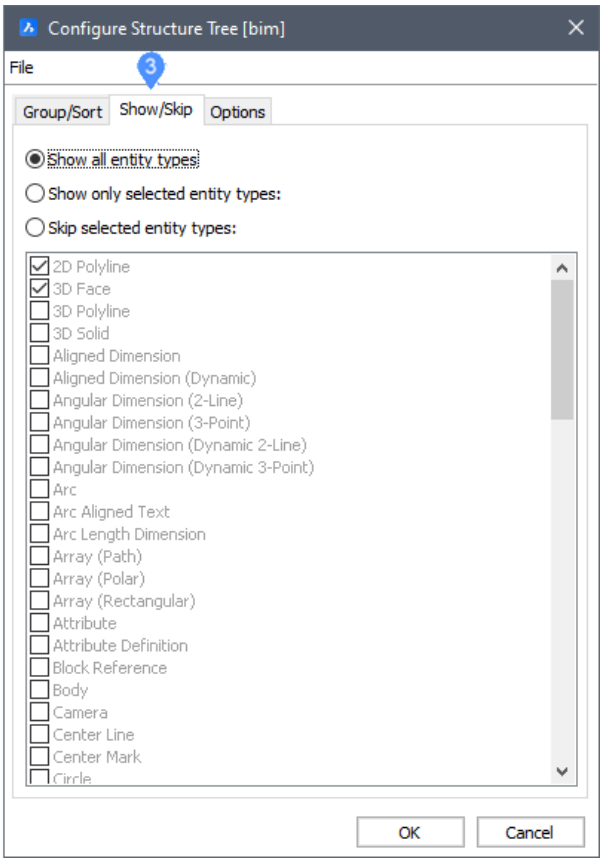
Delete (×) (6)
Deletes a selected rule or property.

Move Up (↑) (7)
Moves Up a selected rule or property.

Move Down (↓) (8)
Moves Down a selected rule or property.

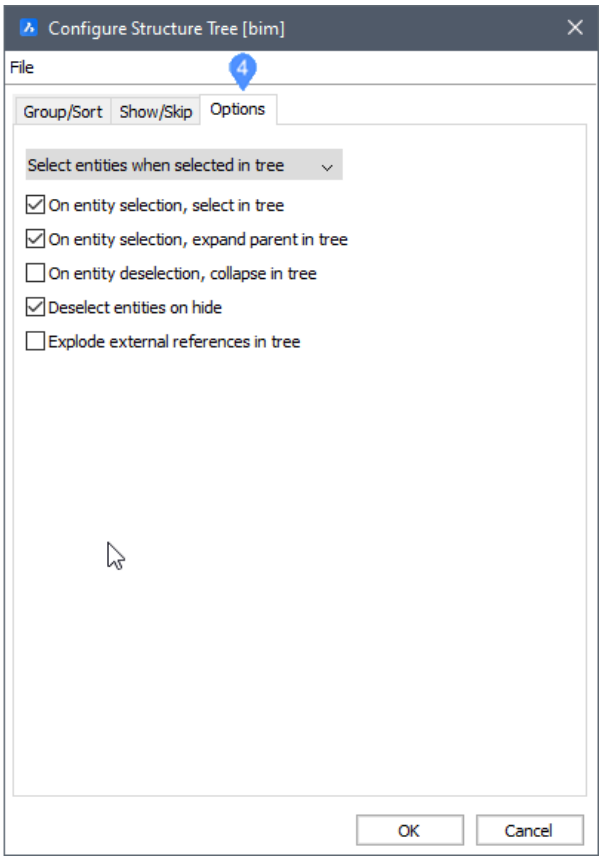
Toggle negation (9)
Toggle negation only for filter properties.

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

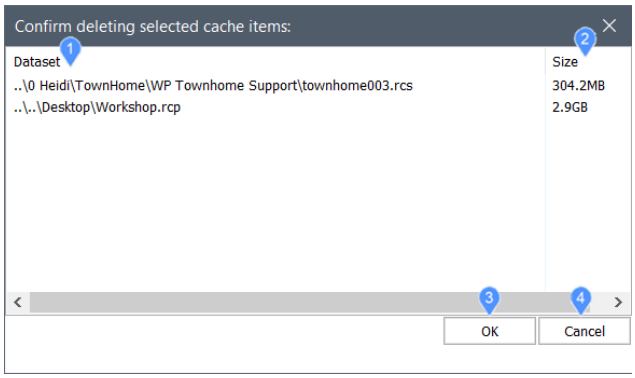


- Choose what happens in the model when entities are selected in the structure tree.
- Select what happens in the structure tree when an entity is selected/deselected in the model.

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

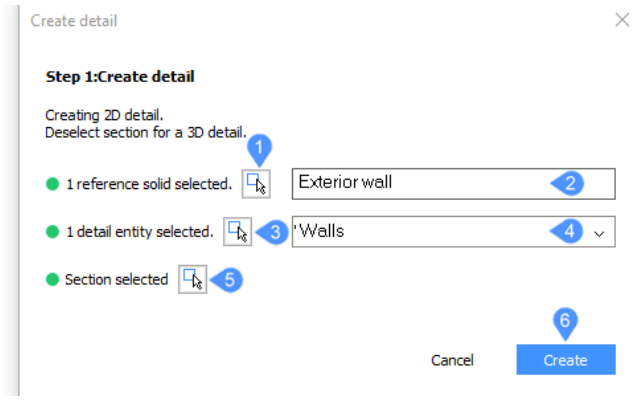
Deletes the point cloud files from the cache. Any drawings that reference these files will be unable to find them.

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*.

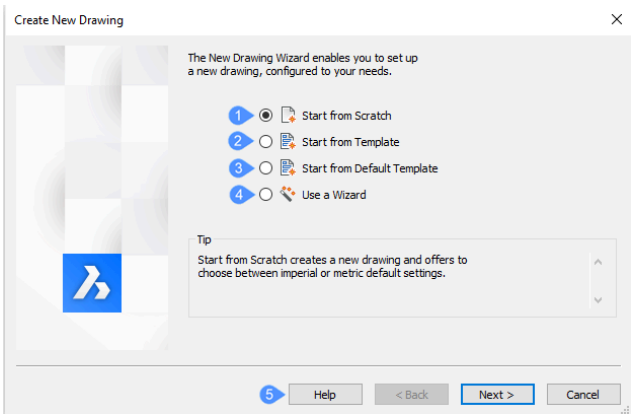
4.12.40.6 Create

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

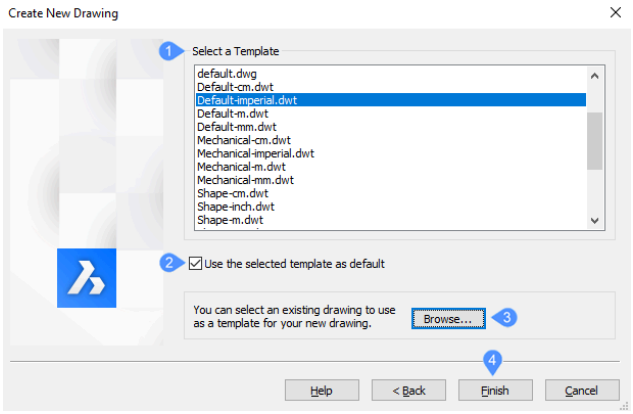
4.12.41.1 Start from Scratch

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

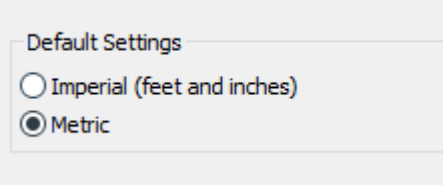
Starts new drawings with the default template defined by the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

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.

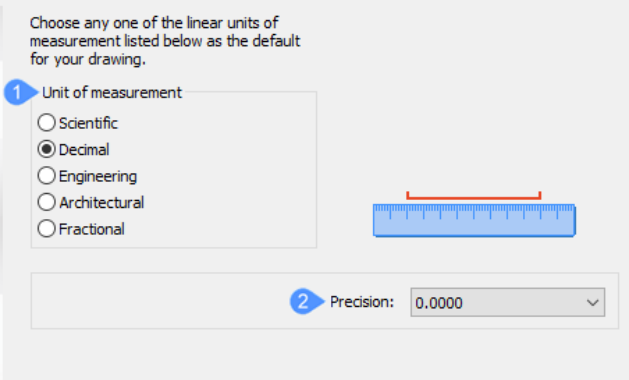
2 Choose a measurement system:



- 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

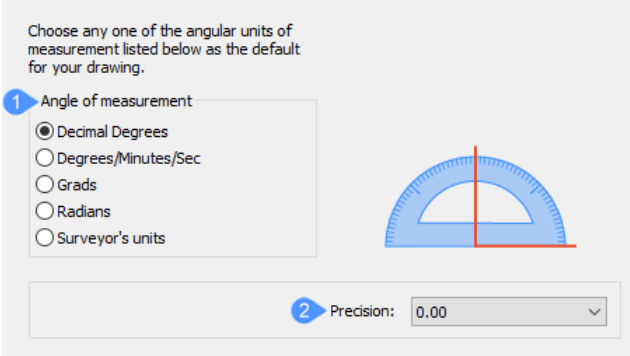
Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the Units command.

Uwaga: This is a display precision: the program makes calculations internally always to eight decimal places.

Uwaga:

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.

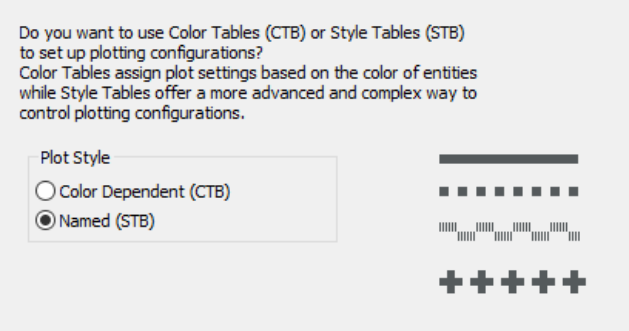
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds	Degrees,minutes,seconds,decimal secs	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units	North/South degrees towards East/West	N12d34'56"E

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:

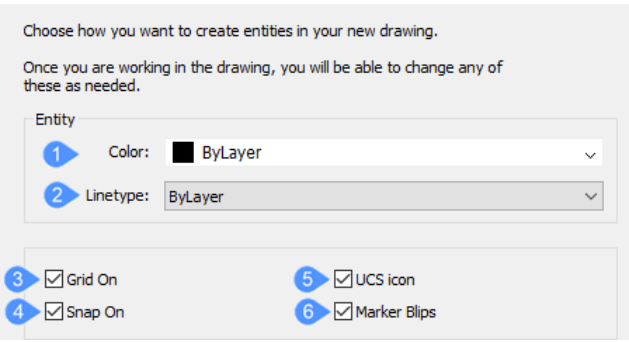


- **Color Dependent (CTB)** - these plot styles are simpler to employ, but less flexible.
- **Named (STB)** - plot styles are more flexible, but complex, because they control every aspect of every entity property during plotting.

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.

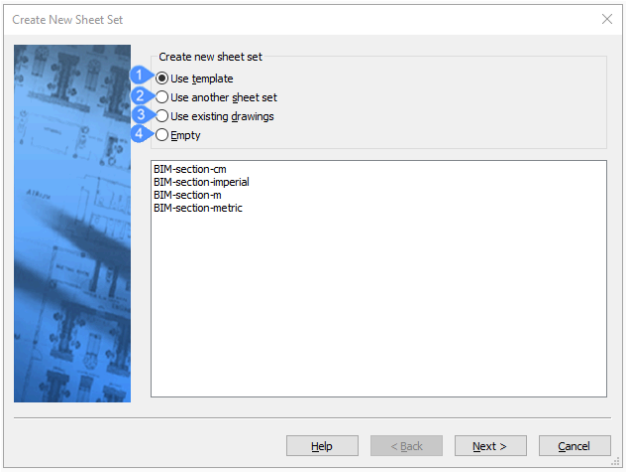
After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file and the options you choose during the wizard.

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.

Ważne: 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.

Uwaga: 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.

Ważne: 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.

Ważne: 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.

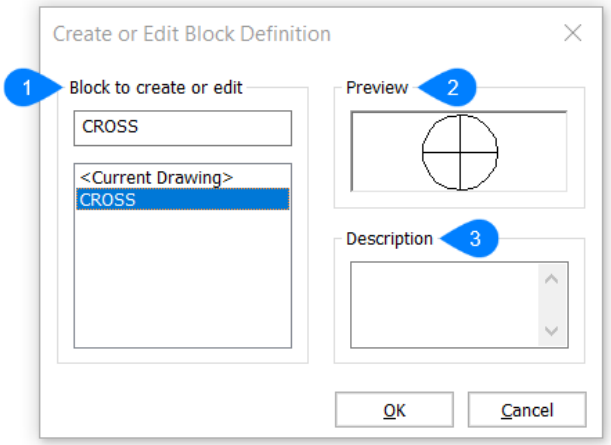
2 Click **Next**.

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.

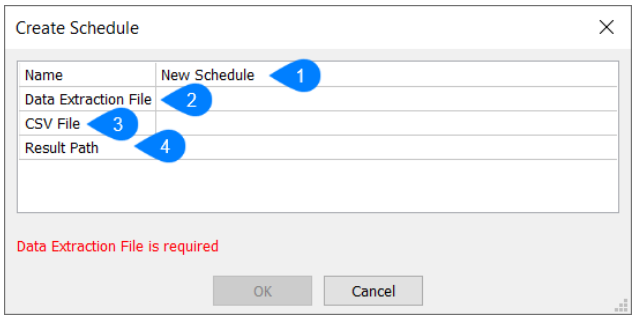
4.12.43.3 Description

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

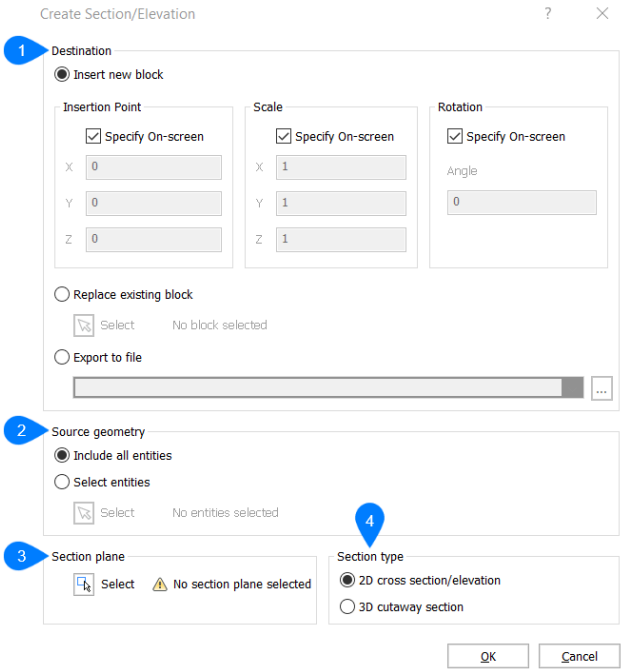
Defines the CSV file that will be used. Opens the Select Delimiter-separated format file dialog box where you can select a file.

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).

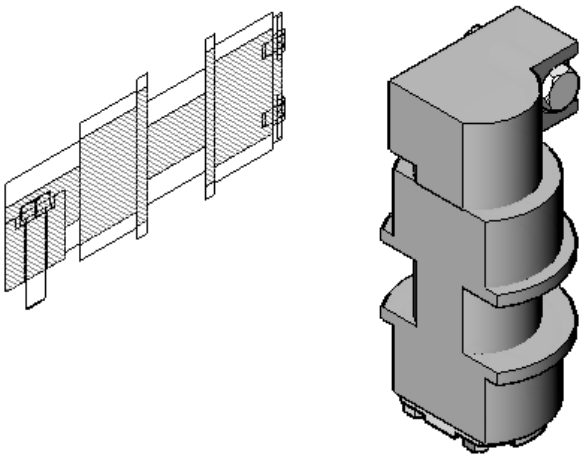
Select entities
Allows to select the entities to be included in the model space.

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.

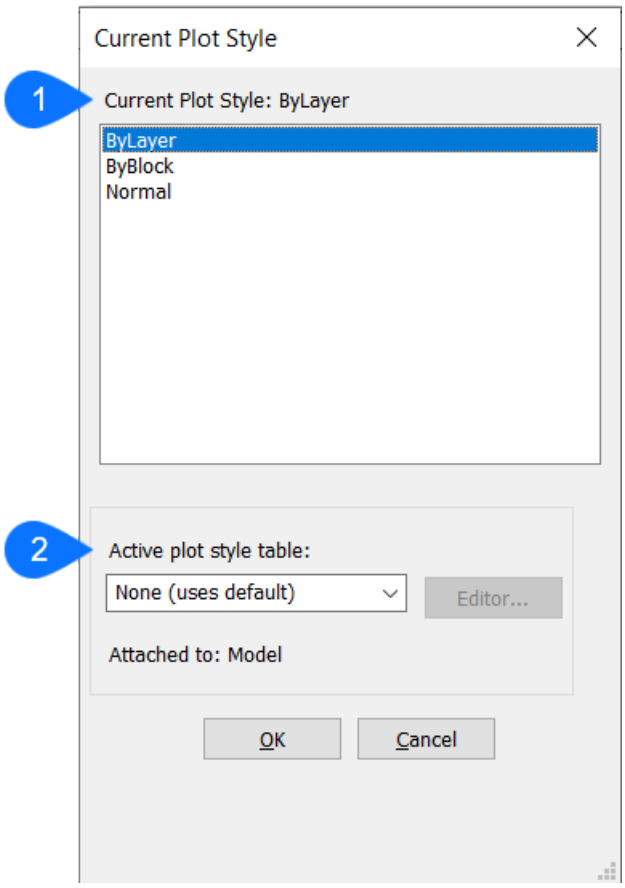


2D cross section/elevation
Uses the 2D section settings defined by the SectionPlaneSettings dialog box to create the block.

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

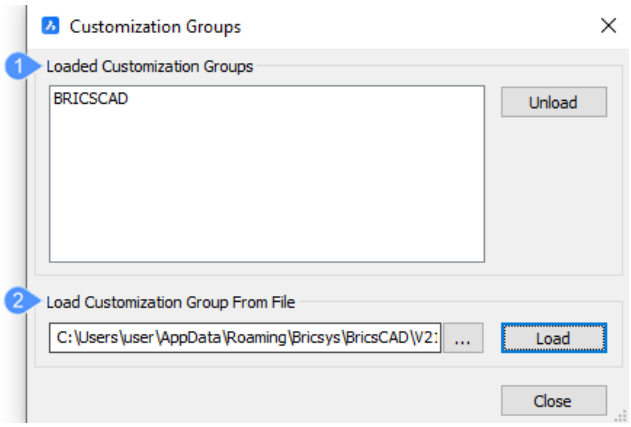
Lists the names of STB files (short for "style table"). To edit the chosen plot style table, click 'Editor...'. The Plot Style Table Editor dialog box will be displayed.

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.

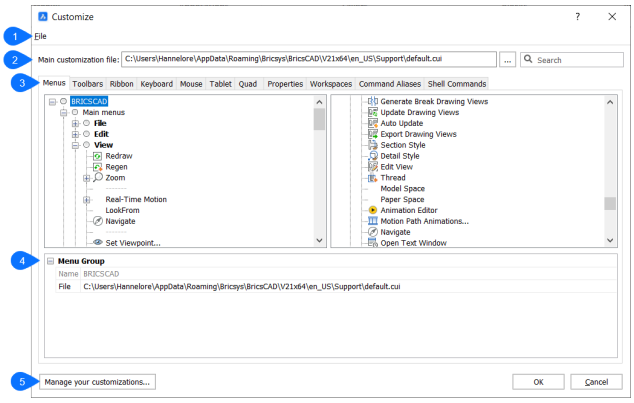
A CUI file can be unloaded by selecting it from the list and clicking the **Unload** button.

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

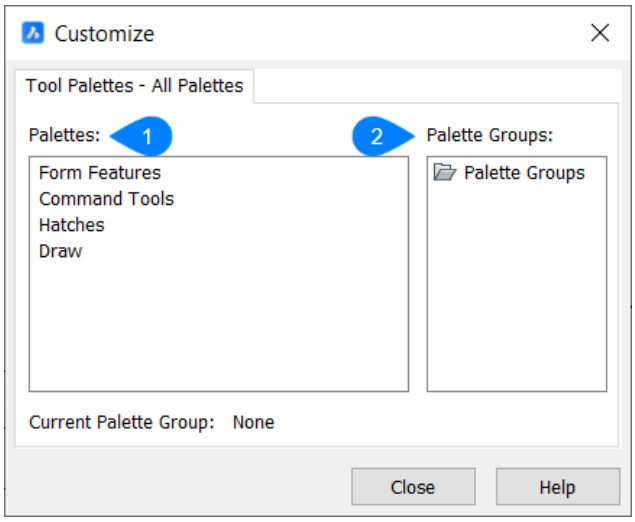
Delete ...
Deletes the menu, submenu, item or separator selected item. Gives a warning where you can confirm you to delete the selected item.

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.

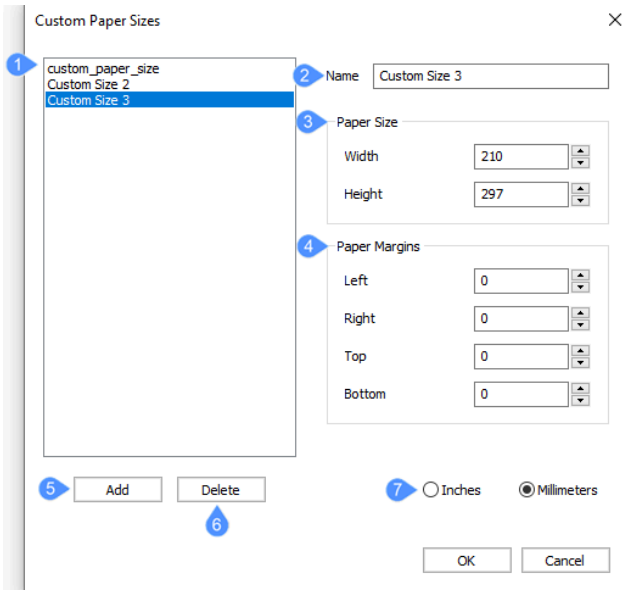
New group
Adds a new sub-group to the list with a generic name that can be changed right away.

> ***Uwaga:*** 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

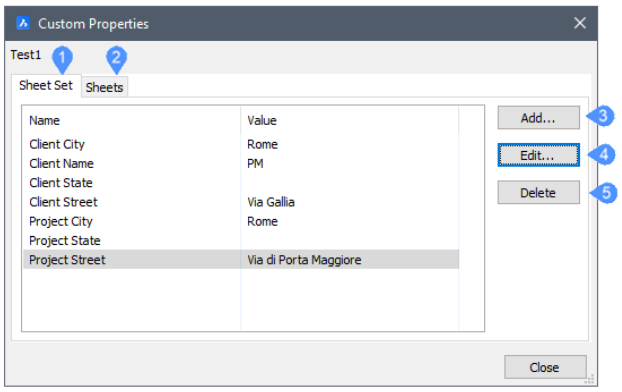
Allows you to select a custom paper size from the list and delete it. You can delete one item at a time.

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

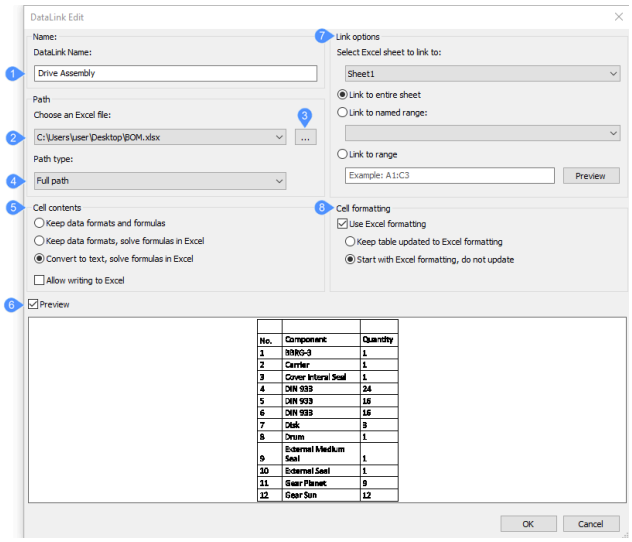
Allows to edit the value of a selected custom property. Displays the **Custom Sheet Property** or **Custom Sheet Set Property** dialog box in which you edit the name of the property.

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.

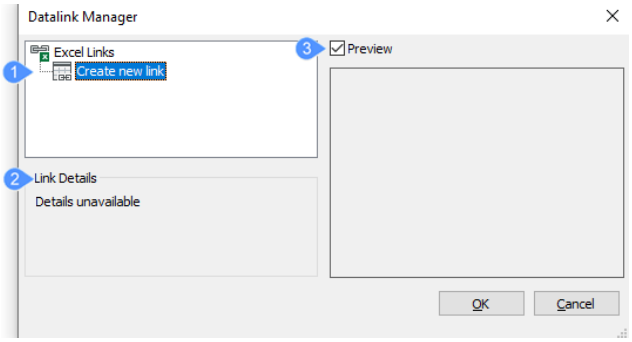
4.12.52.8 Cell formatting

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

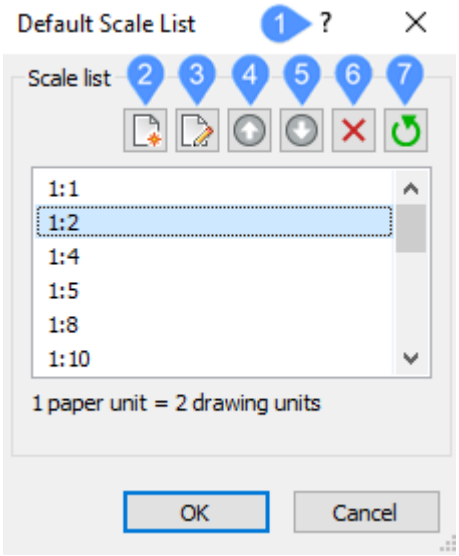
Displays link details.

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

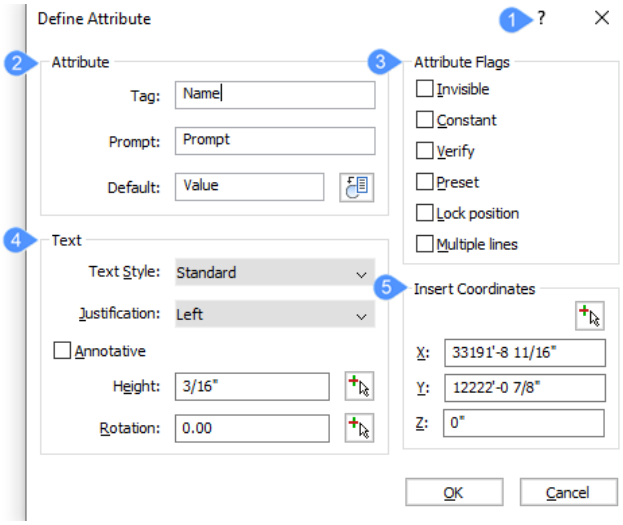
Deletes the selected scale factor(s). Press the Ctrl key to select and delete more than one factor at a time. Scale 1:1 cannot be deleted.

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.

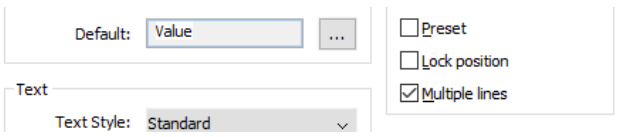
- **Preset**

  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**

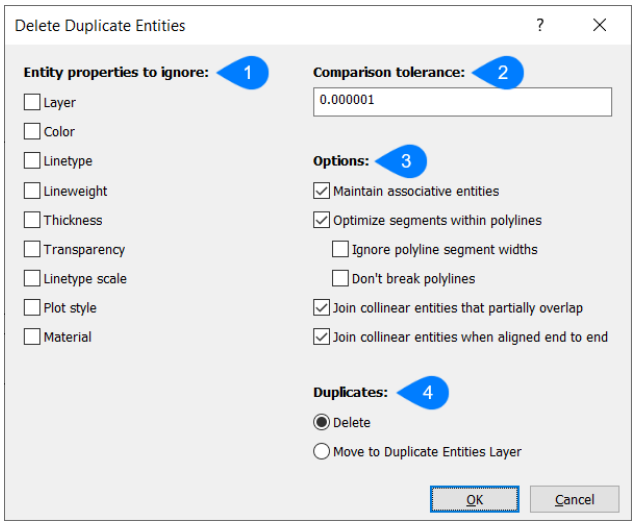
  Specify the insertion point directly in model.

- **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.

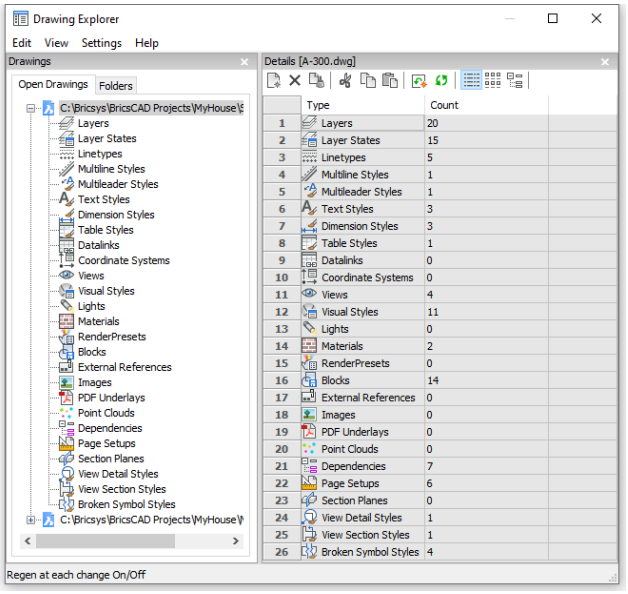
4.12.56.4 Duplicates

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.

Uwaga:

- 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.

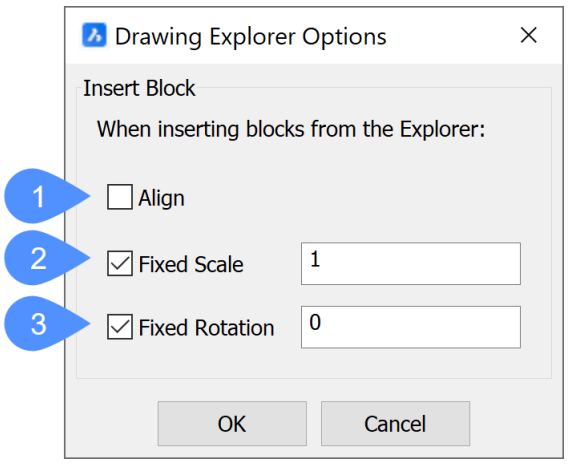
4.12.57.5 The Context menu

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

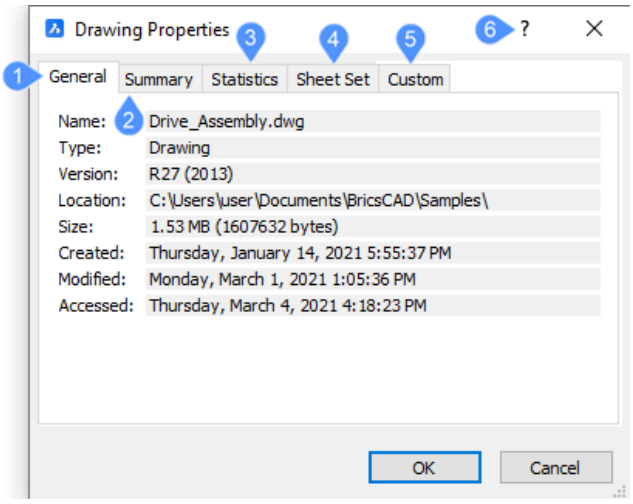
Tick the checkbox to specify the scale factor to be used for all block insertions.

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

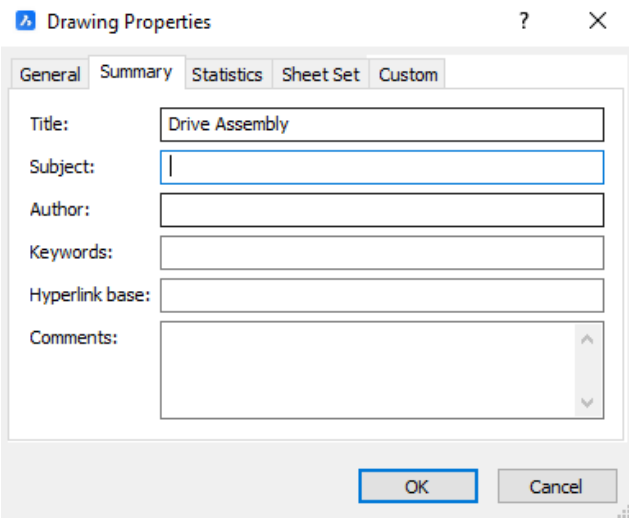
6 Command reference

4.12.59.1 General

Displays the general properties of the drawing, such as creation date and time.

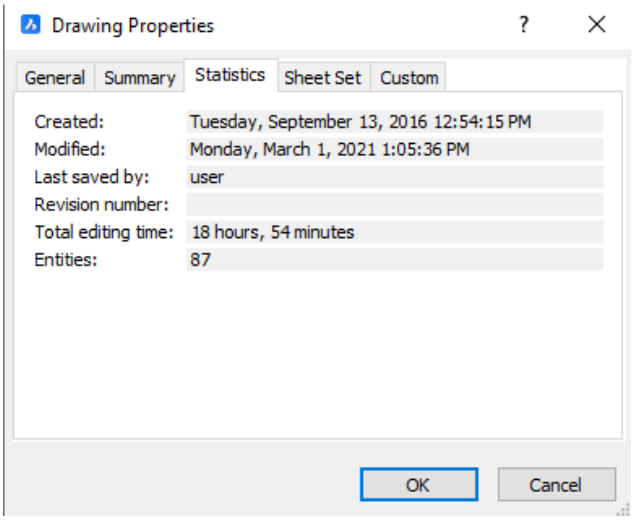
4.12.59.2 Summary

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.



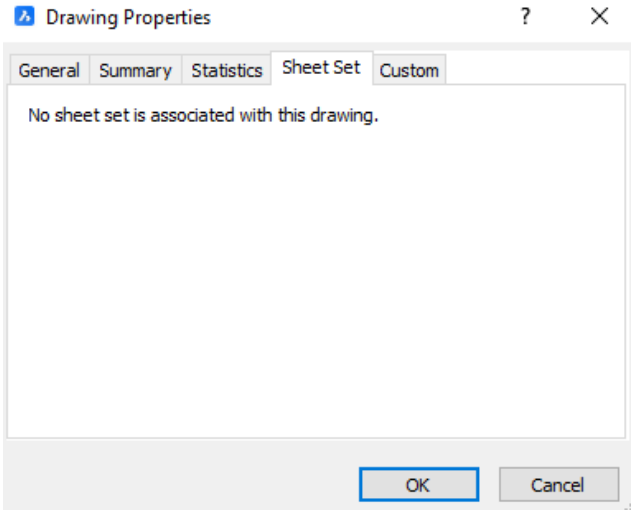
4.12.59.3 Statistics

Displays statistical information about the drawing, such as when it was created, modified.



4.12.59.4 Sheet Set

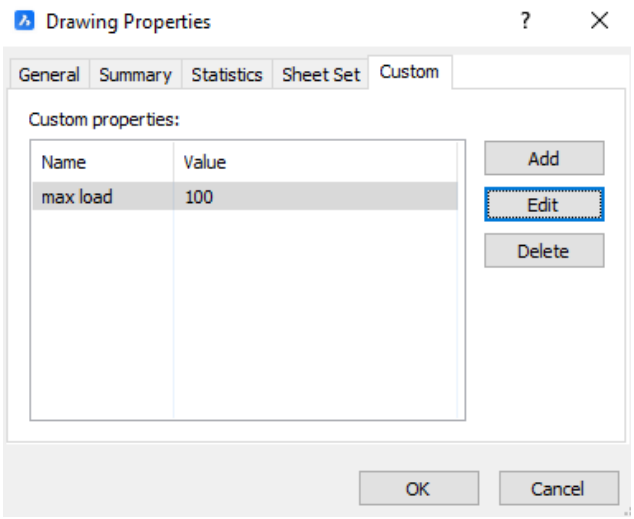
Displays sheet set data when the drawing is associated with a sheet set.



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.

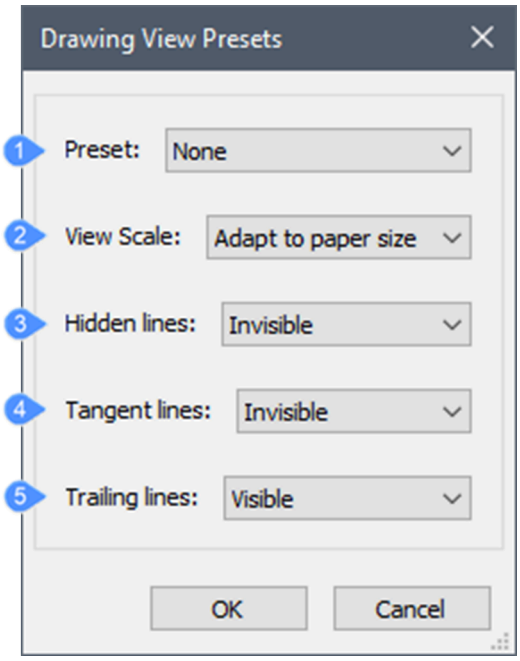


4.12.59.6 Command reference

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

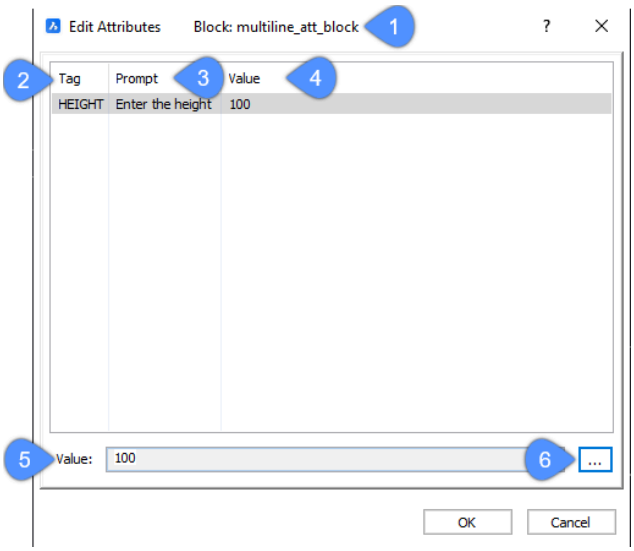
Sets the default for **Tangent Lines** in drawing views.

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.

Uwaga: 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.

Uwaga: 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.

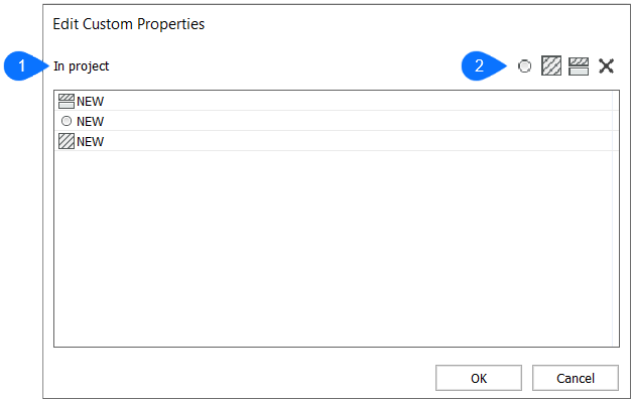
4.12.61.5 ... Text Formatting

Changes the value of the attribute text.

Uwaga: 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.

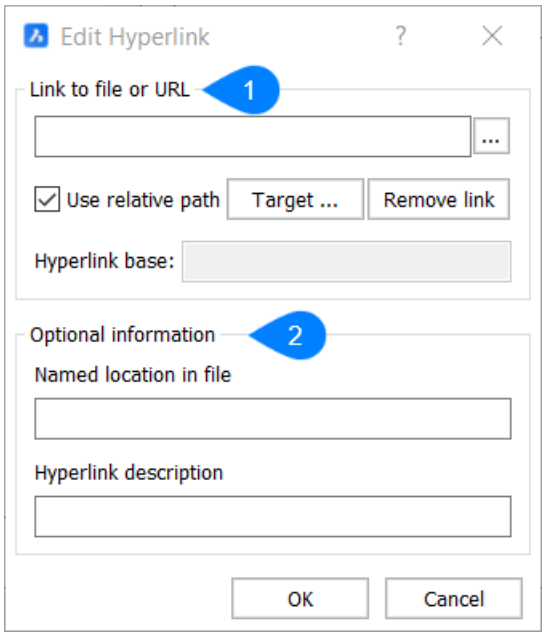
Add Material Custom Property
Creates a new material custom property, which can be assigned to materials in the project.

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.

Uwaga: To remove the link, press **Remove** link.

Hyperlink base
Displays the hyperlink base path.

4.12.63.2 Optional information

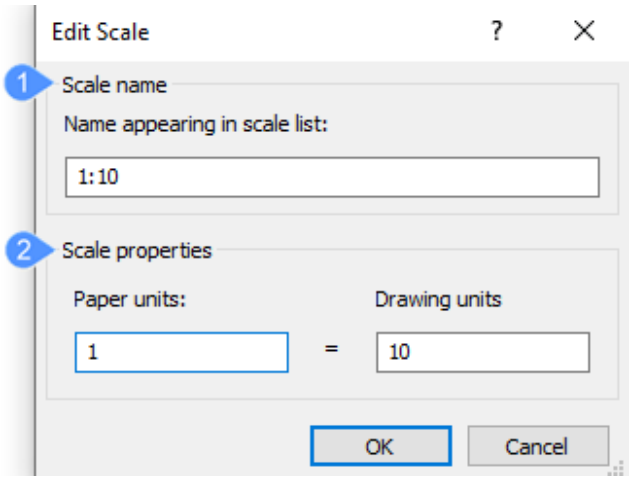
Specifies additional information

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

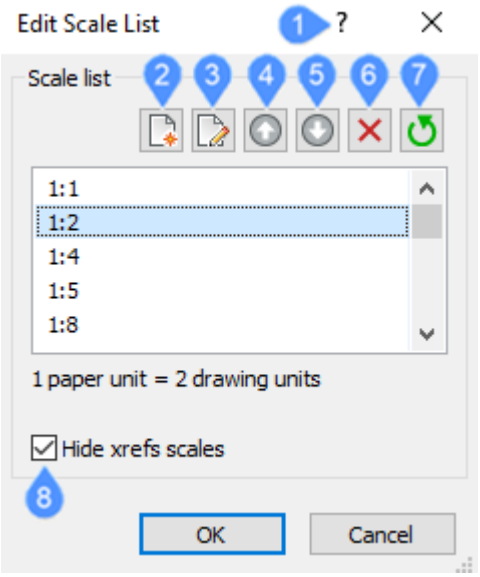
Edit the name by typing the new name.

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

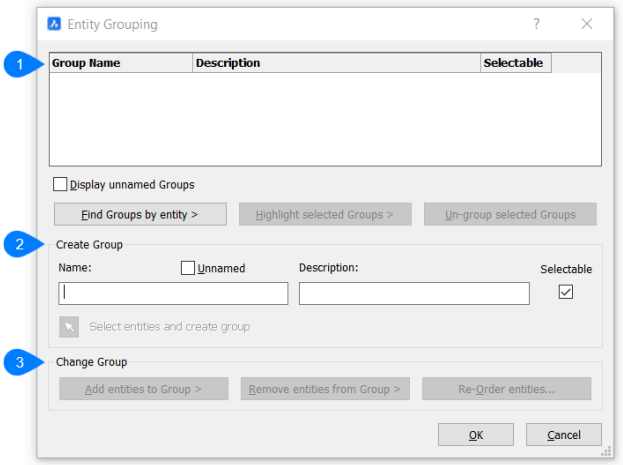
Determines if the scale factors from xrefs are displayed in the list:

- **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.

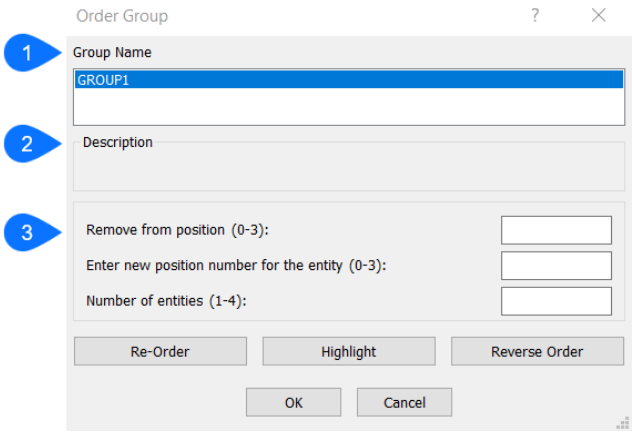
4.12.66.3 Change Group

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

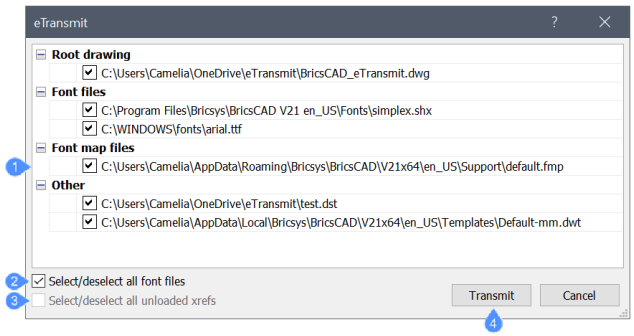


a Group Name
b Description
c Options

- **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

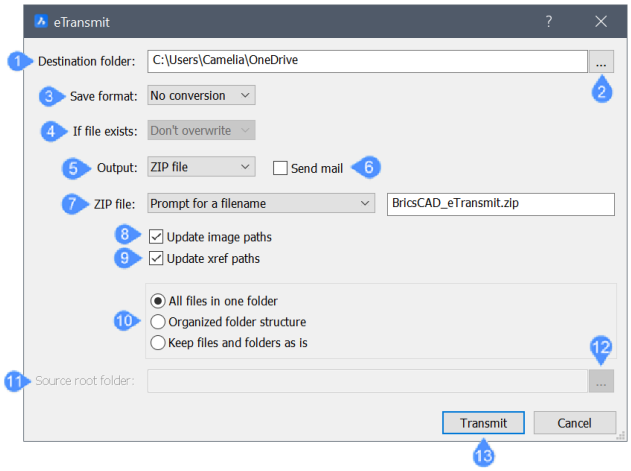
Tick the checkbox to automatically select all font files.

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.

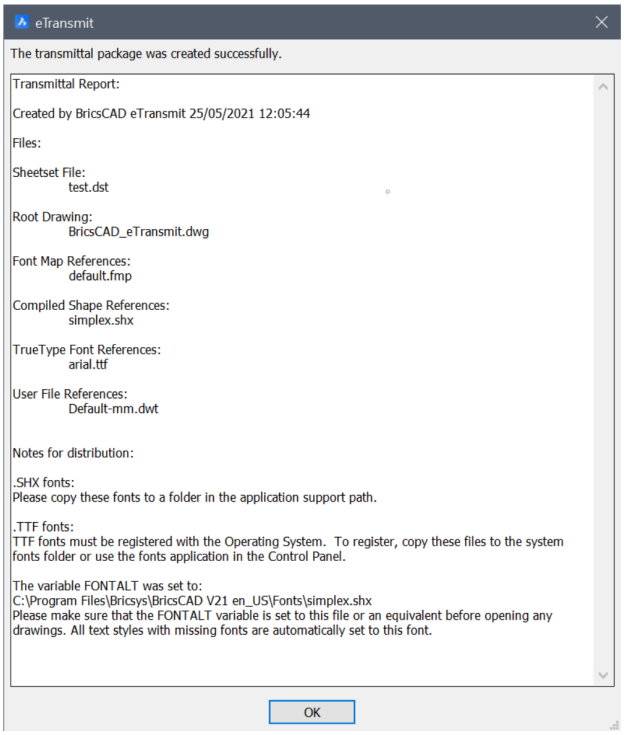
Organized folder structure
Creates a hierarchical folder tree with subfolders for all drawing file dependencies.

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.

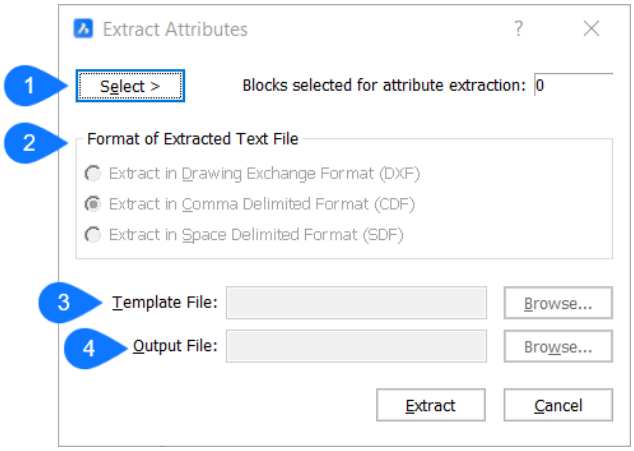
Browse (root folder)
Opens the Select the source root folder dialog box to allow you to choose a root folder.

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.

Uwaga: 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

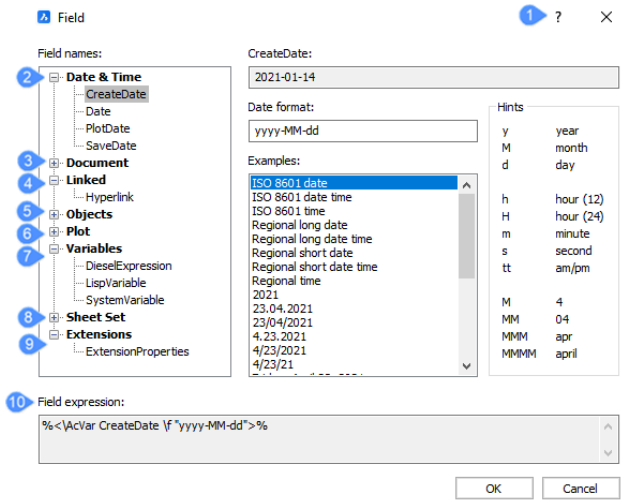
Specifies the location path of the output file by entering the path or through the Attribute Extract Output file dialog box after pressing the Browse button.

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

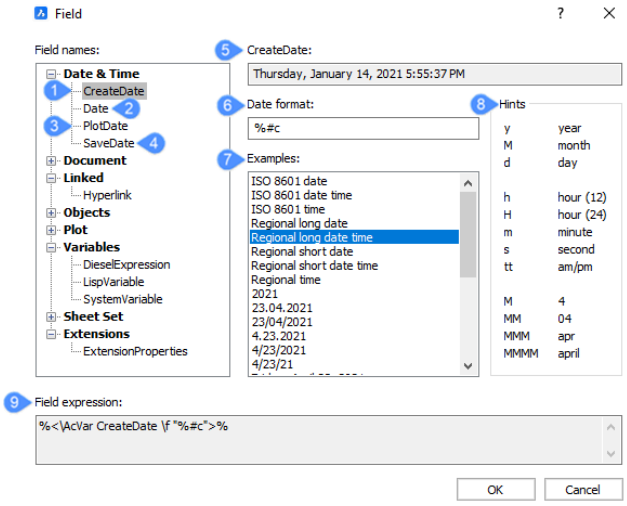
10 Field expression

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.

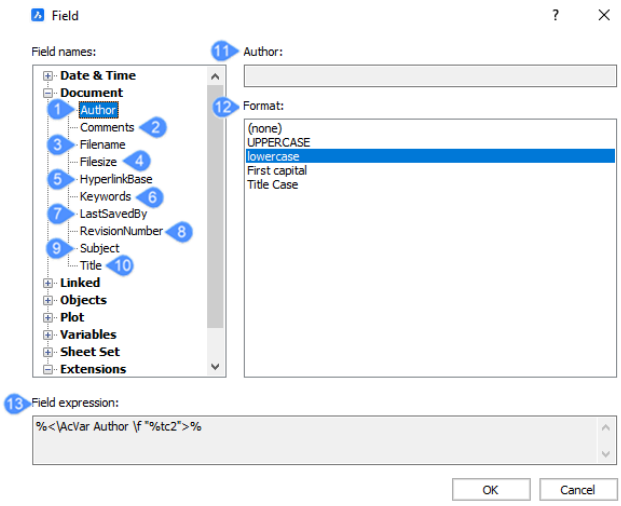
Examples
Displays date formats examples.

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.

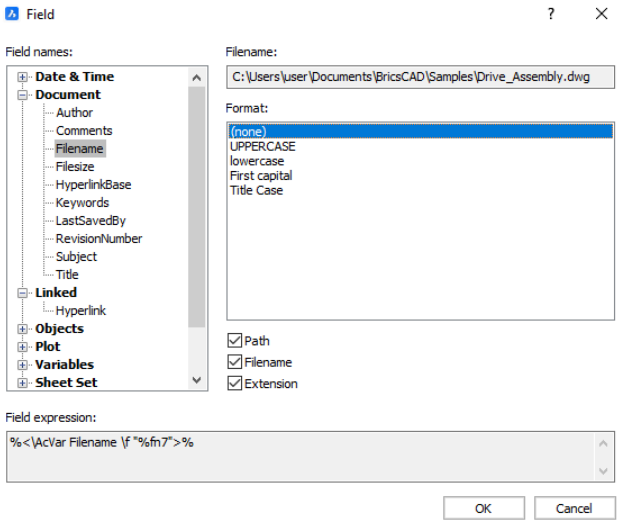


1 Author
2 Comments
3 Filename
4 Filesize
5 HyperlinkBase
6 Keywords
7 LastSavedBy
8 RevisionNumber
9 Subject
10 Title
11 Author
12 Format
13 Field expression

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.

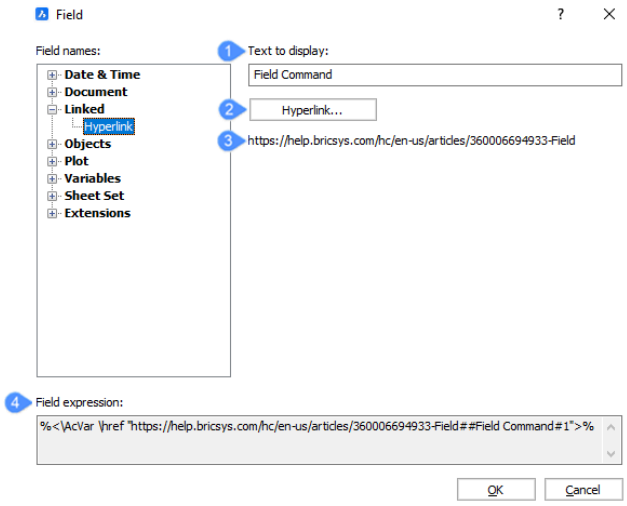
Subject
Displays the subject of the drawing.

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.

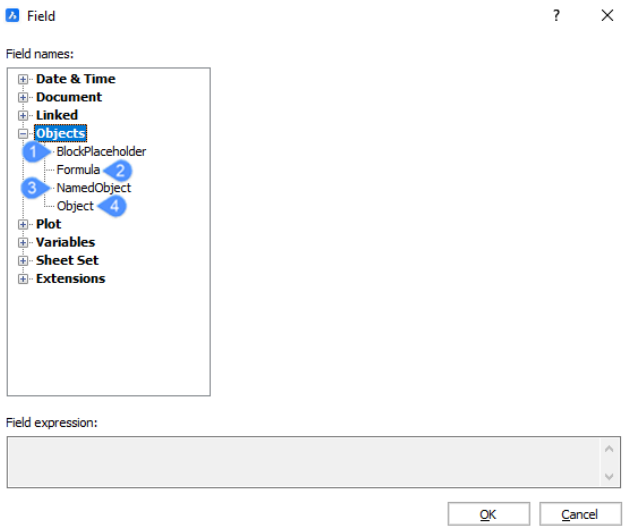
Hyperlink
Opens the **Edit Hyperlink** dialog box.

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

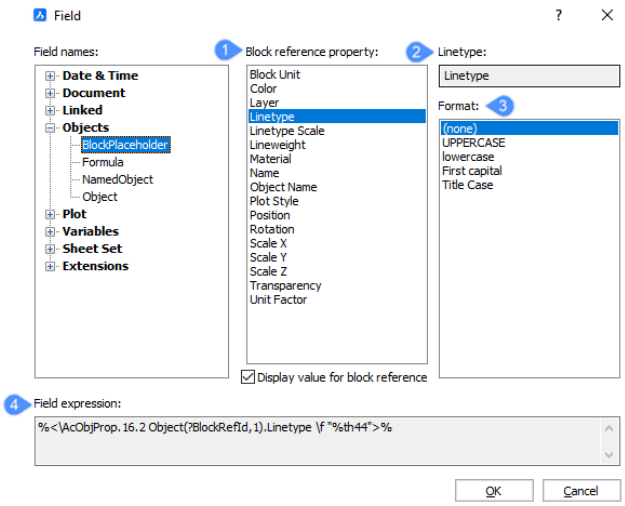
2 Formula

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

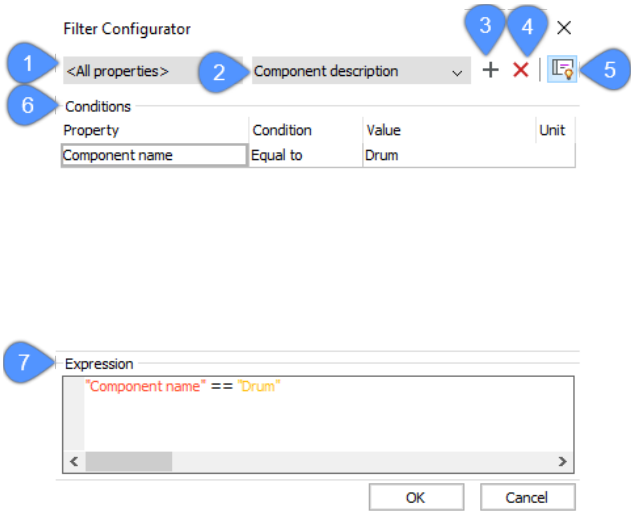
Allows to create fields showing properties of entities created by extensions, such as BIM or Mechanical.

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.

Uwaga: 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

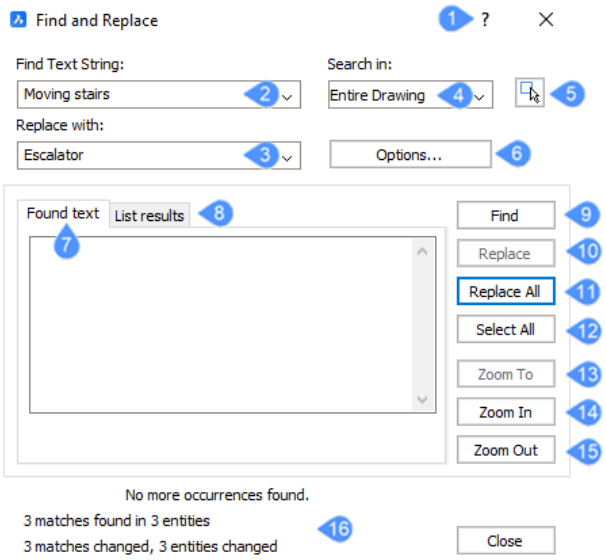
Displays all the added conditions that will be used to filter the content of the BOM table.

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.

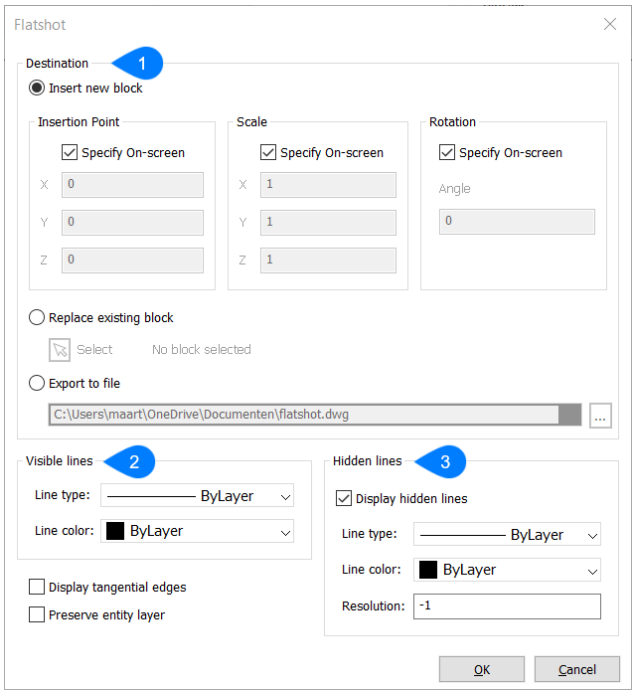
4.12.71.1 Search report

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.

Uwaga: 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

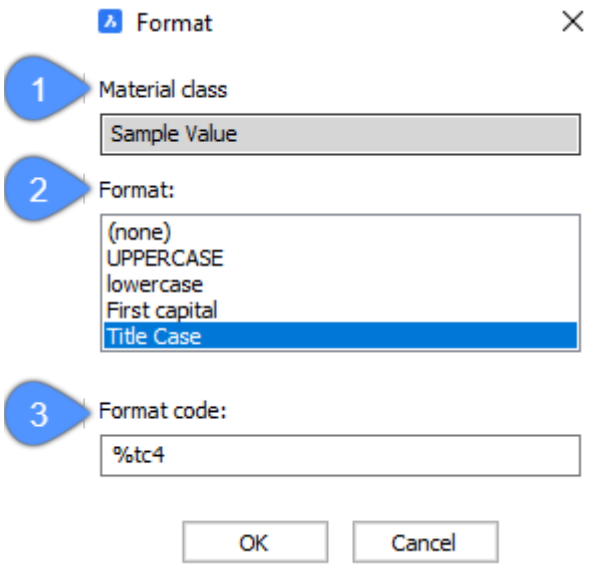
Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces.

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

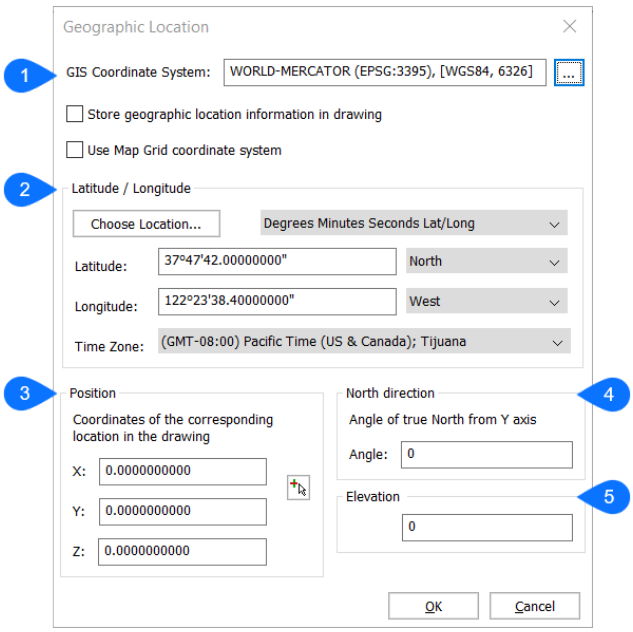
Specifies the desired format for the selected component from the list.

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

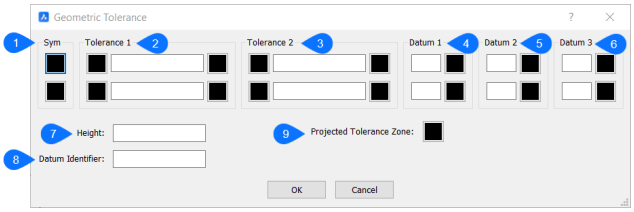
Uwaga: The setting is saved to the NORTHDIRECTION system variable.

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

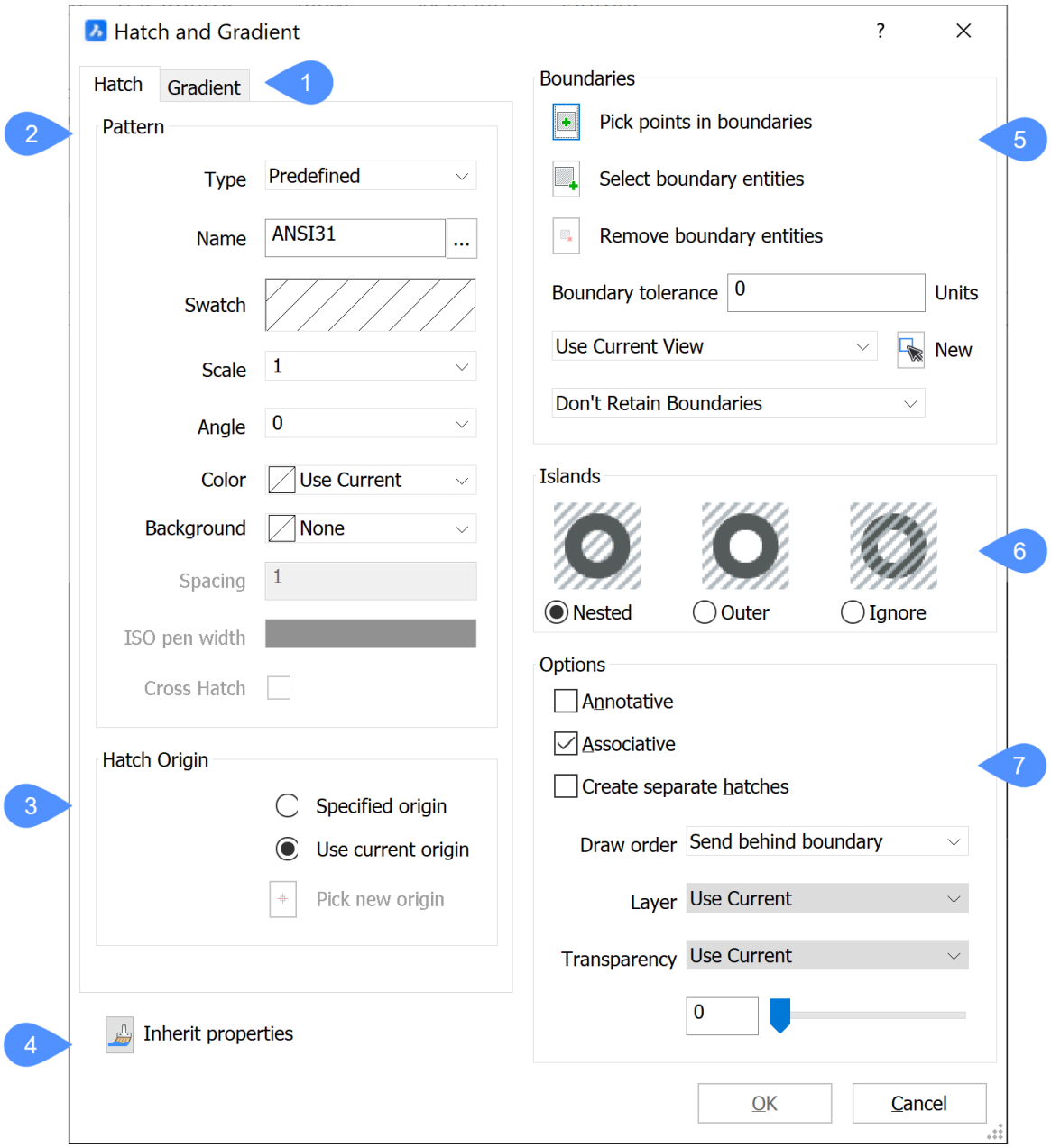
Symbol	Characteristic	Type
	Flatness	Form
	Circularity or Roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular Run-out	Run-out
	Total Run-out	Run-out

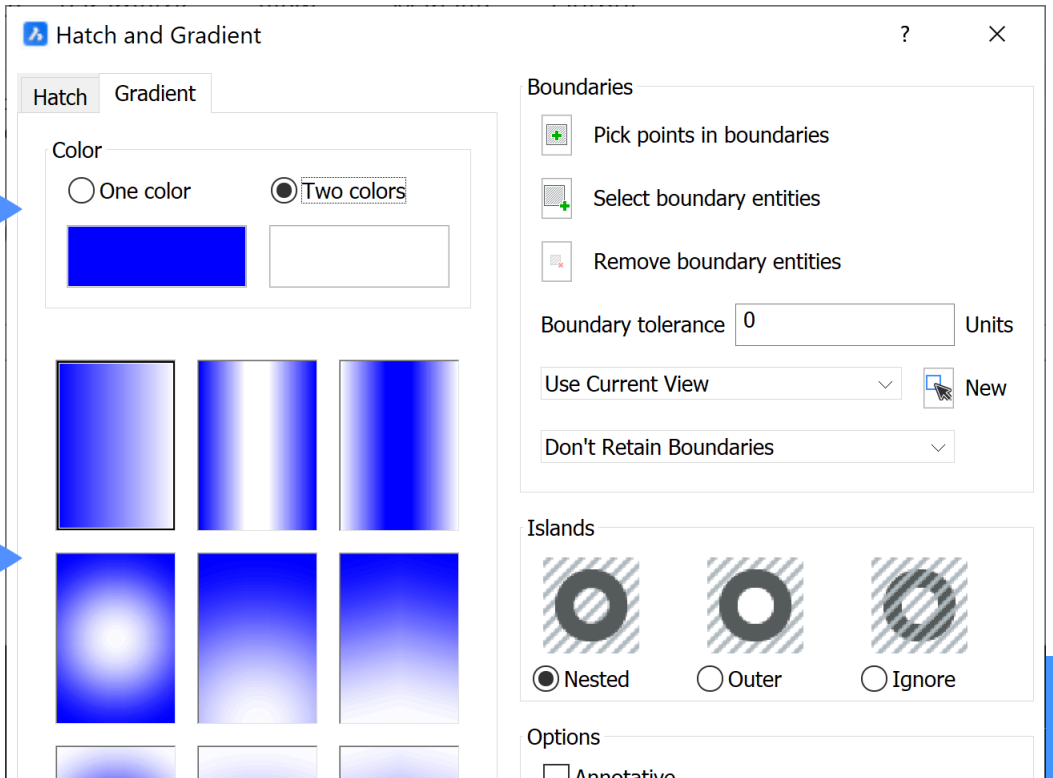
Material Condition Symbols

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.

Uwaga: This option is unavailable for the User-defined patterns.

Swatch
Displays the Hatch Pattern Palette dialog box.

Uwaga: 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.

Uwaga: The scale factor of hatch patterns is generally the same as that for text and linetypes.

Angle
Sets the angle of the pattern.

Uwaga: This angle is measured relative to the positive x-axis of the current UCS.

Color
Sets the color of the hatch pattern lines.

Uwaga: 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.

Uwaga: This option is available for only user defined patterns.

ISO pen width
Specifies the lineweight.

Uwaga: This option is available only for ISO hatch patterns.

Cross Hatch
Determines whether the pattern is crossed (repeated at 90 degrees to the original).

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

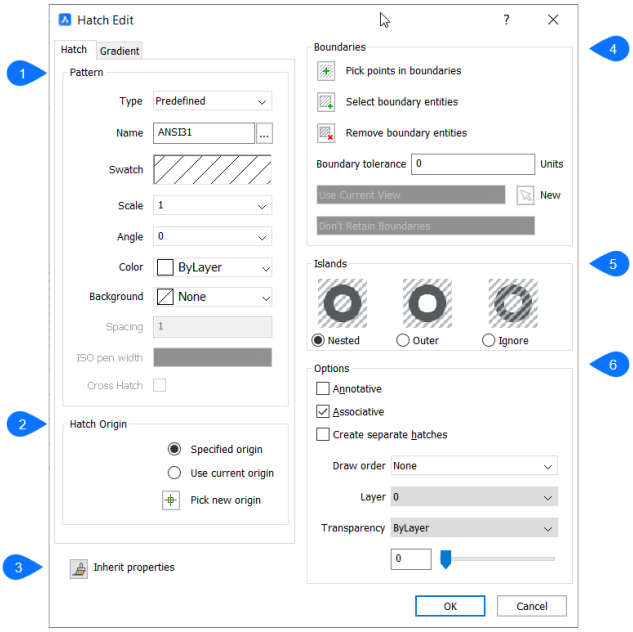
4.12.76.9 Gradient Orientation

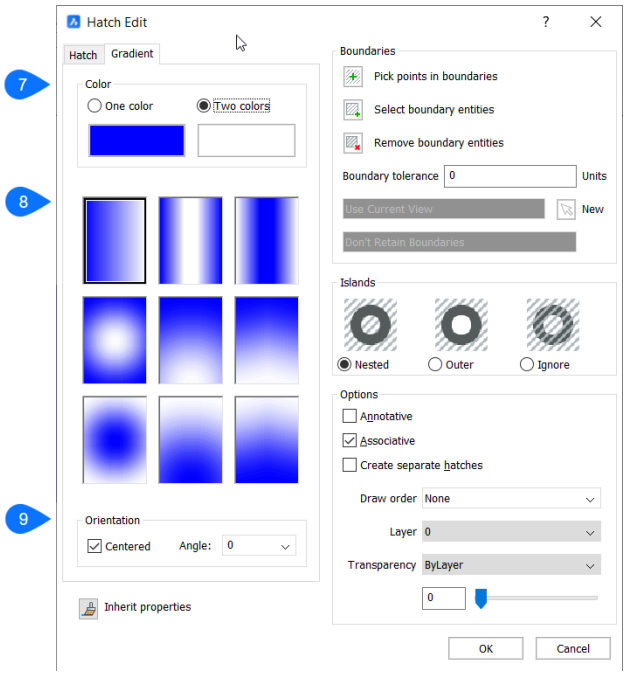
Specifies how the fill is placed in the boundary.

- 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

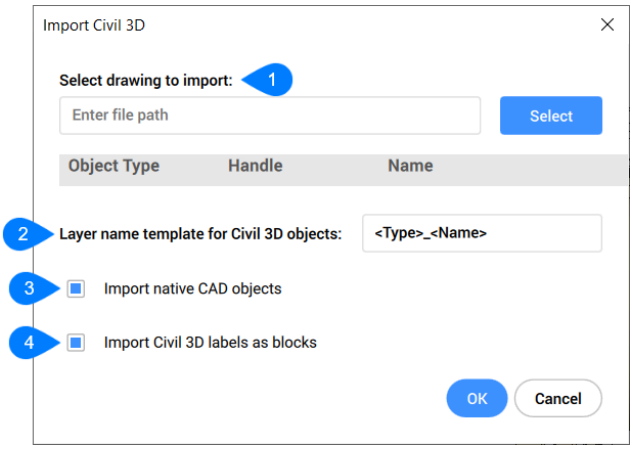
7 Color

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

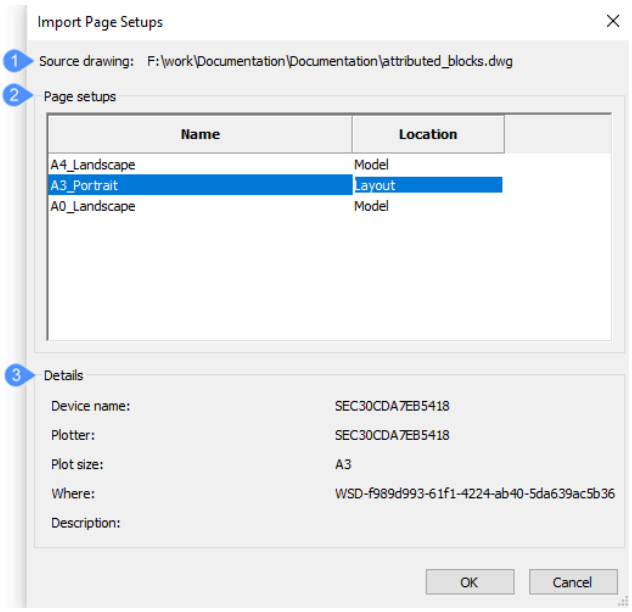
Toggles the option to also import native CAD objects.

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:

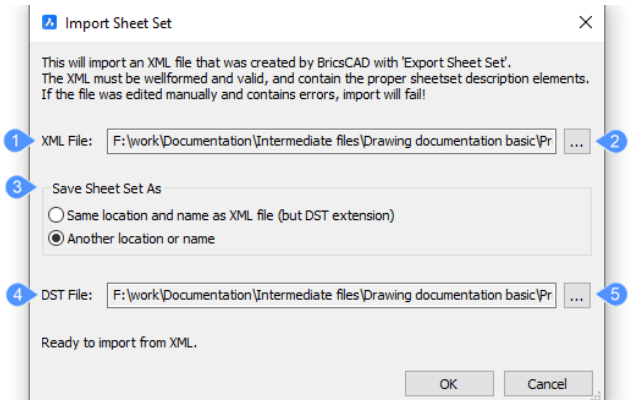
- **Name**: specifies the names of page setups in the drawing. When the drawing contains no page setups, then the dialog box is blank.
- **Location**: specifies the layout and model locations of page setups.

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

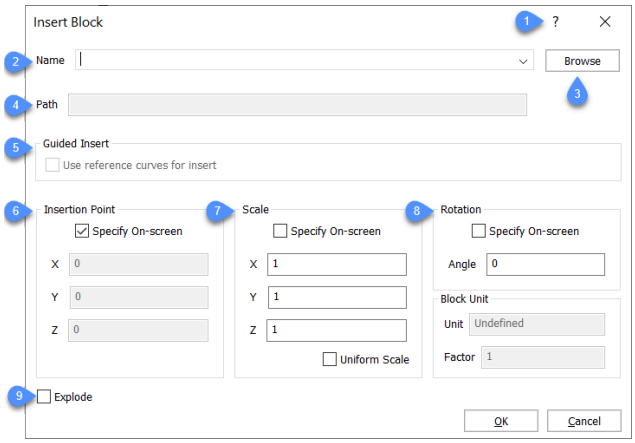
When you choose to save the imported XML file as DST file to a new location or name is displayed the path of this file.

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.



Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

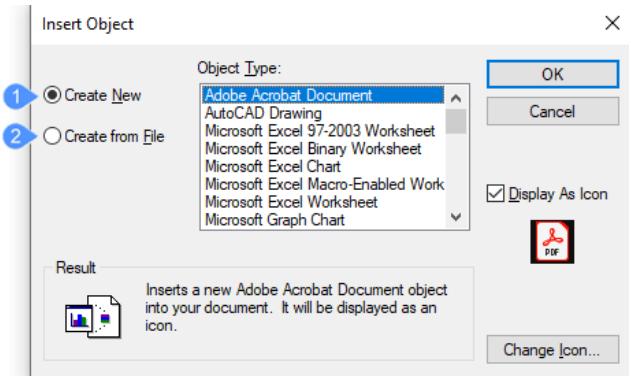
4.12.81.1 Explode

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.

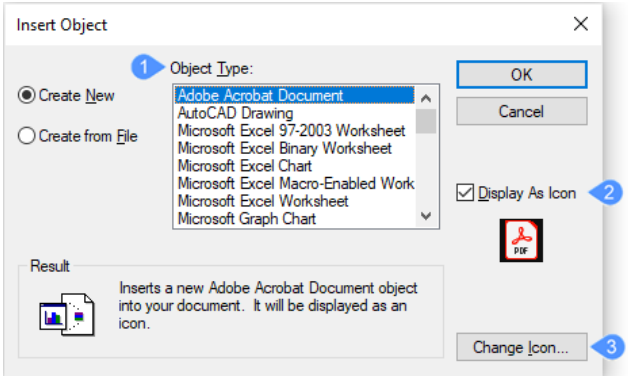


1 Create new Object

2 Create from File

Uwaga: 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.

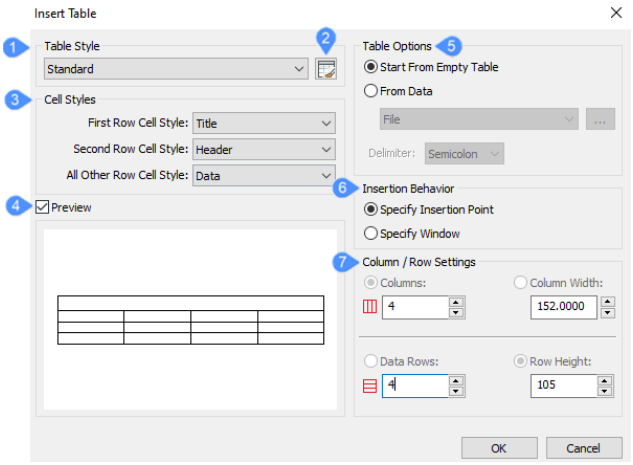
4.12.82.2 Create from File

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.

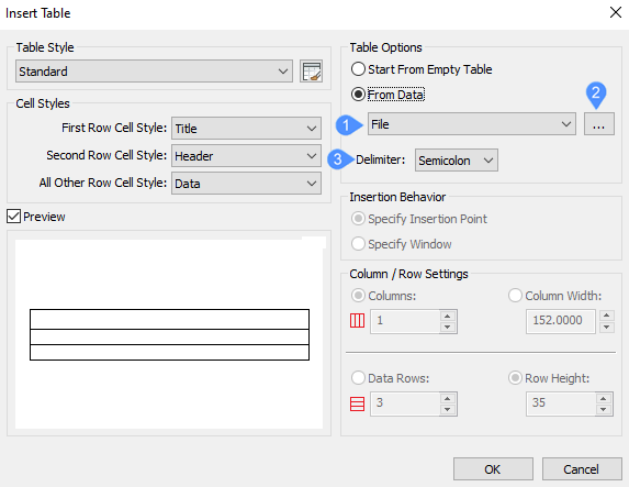
4.12.83.4 Preview

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**

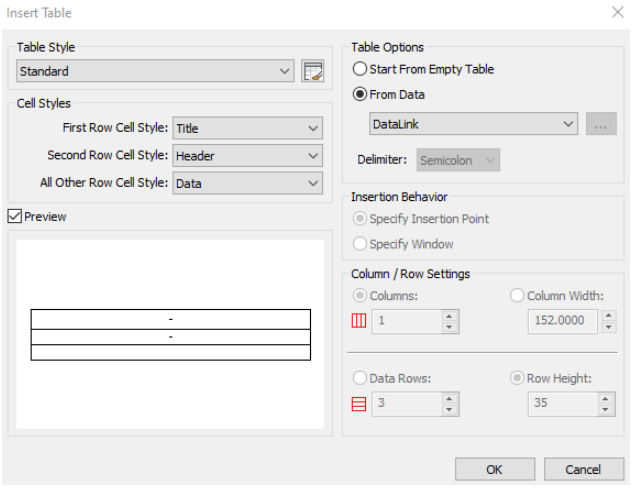
  Displays the **Open** dialog box from where you will select the file.

- **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.

  Uwaga: 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.

Uwaga: 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

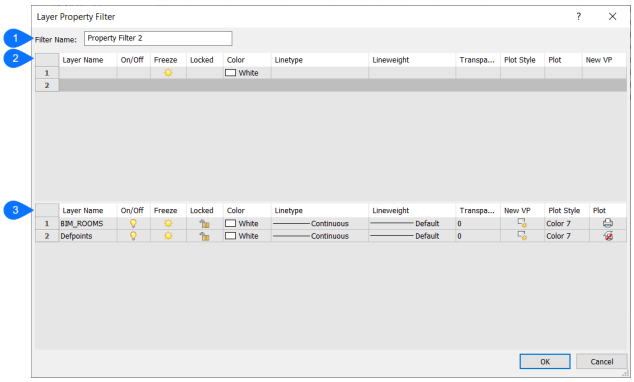
And choose one of:

- **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).

Uwaga: 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.

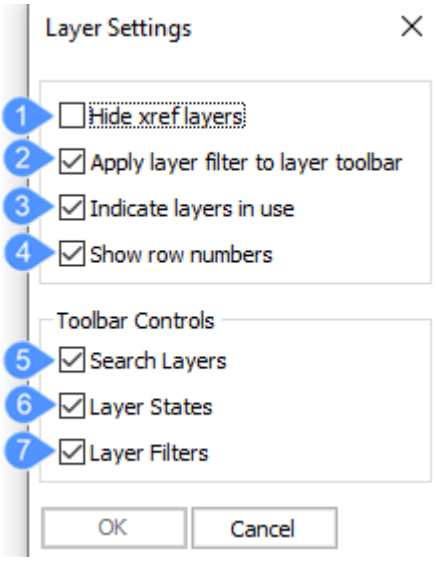
Properties
Specifies which layers to include according to their properties, such as freeze and color.

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

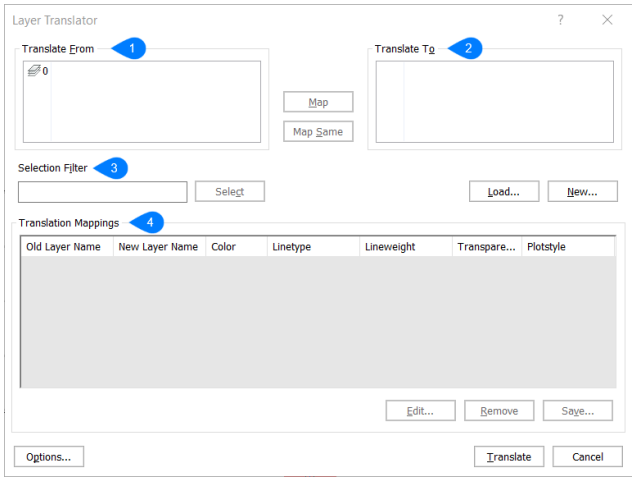
Toggles the display of the **Layer States** drop-down list.

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.

Uwaga: You can specify the layers to which you want them to map by loading layer information from an existing DWG, DWS, or DWT file.

Uwaga: 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.

Uwaga: 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

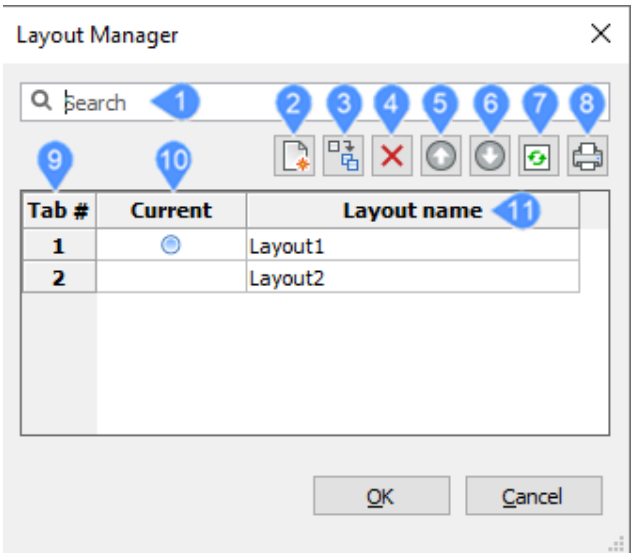
Displays an overview of which layer and its properties are translated to another layer.

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.

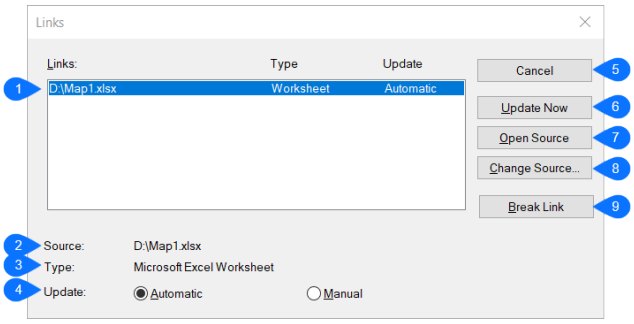
4.12.87.1 Layout name

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

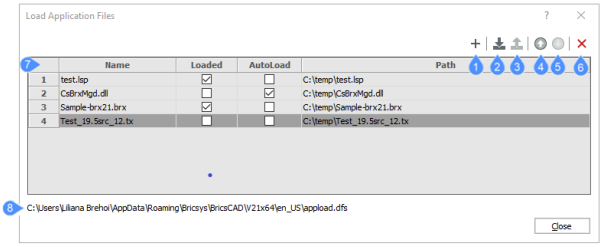
Breaks the link with the source file.

Uwaga: 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.

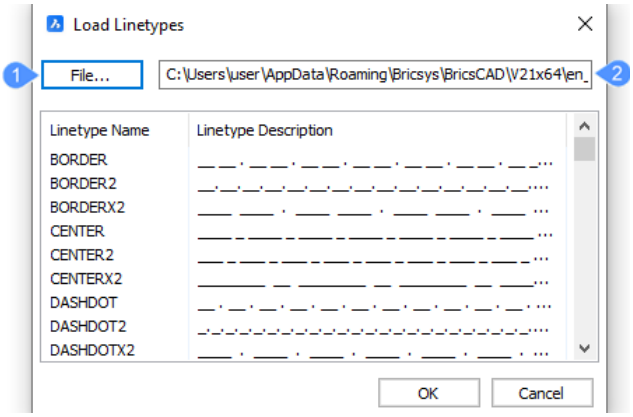
- **Name**: specifies the name and the extension of the application file from the list.
- **Loaded**: specifies if a file is loaded or unloaded in the current BricsCAD session.
- **Autoload**: specifies if a file will be loaded automatically in the next BricsCAD session.
- **Path**: displays the location 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

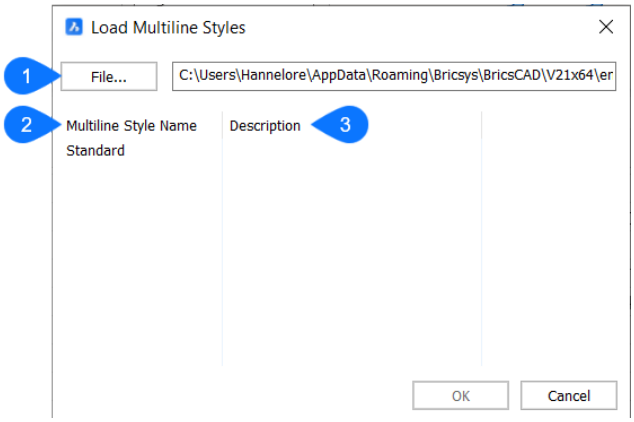
Opens the **Choose a Linetype File** dialog box.

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

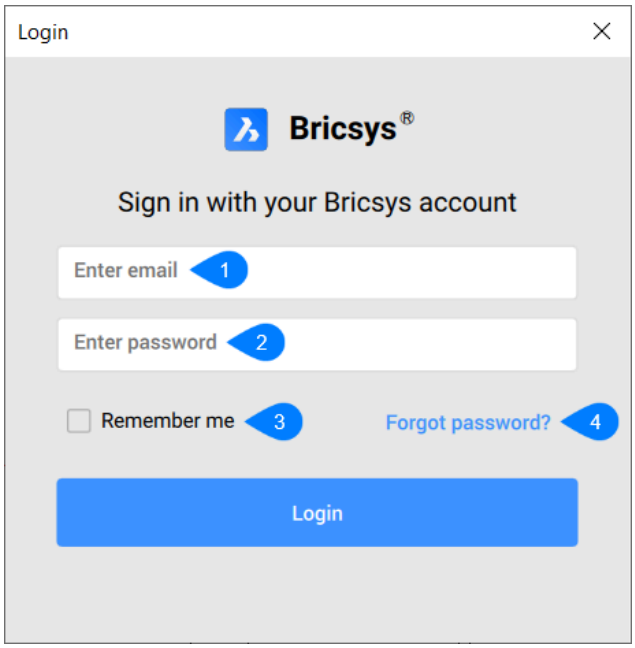
Displays the names of the available multiline style files in the drawing. You can select a multiline style file from this list.

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

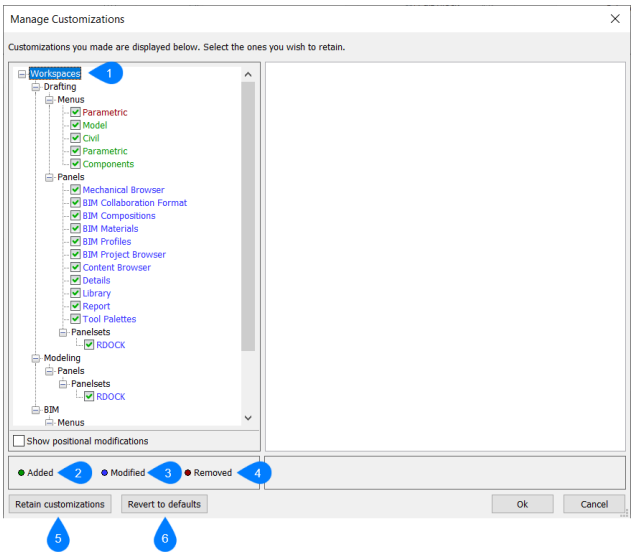
Check this box to keep being logged in.

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.

Uwaga: 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.

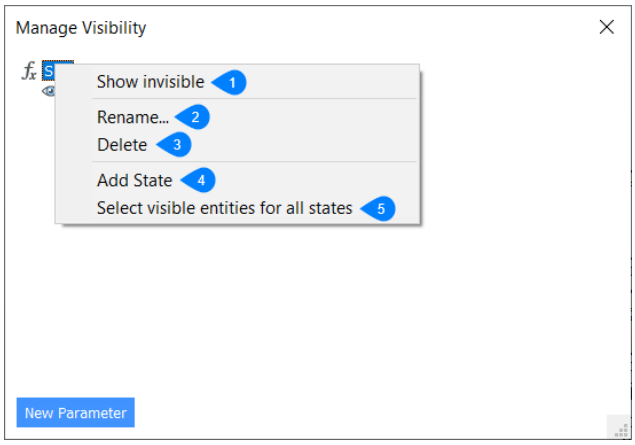
4.12.93.6 Revert to defaults

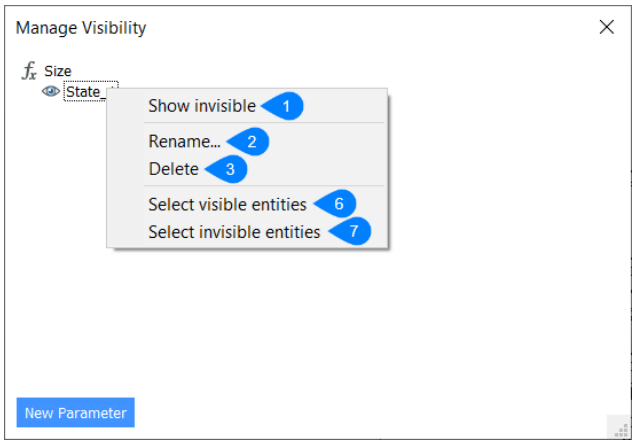
Reverts your local default CUI to the default one installed by BricsCAD in:

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

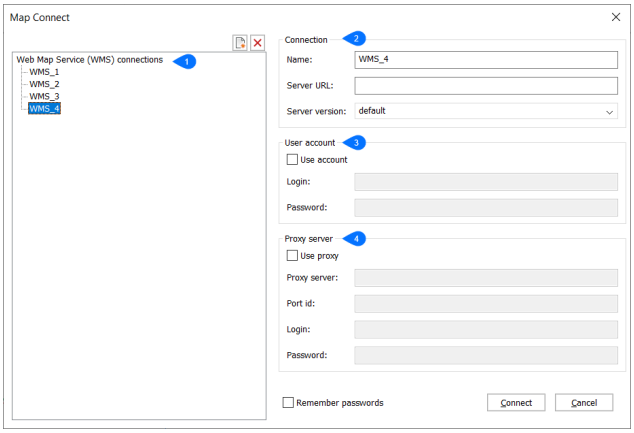
Visualizes all entities that are selected.

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.

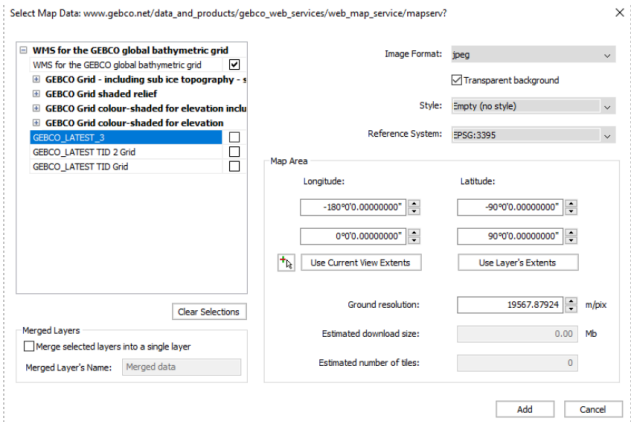
Proxy server
Specifies the URL of the proxy server.

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.

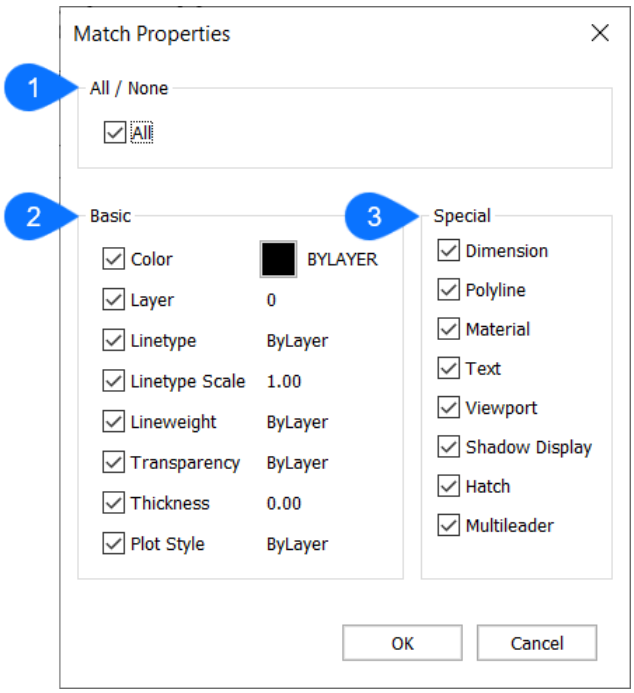
The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved.

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.

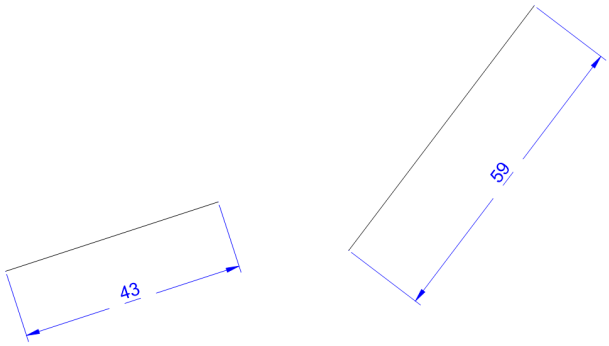
Thickness
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

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.

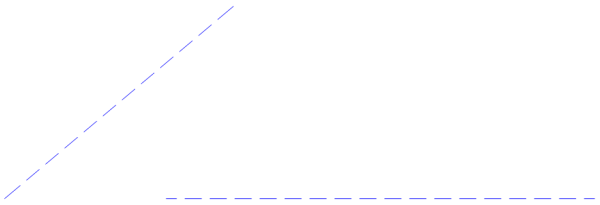
4.12.96.3 Special

Selects other less basic properties to copy.

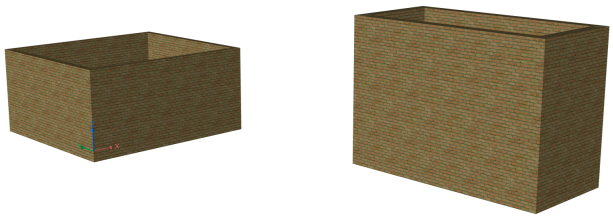
Dimension
When checked on, the dimension style and annotative properties of dimensions, leaders and tolerances is copied.



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.



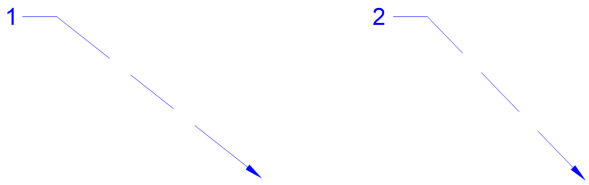
Viewport
When checked on, the viewport-specific properties are copied, such as on or off, snap and grid status, and scale factor. Other properties, such as clipping or frozen layer states, are not copied.

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.

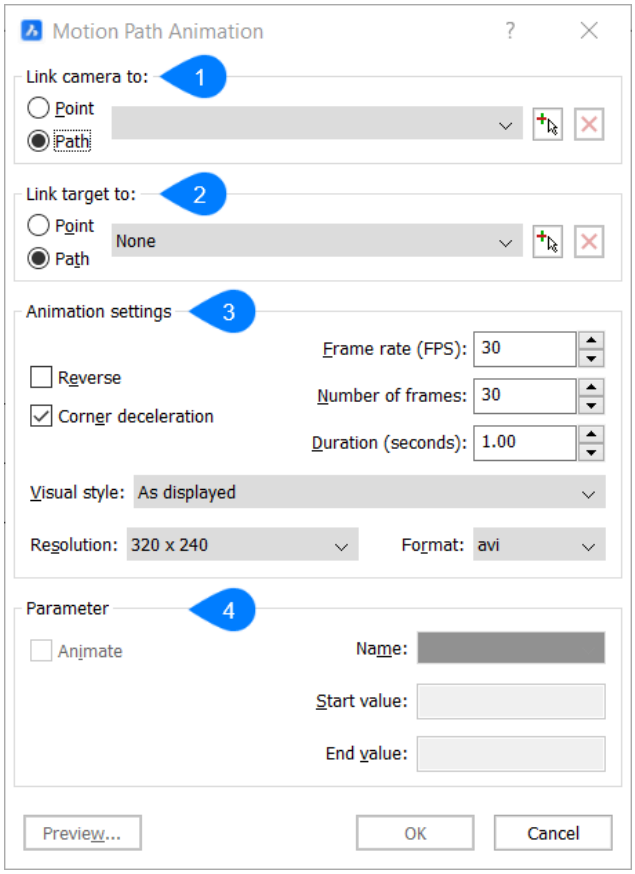


Multileader
When checked on, the multileader style and annotative scaling is copied.



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.

Uwaga: 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.

Uwaga: After selecting a point or path, the option to name the entity is available.

Uwaga: 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.

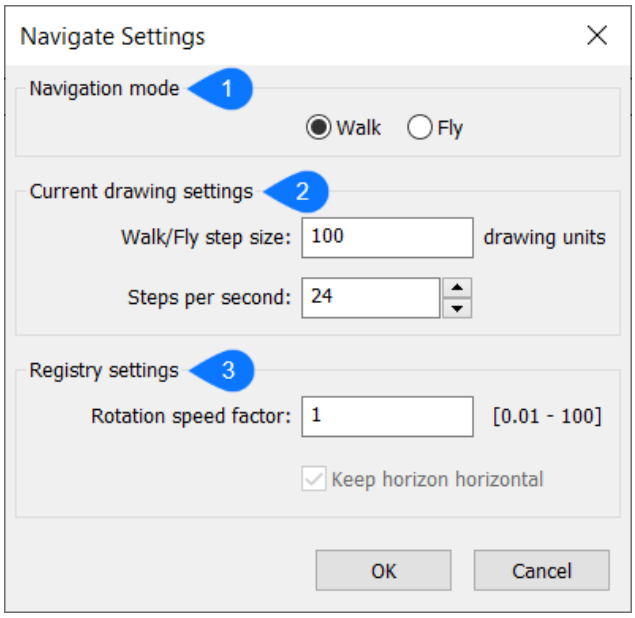
Name
Specify the name of the movie.

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

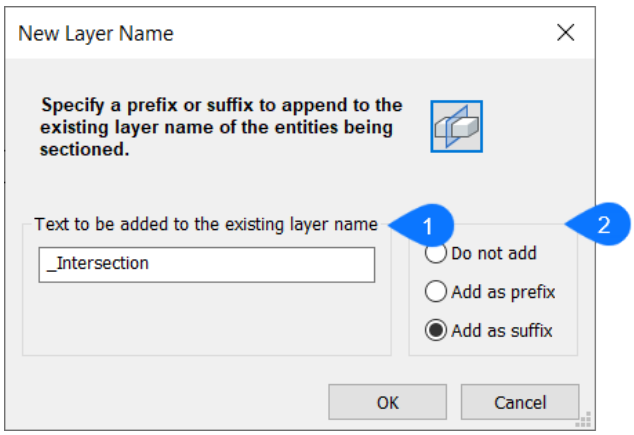
Settings which are saved with the registry, they will be the same in other drawings.

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

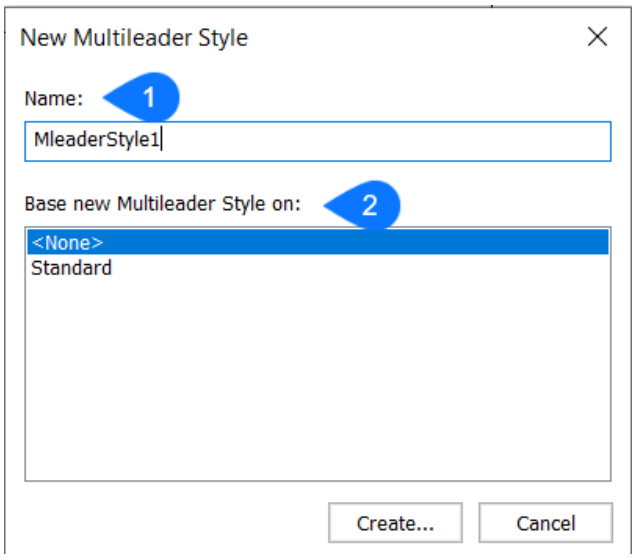
Do not add
No text will be added to name of the layer when a copy is created.

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.

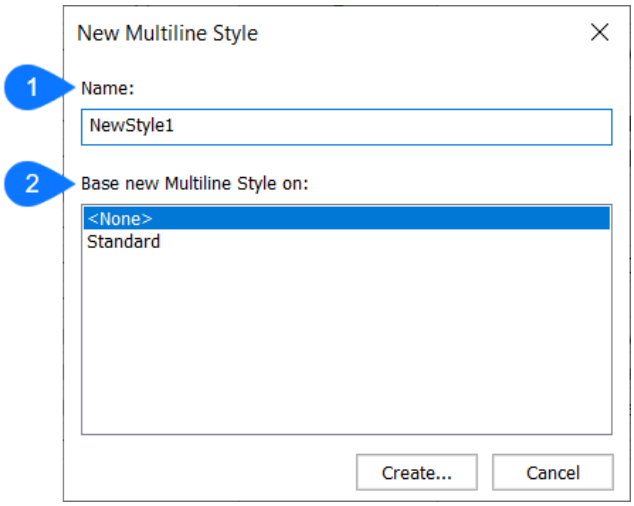
4.12.100. Base new Multileader Style on

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.

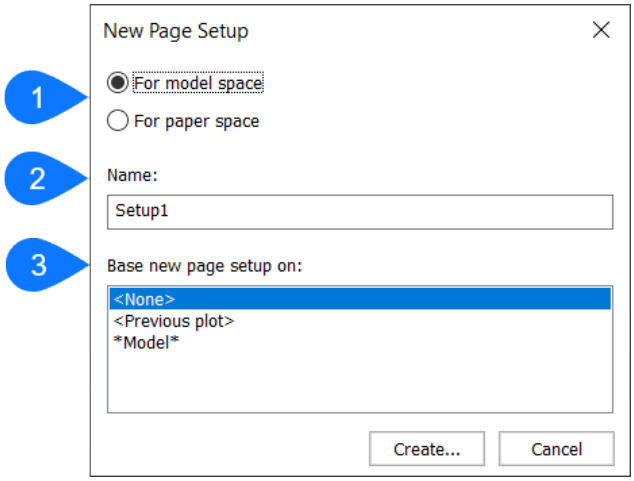
4.12.101. Base new Multiline Style on

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

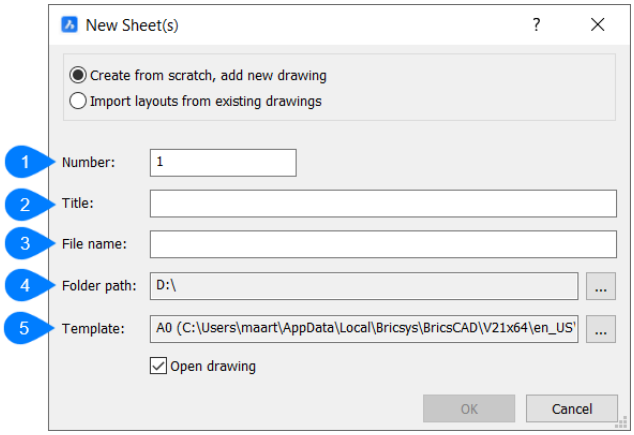
3

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.



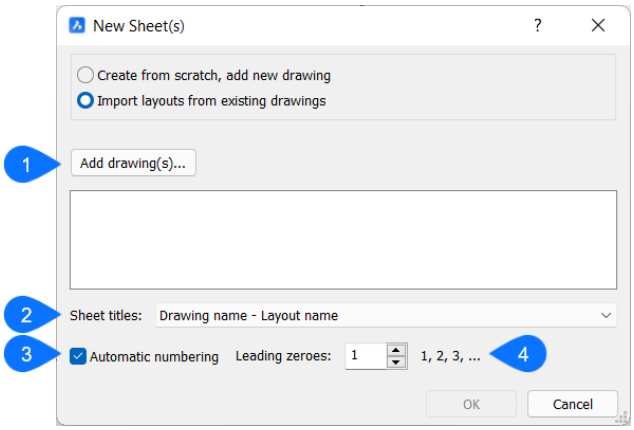
1 Number

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

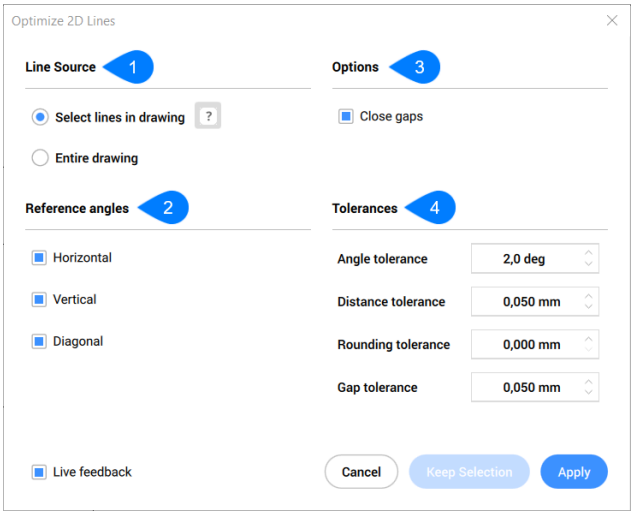
2

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.

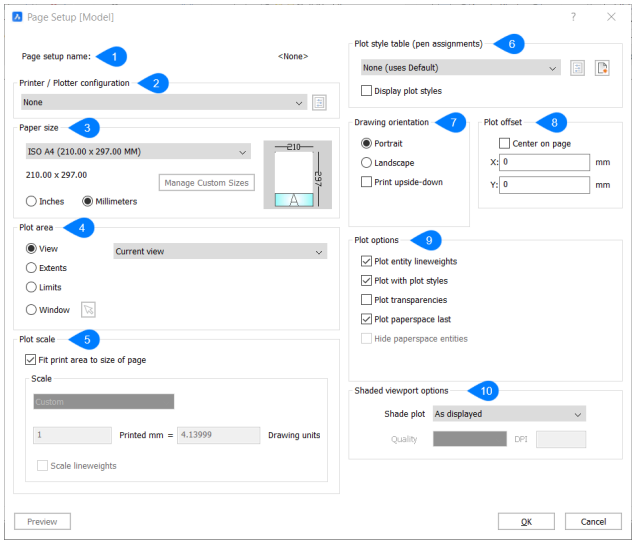
4.12.104. Tolerances

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: This option is only available when printing layouts.

Hide paperspace entities
Removes hidden lines from 3D entities in paper space when checked.

Uwaga: This option is disabled when printing model space.

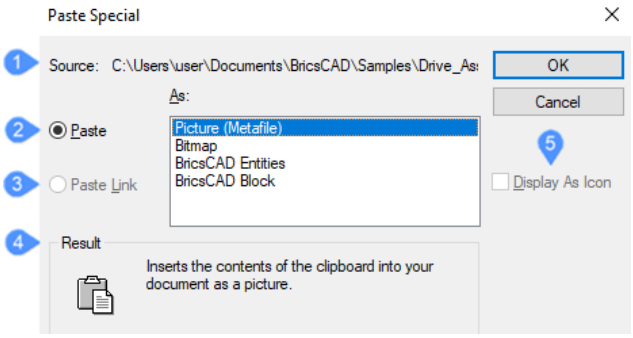
4.12.105. Shaded viewport options 10

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:*

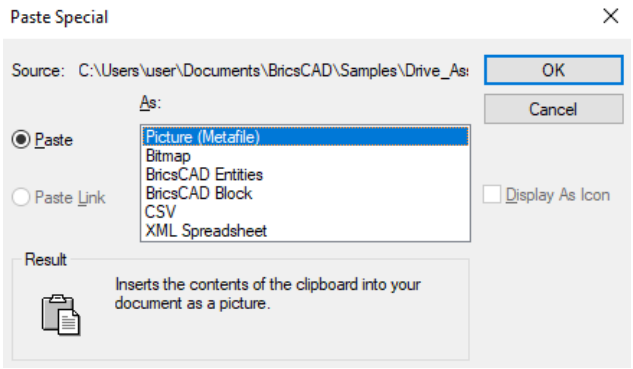
Select mirroring control point – *(Pick a second point for the mirror line)*

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.

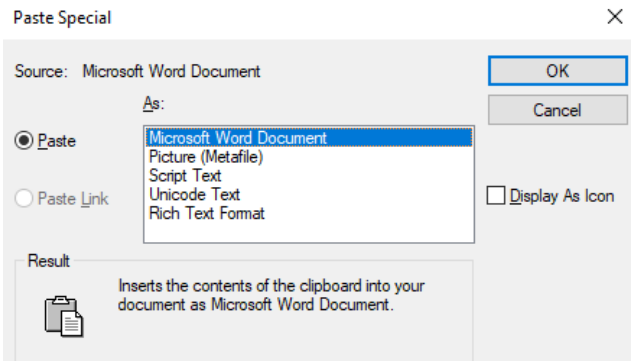
Paste As BricsCAD Block

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.

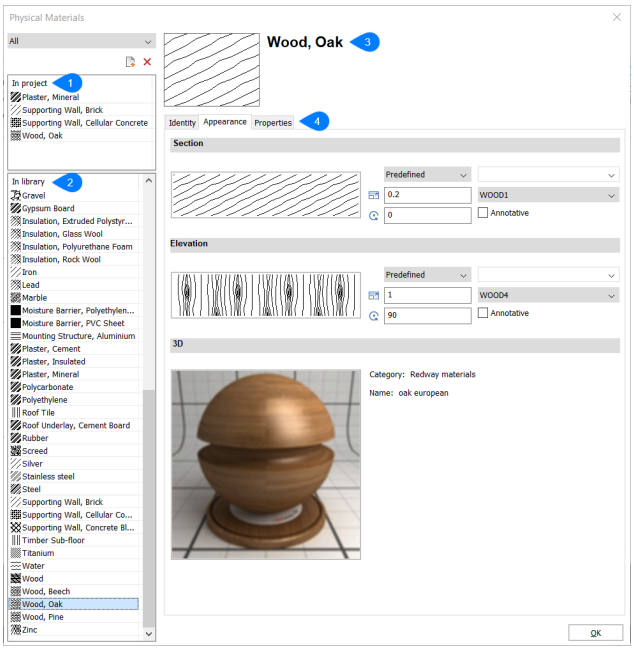
4.12.106. Display As Icon

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

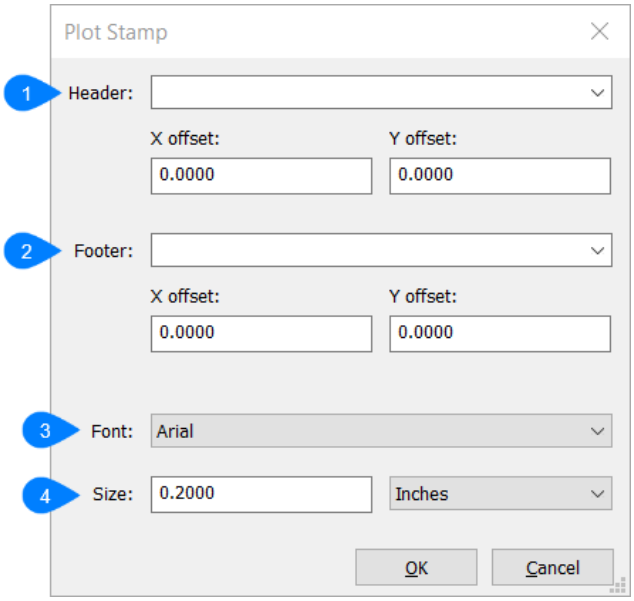
- **Section**: specifies the hatch pattern used to display the material in a section view.
- **Elevation**: specifies the hatch pattern used to display the material in an elevation view.
- **3D**: shows the three dimensional texture of the material that will be used in all 3D views.

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

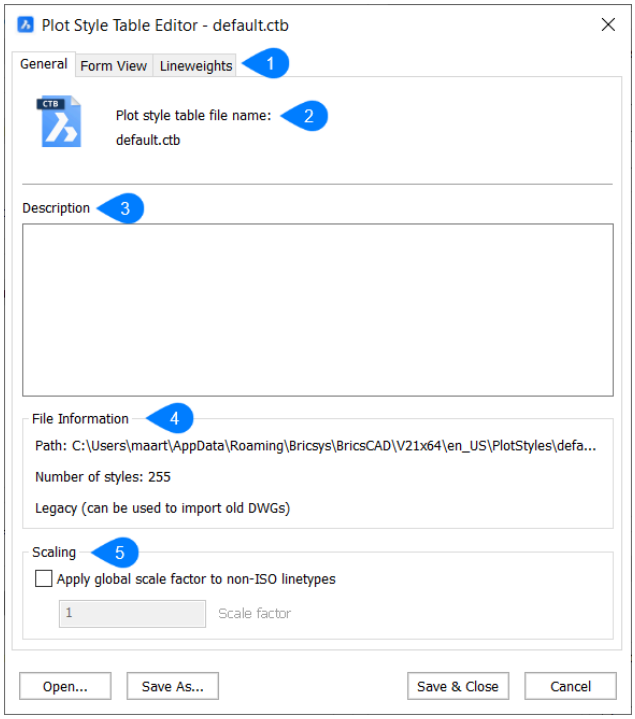
4

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

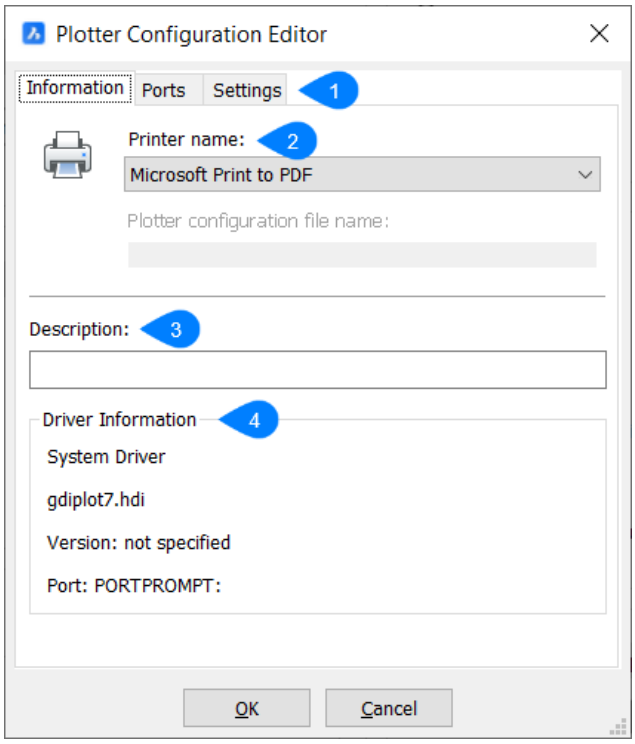
Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.

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.

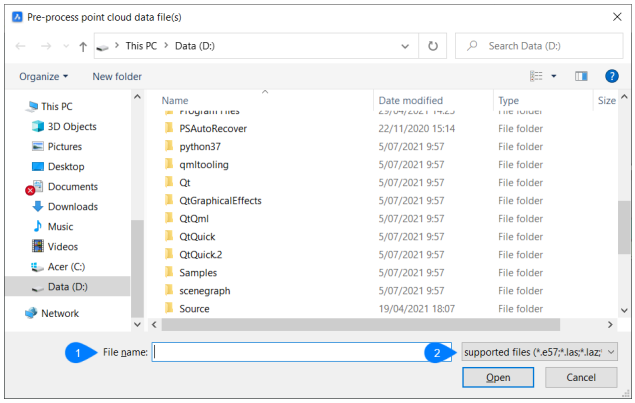
4.12.110. Driver Information

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.

Uwaga: A manual search can be done in the window above.

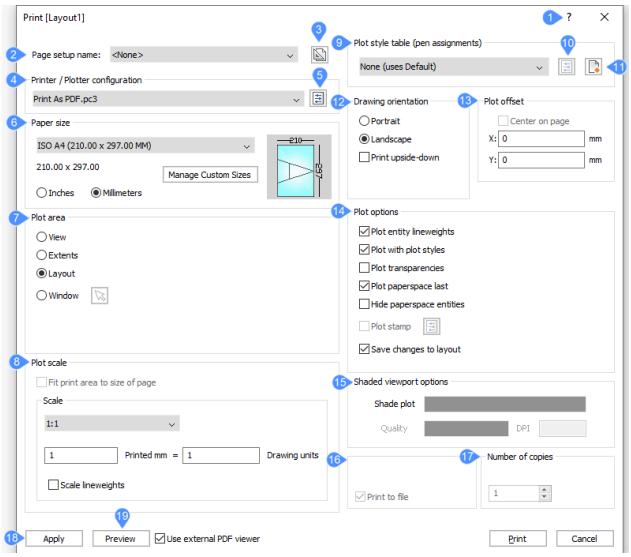
4.12.111. Supported files

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

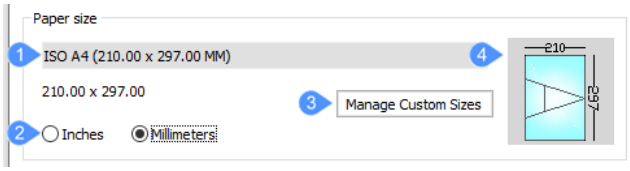
Allows you to select from the drop-down list the printer or plotter.

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

Uwaga: This option is only available for the PDF output.

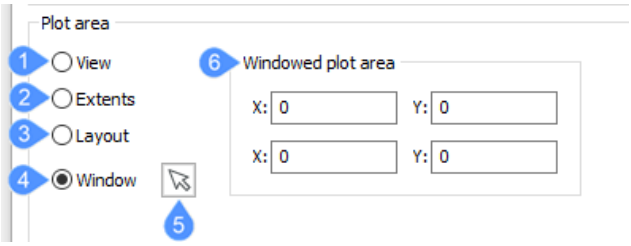
Manages custom paper sizes when Print as PDF.pc3 is selected in the **Printer/Plotter Configuration** list. The **Custom Paper Sizes** dialog box opens where you can create your own paper size.

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

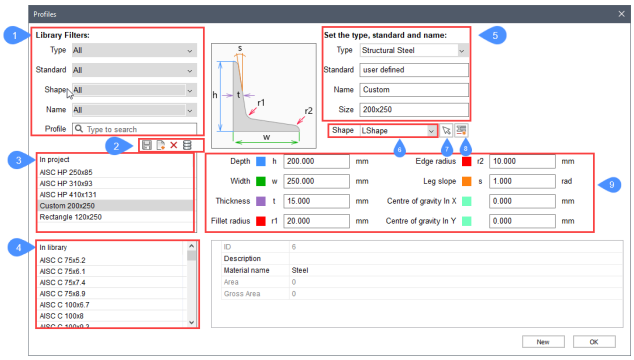
Displays a preview of the print. In the preview window, click the **Print Settings** button to return to this dialog box. See the PREVIEW command.

Uwaga: 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.

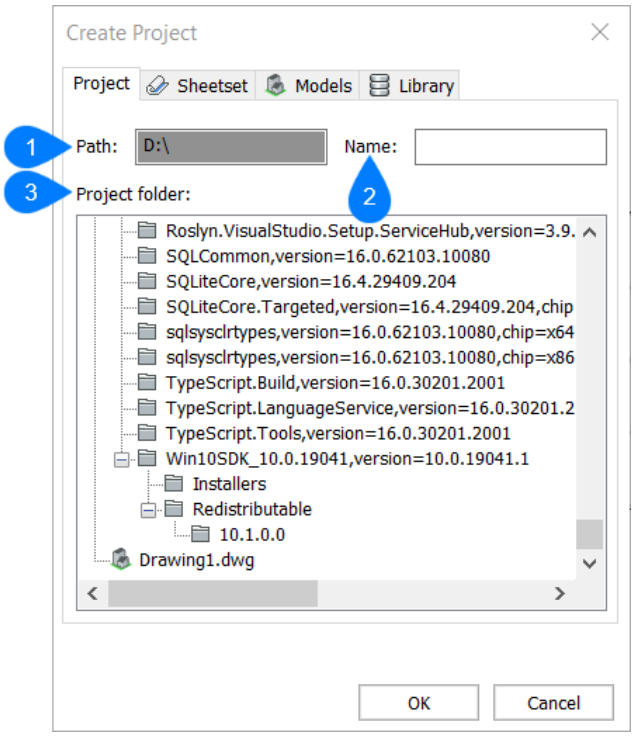
4.12.113. Properties of the profile

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.

Uwaga: By default, the name is that of the parental folder.

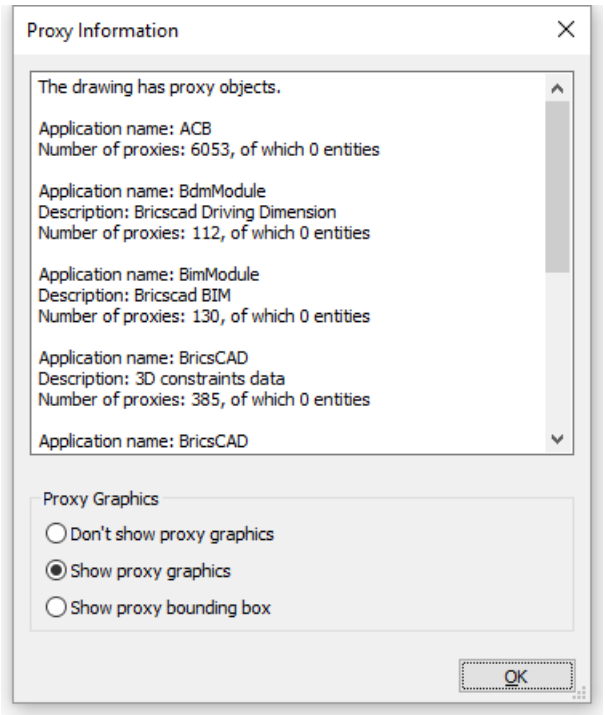
4.12.114. Project folder

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.

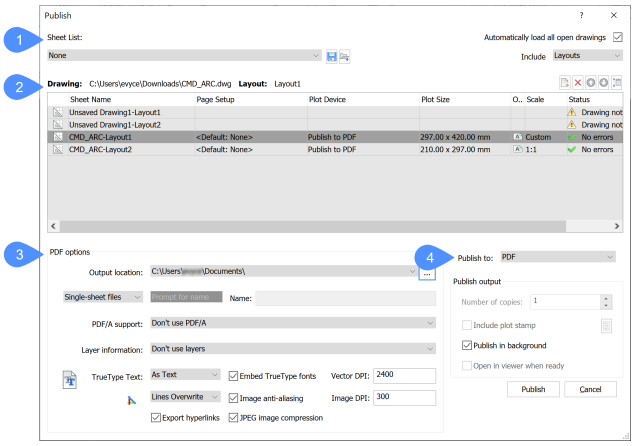
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command

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.

Uwaga: 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

Uwaga: 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.

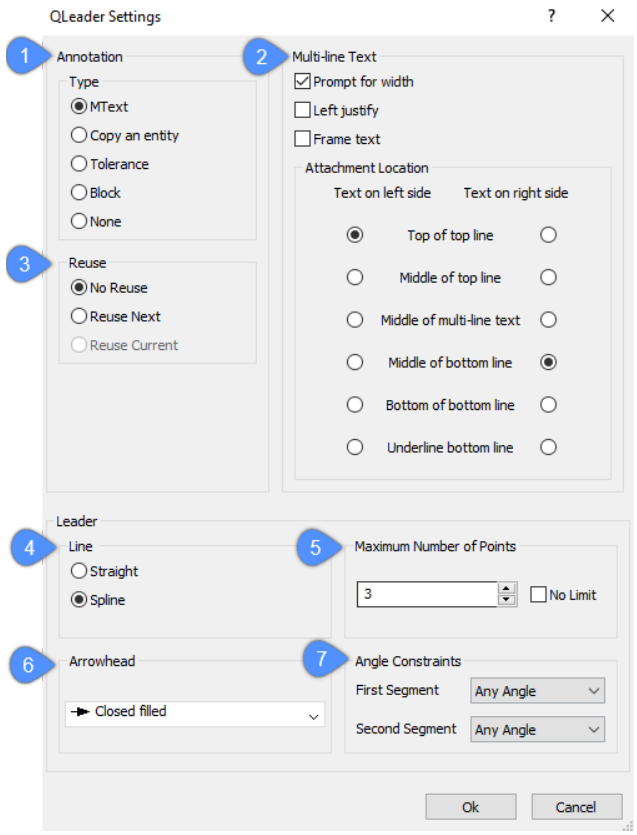
Open in viewer when ready

Determines if the sheet set will be opened in the viewer when it is ready.

Uwaga: 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.

  Uwaga: 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

Uwaga: 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

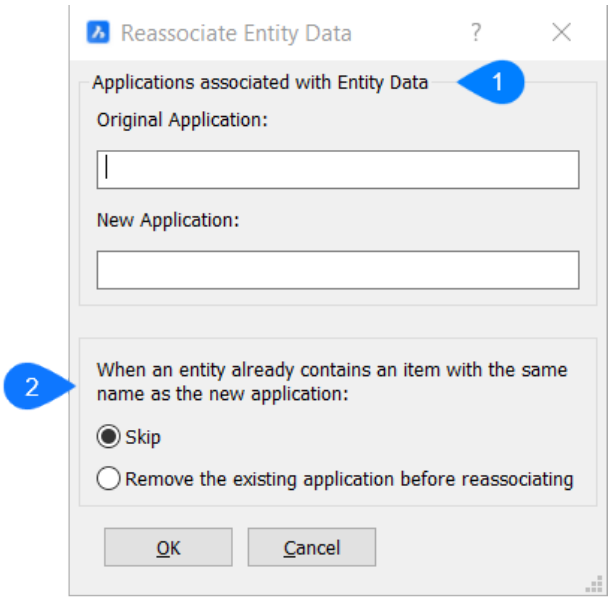
Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees.

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

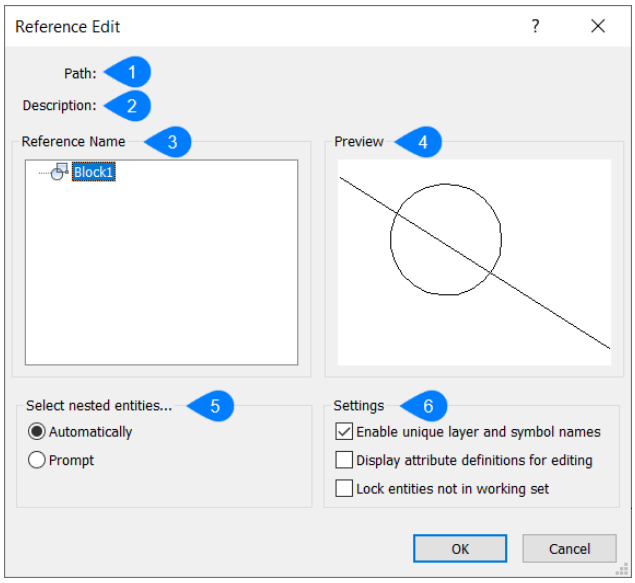
Skips entities that already contains an item with the same name as the new application.

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.

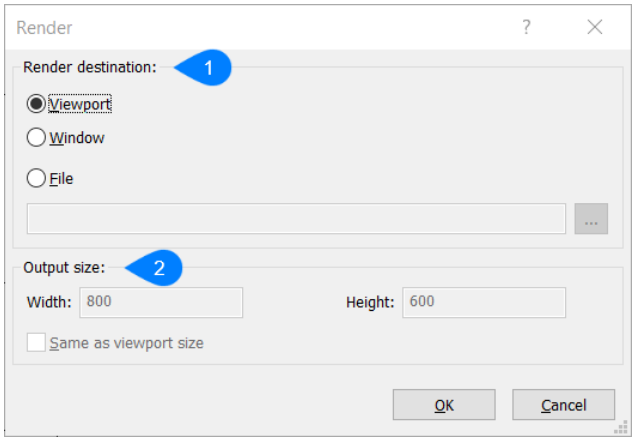
Enable unique layer and symbol names
Toggles the use of 0 in layer and other names.

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

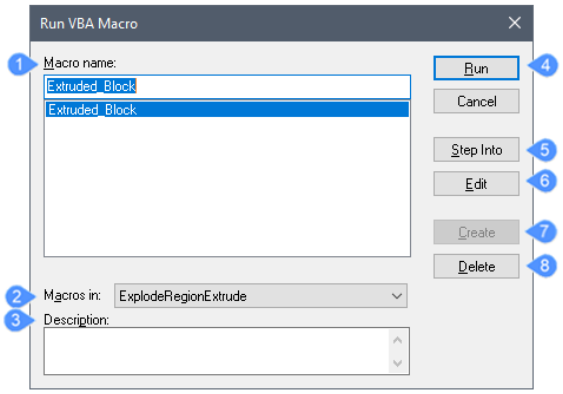
Specifies the height of the rendering, in pixels.

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.

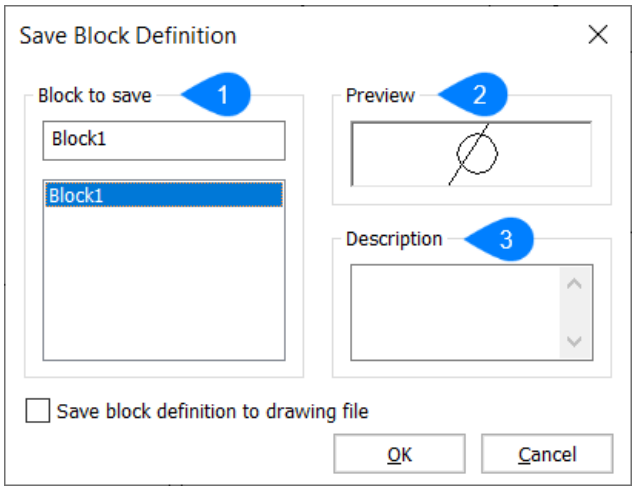
4.12.121. Delete

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.

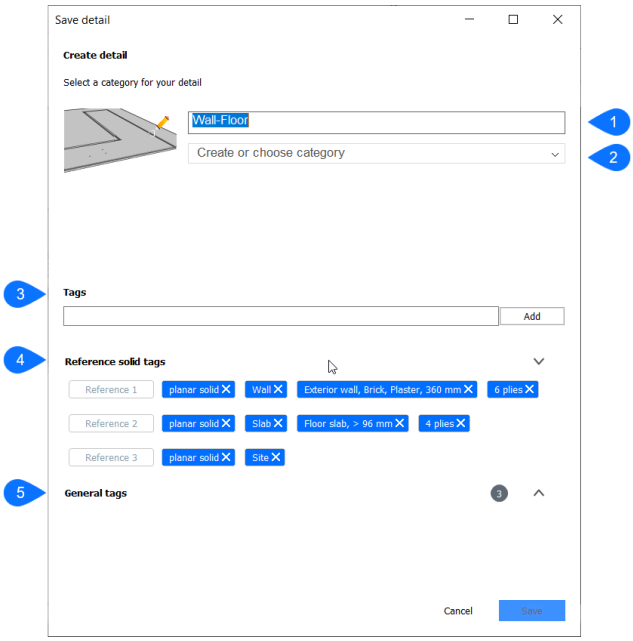
4.12.122. Description

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.

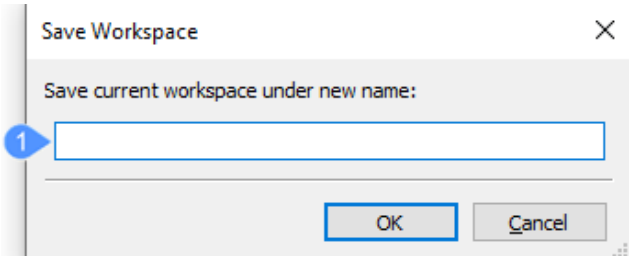
4.12.123. General tags

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

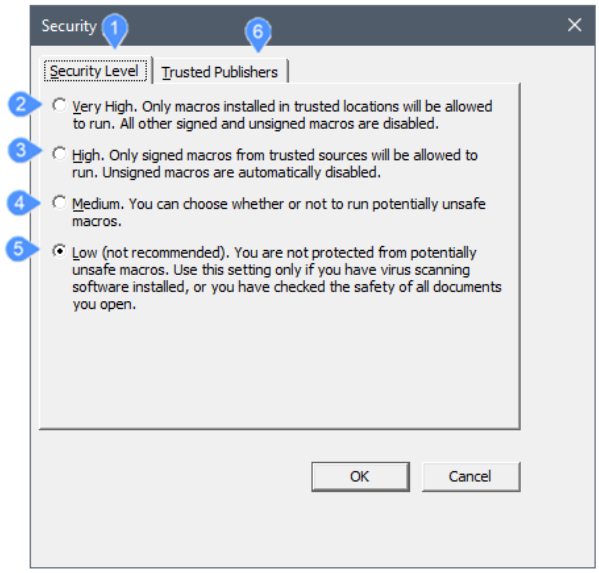
1

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

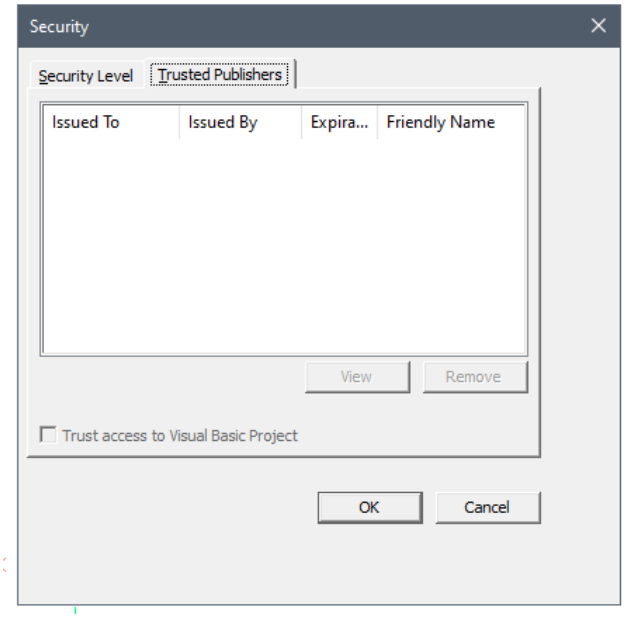
4

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

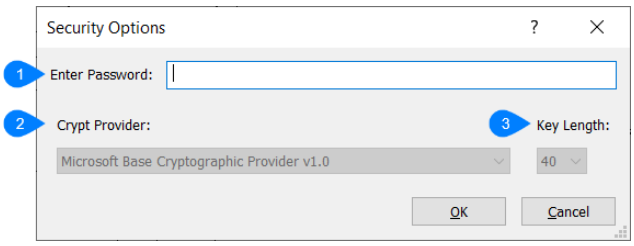
Removes the selected VBA macro from the list.

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

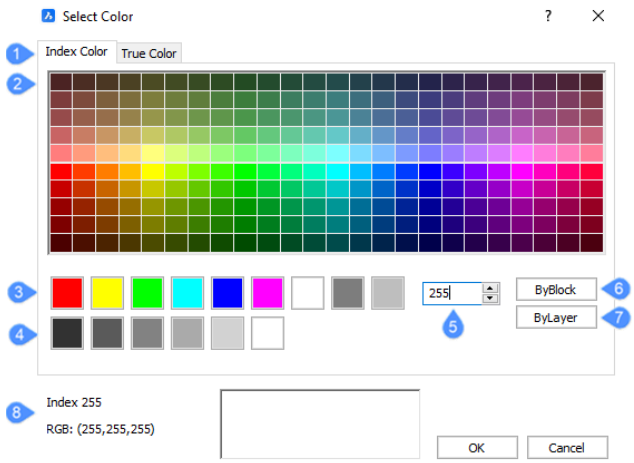
Select a length for the encryption key. The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.

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.

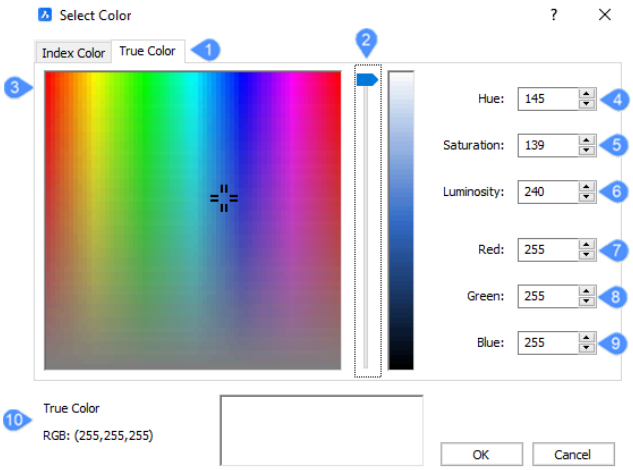
ByBlock
Sets the color to ByBlock in which entities take on the color defined by their block.

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).

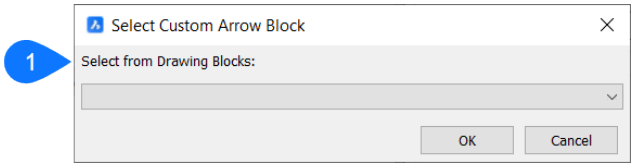
Green
Specifies the amount of green in the color, ranges from 0 (no green) to 255 (full green).

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.



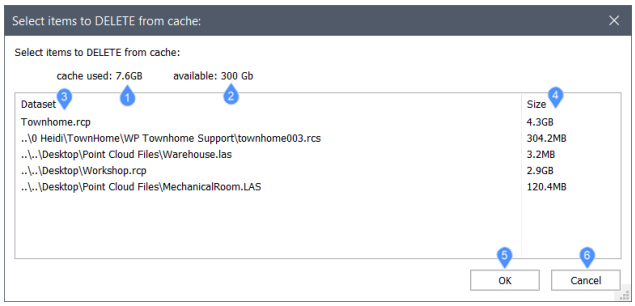
1 Select from Drawing Blocks

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.

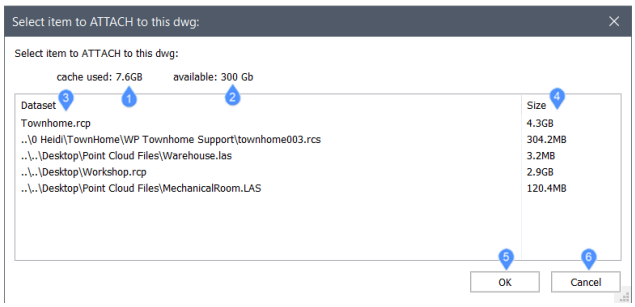
4.12.129. Cancel

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.

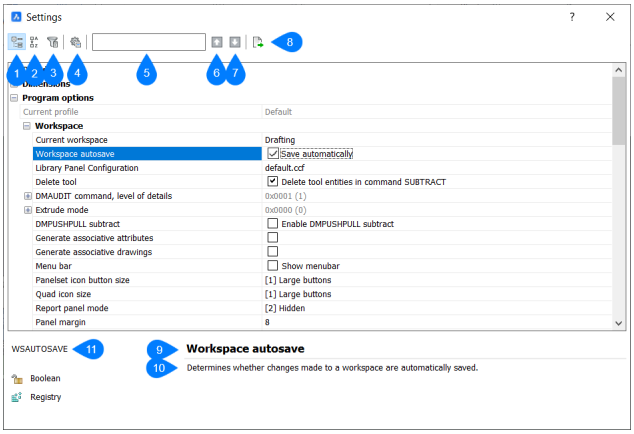
4.12.130. Cancel

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

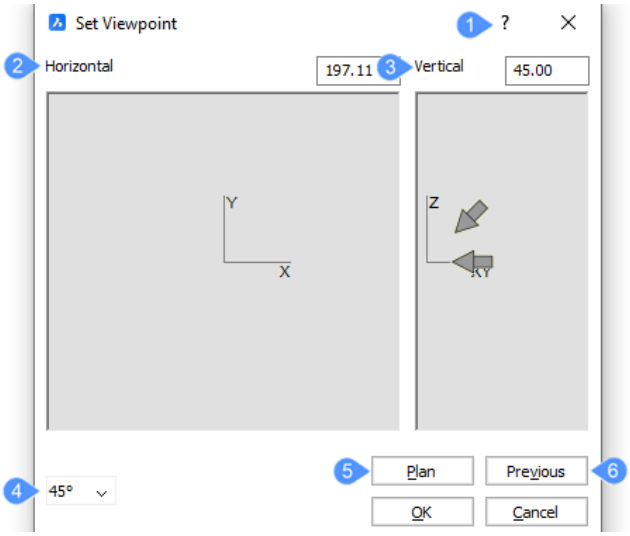
11

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.

Uwaga: 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

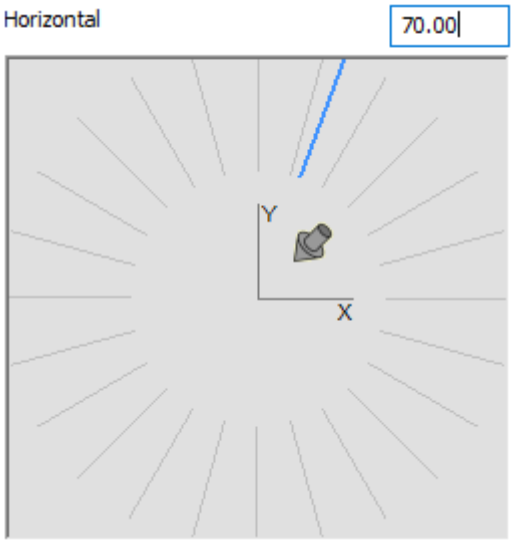
4.12.132. Command reference

1

Opens the Bricsys Help article about the DDVPOINT command.

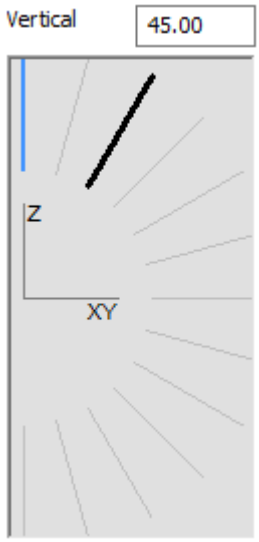
4.12.132. Horizontal
2

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.

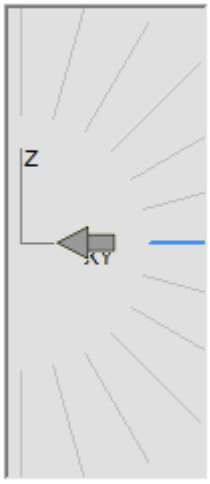


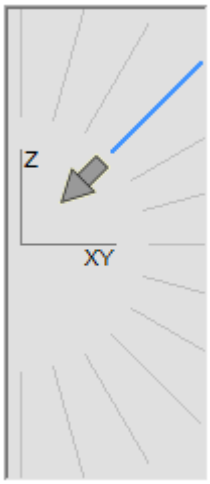
4.12.132. Vertical
3

Specifies the vertical angle from the XY plane. Click a preset angle, or else type a value in the Vertical field.

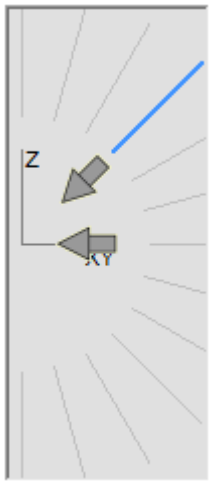


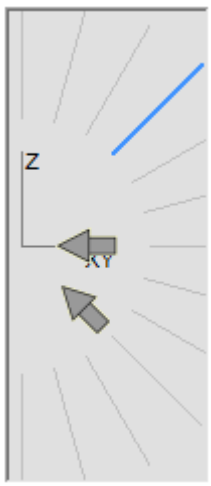
Click in the inner half circle. (1) View direction is horizontal. (2) View direction is 45° down (not available if PresetAngle = 45°).

(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.

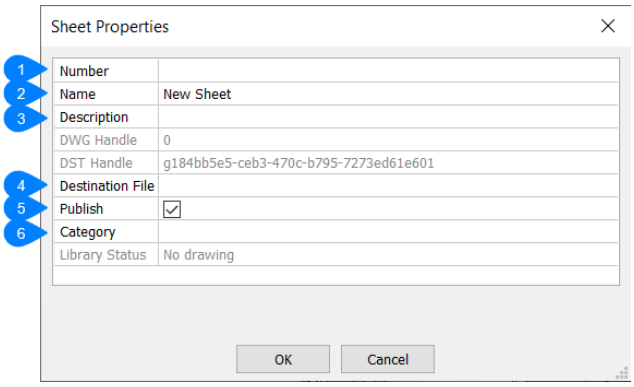
4.12.132. Previous

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

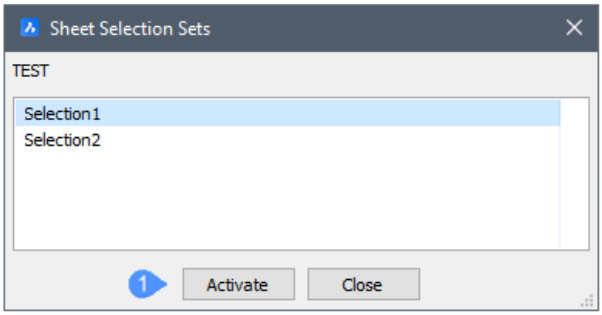
6

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

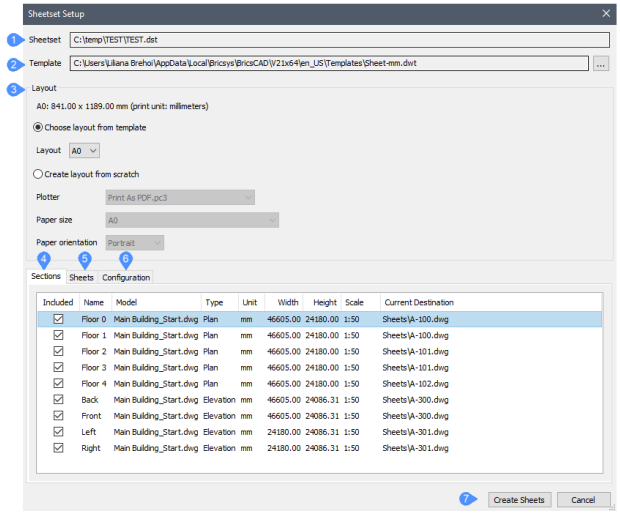
1

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

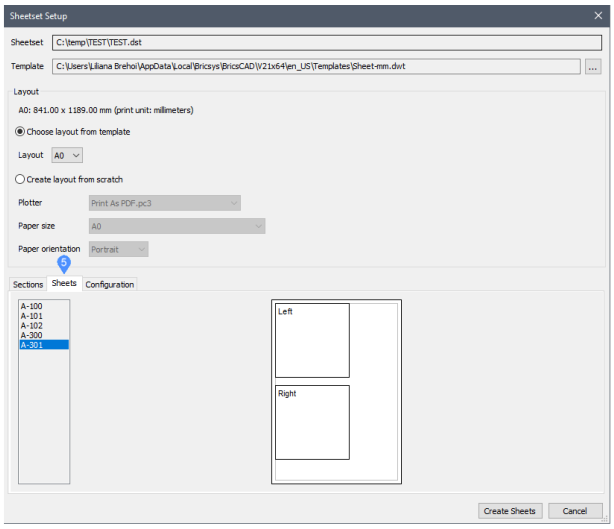
Lists all sections. For each section with an empty **Current Destination** field, a viewport and a sheet view is generated.

- **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.

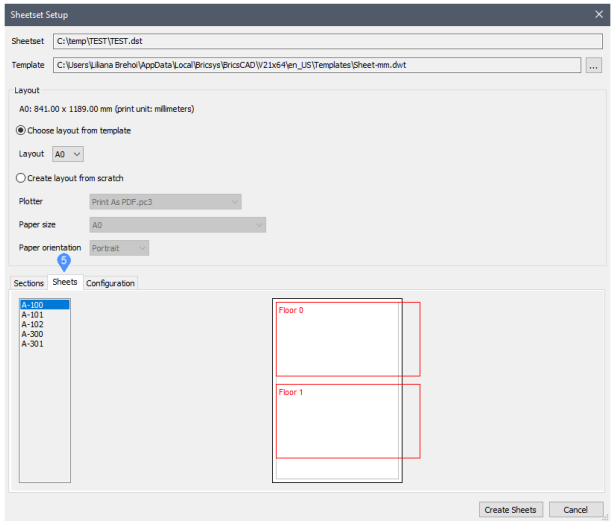
4.12.135. Sheets Tab

5

Shows how the viewports will be displayed in the layout.

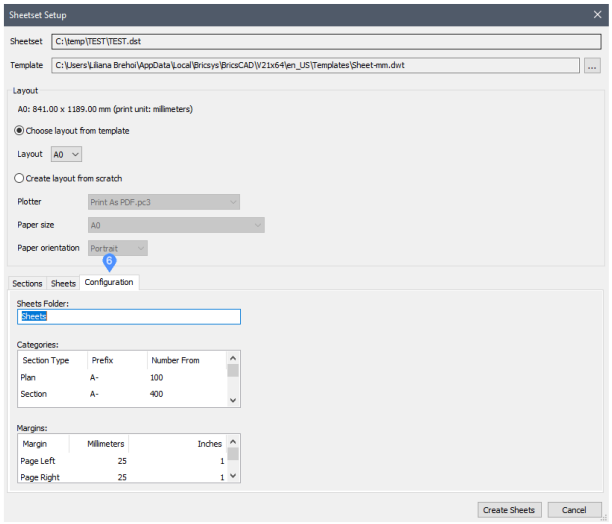


Uwaga: When the viewport is larger than the layout, it displays in red.



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.

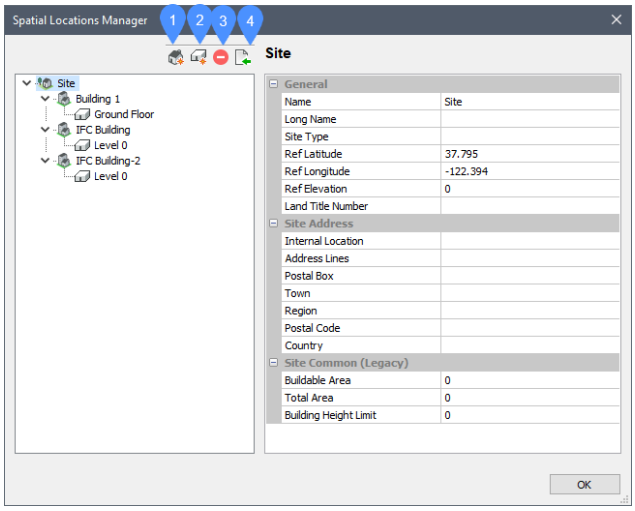
4.12.135. Create Sheets

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

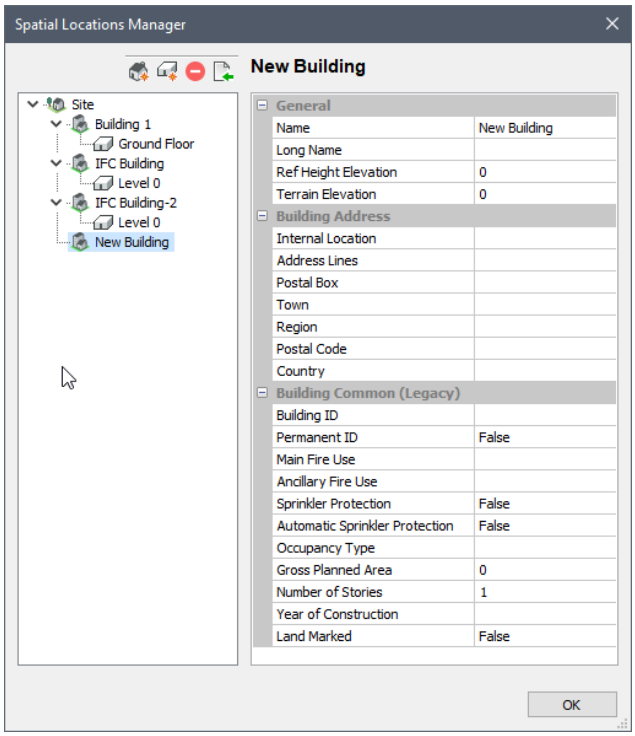
4 Import

4.12.136. New Building

1

Adds a new building to the model.

Fill out the properties grid:

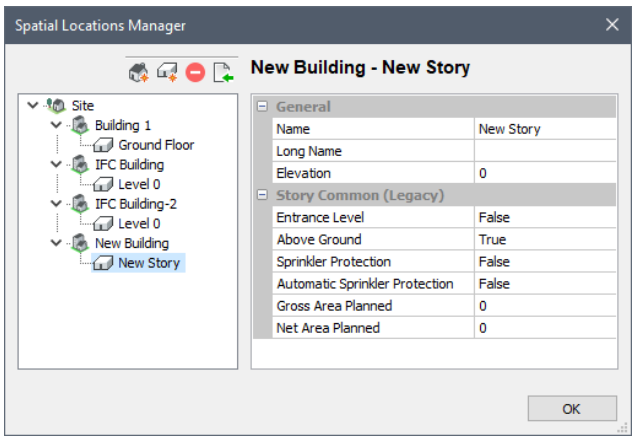


4.12.136. New Story

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

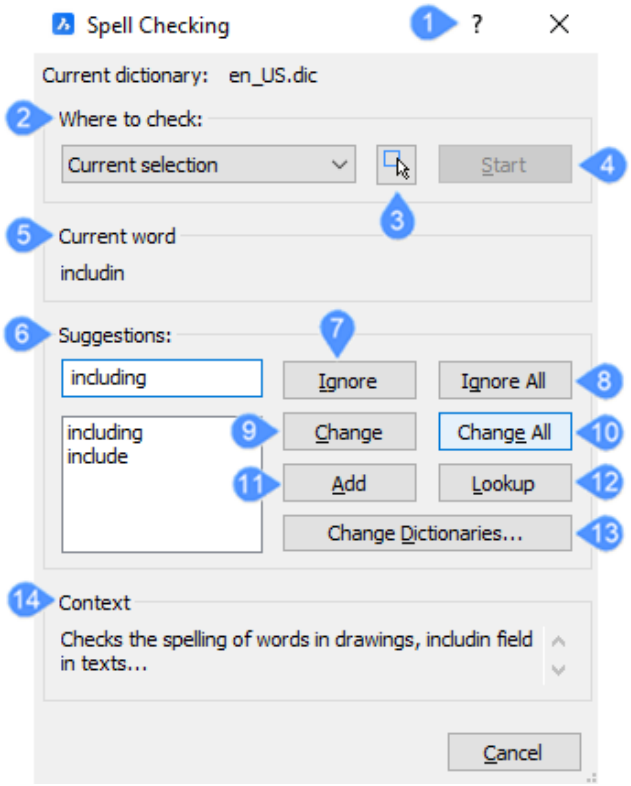
4

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.

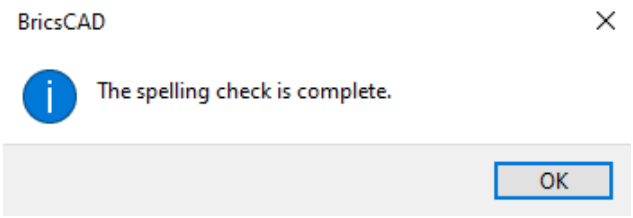
4.12.137. Change Dictionaries...
13

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.

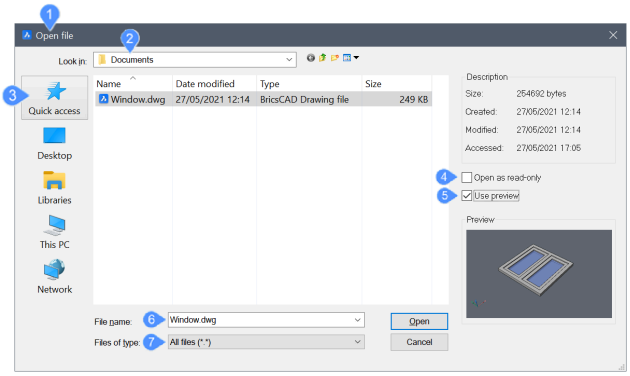
When the spelling check is completed, the following message box appears:



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

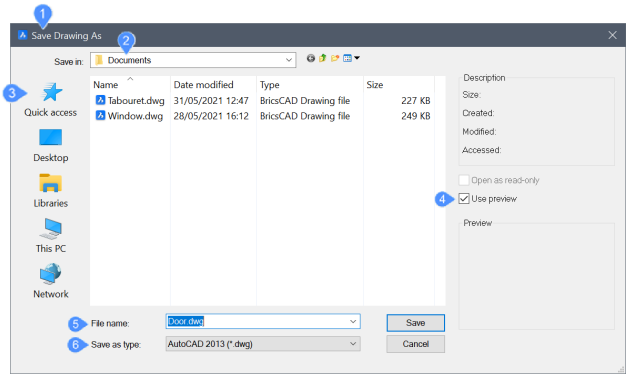
7

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.

Uwaga: 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.

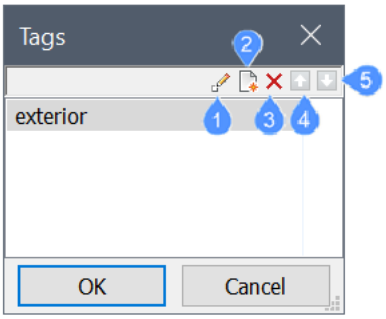
4.12.139. Files of type

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.

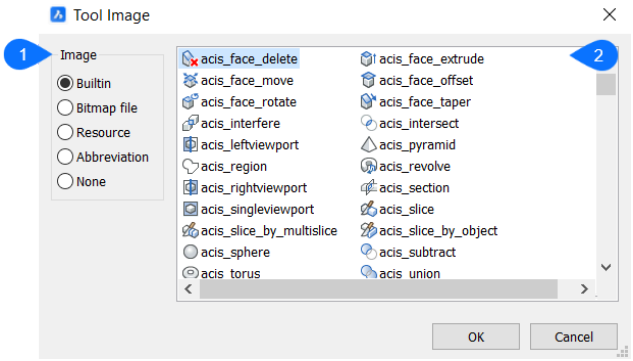
4.12.140. Move down

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.

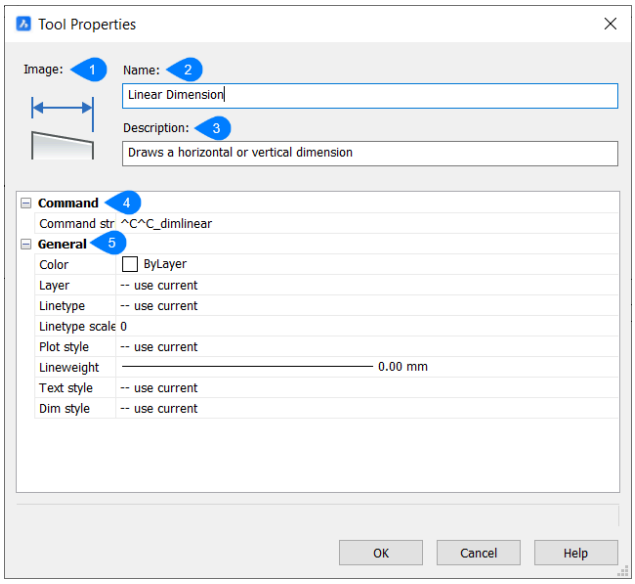
None
No image or abbreviation is used for the tool.

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

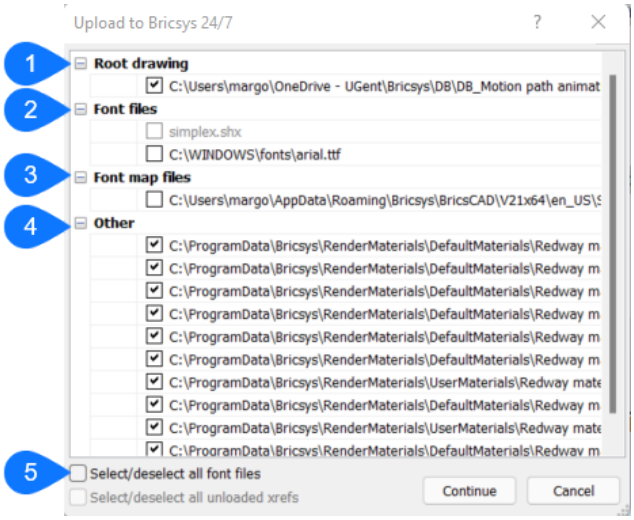
5

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

Uwaga: 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.

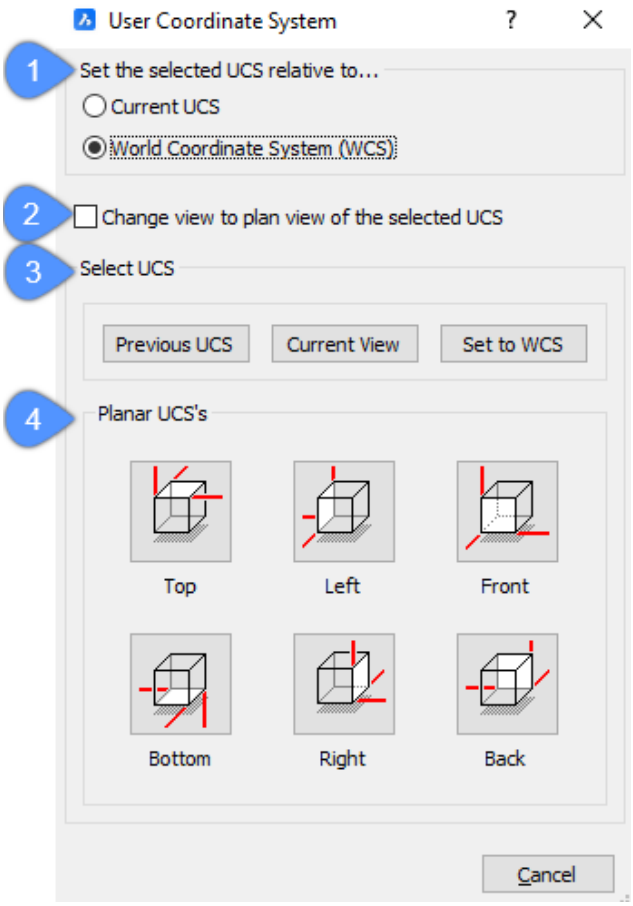
4.12.143. Select/deselect all font files

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.

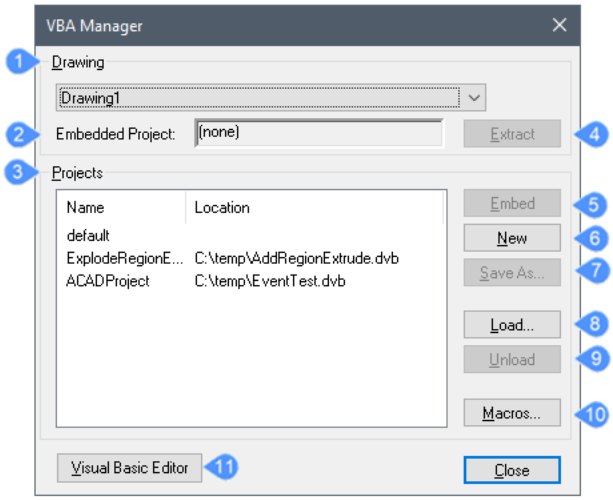
- **Top**: Changes the UCS to match the top view.
- **Left**: Changes the UCS to match the left view.
- **Front**: Changes the UCS to match the front view.
- **Bottom**: Changes the UCS to match the bottom view.
- **Right**: Changes the UCS to match the right view.
- **Back**: Changes the UCS to match 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

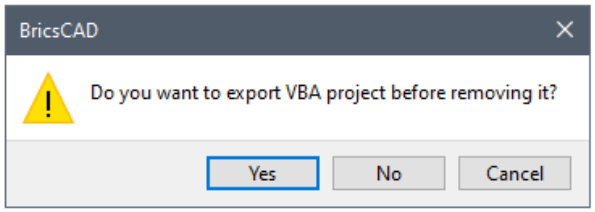
3

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.

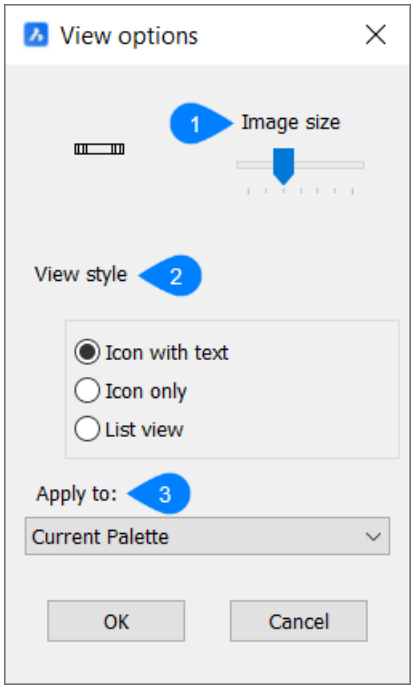
4.12.146. Visual Basic Editor

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

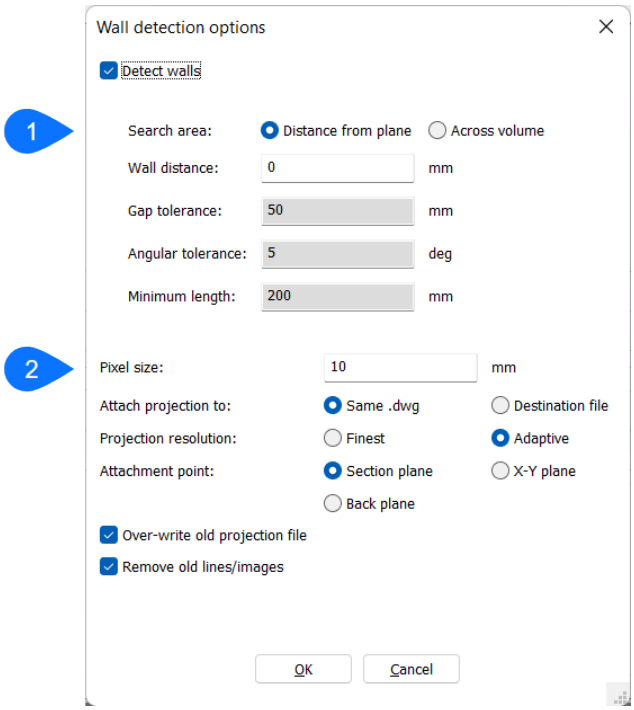
Displays the tools in a grid view with their icon only.

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

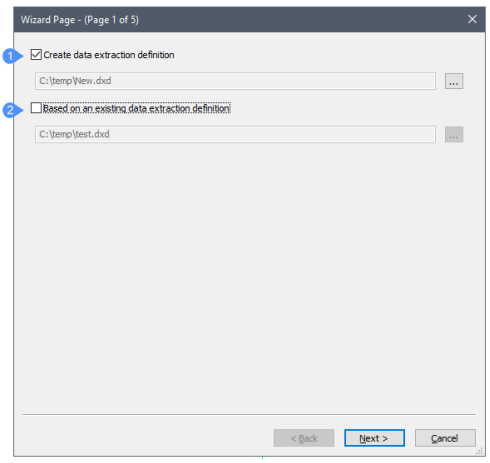
  Uwaga: The section can be edited in Top view as a regular 2D drawing. This is practical for vertical sections.

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.

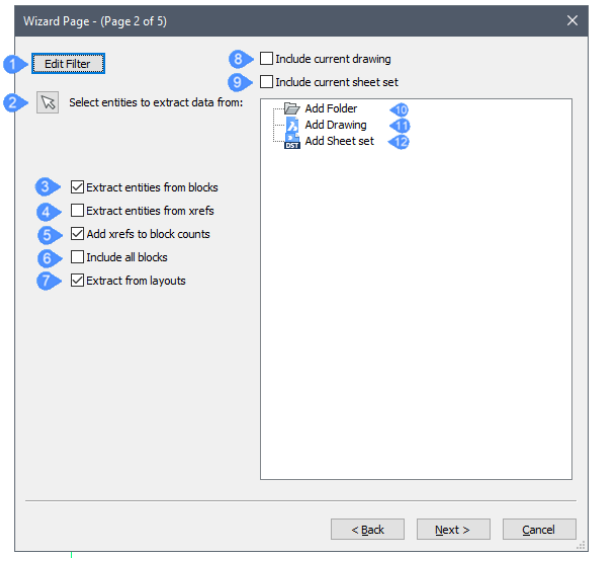
4 Click the Open button.

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.

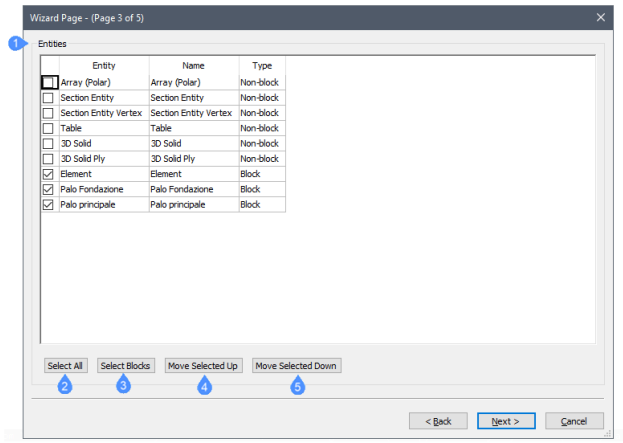
Add Folder
Adds a folder of drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **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

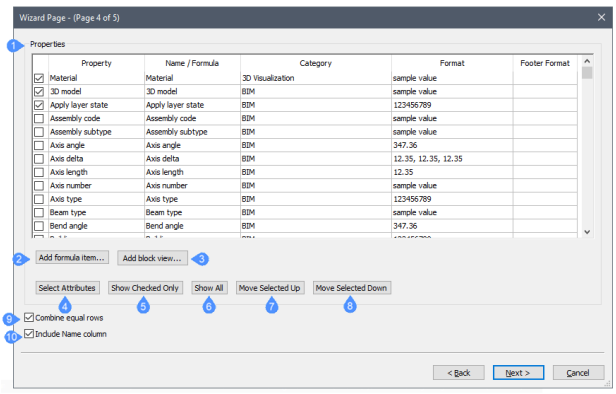
Move the selected item(s) up the list.

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.

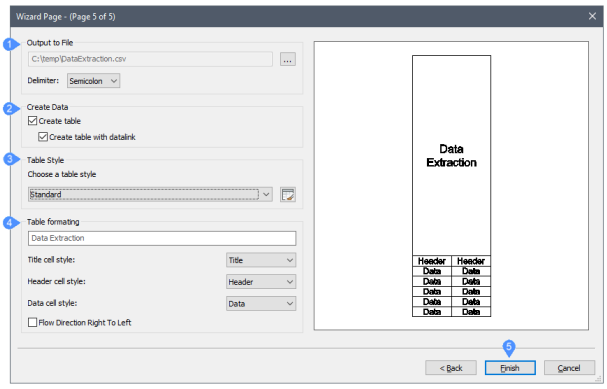
Move Selected Down
Move the selected item(s) down the list.

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

  Uwaga: 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.

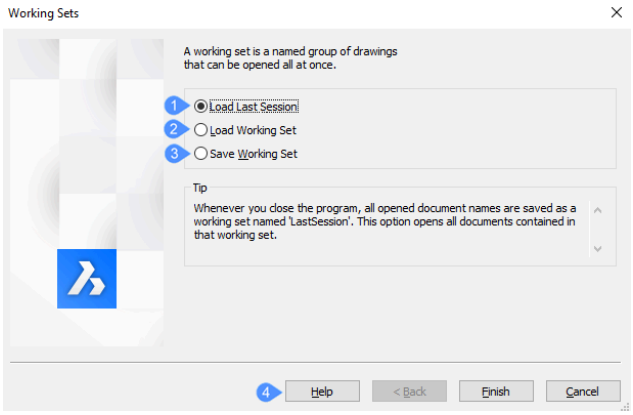
- Title cell style: sets the title cell style.
- Header cell style: sets the Header cell style.
- Data cell style: sets the Data cell style.
- Flow Direction Right to Left: indicates the flow direction from right to left.

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.

Uwaga: 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.

Uwaga: Delete removes a named working set. Redefine (overwrites) an existing named working set.

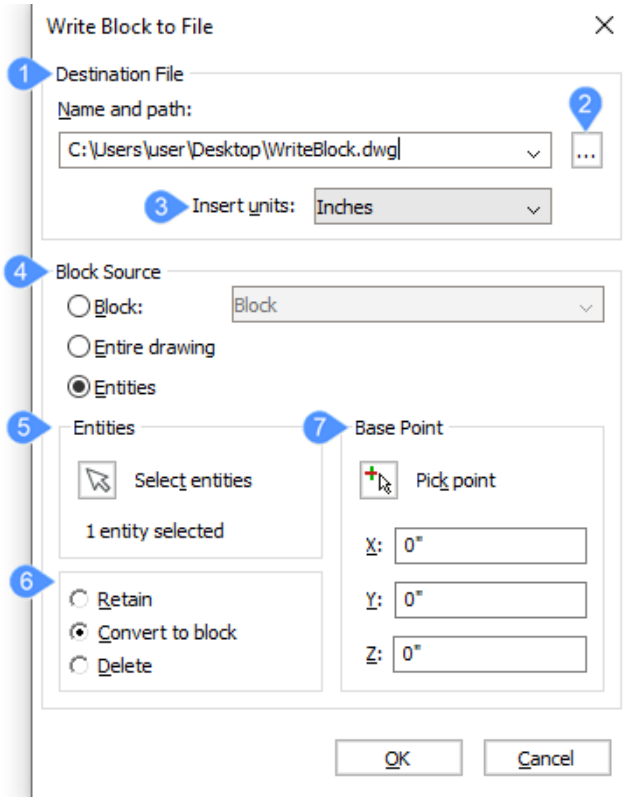
4.12.151. Help

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.

  Uwaga: 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. Managing drawings

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.

Uwaga: 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.

Uwaga: 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.

Uwaga:

- 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.

Uwaga:

- 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.

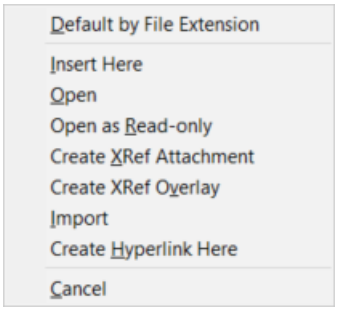
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(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.

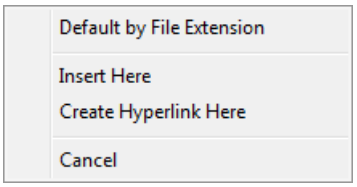
5.5.1 Using drag-and-drop to open a drawing

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.

Uwaga:

- 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.

Uwaga: 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 #.

Uwaga:

- 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. Rysowanie precyzyjne

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.

Uwaga:

- 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.

Uwaga:

- 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 2D constraints

Parametric constraints are used to maintain relationships and control dimensions of 2D geometry.

There are two types of constraints:

- **Geometrical**constraints control relationships between entities.

- **Dimensional**constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

A dimensional constraint can use a value or a formula, with other dimension constraints or parameters. Dimensional constraints can be edited in the Parameters and Constraints panel and in the Mechanical Browser panel.

Uwaga:

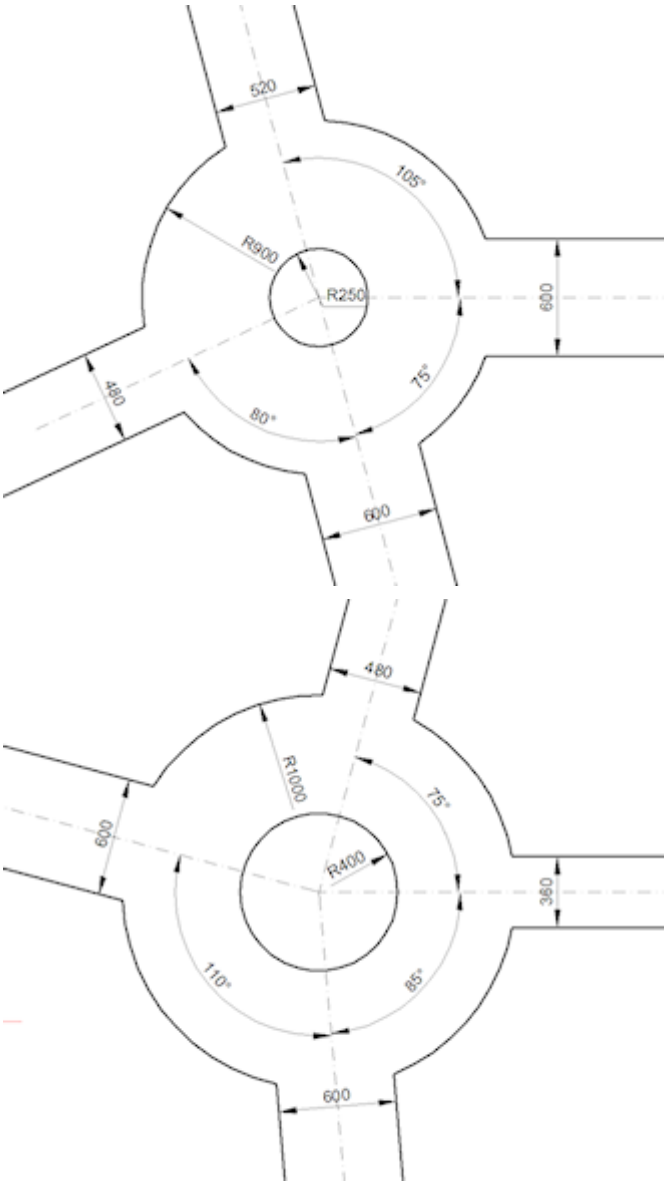
- It is recommended to first apply geometric constraints, then dimensional constraints.
- ThePARAMETERCOPYMODEsystem variable controls how constraints and related parameters are processed when constrained entities are copied.
- The PARAMETRIZE2D command automatically creates geometrical and dimensional parameters.

When using constraints a drawing can be in one of three states:

- **Underconstrained**: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.
- **Fully constrained**: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.
- **Overconstrained**: one or more constraints violate or contradicts other constraints.

Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constrains.



Rysunek 1: Two versions of the same fully constrained drawing representing a roundabout

6.4.1 Commands and toolbars

2D parametric constraints tools are located on the**2D Constraints**toolbar and the **Parametric** ribbon tab.

Tools to control the display of constraint bars and geometrical constraints are located on two flyout toolbars: **Dimensional Constraint**and **Geometric Constraint**.

Command	Description
	Allows to set dimensional constraints to be displayed or hidden.

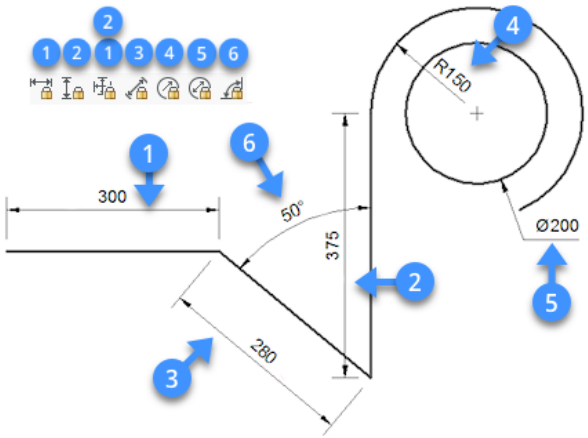
Command	Description
	Applies geometric relationships between entities and/or valid constraint points on entities.
	Constrains points and entities at a fixed position.
	Applies a coincident geometrical constraint to two points or constraints a point to an entity.
	Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
	Forces lines or polyline segments to be collinear.
	Forces two lines or linear polyline segments to be parallel to each other.
	Constrains two lines or polyline segments to lie perpendicularly to each other.
	Constrains two entities to maintain a point of tangency to each other or their extensions. The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs.
	Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system.
	Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system.
	Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
	Constrains two entities or points to lie symmetrically with respect to a selected line.
	Constrains arcs and circles to the same radius, or lines and polyline segments to the same length.

Command	Description
	Controls the display of the constraint bar of entities. Constraint bars are always hidden when opening a drawing.
	Applies a dimensional constraint to an entity or between constraint points on entities. Converts associative dimensions to dynamic dimensions.
	Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
	Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.
	Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
	Constrains the distance between two points or the length of a line or polyline segment.
	Constrains the radius of a circle, an arc or an arc polyline segment.
	Constrains the diameter of a circle, an arc or an arc polyline segment.
	Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
	Converts an associative dimension to a dimensional constraint.
	Removes all dimensional and geometrical constraints from an entity.

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).



Uwaga: 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.

Uwaga: 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.

Uwaga:

- 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:

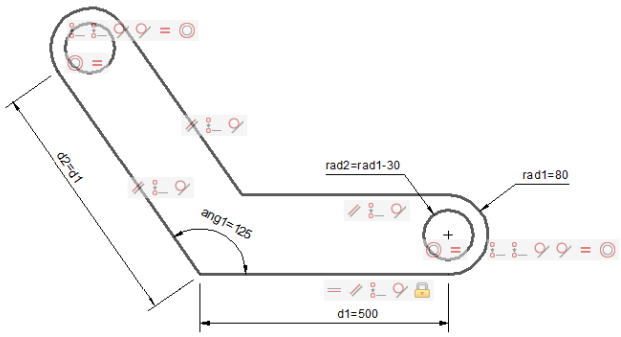
- distance: d1, d2, d3, ...
- radius: rad1, rad2, rad3, ...
- diameter: dia1, dia2, dia3, ..
- angular: ang1, ang2, ang3, ..

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)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)

Function	Syntax
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)

Function	Syntax
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.

Uwaga:

- 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.

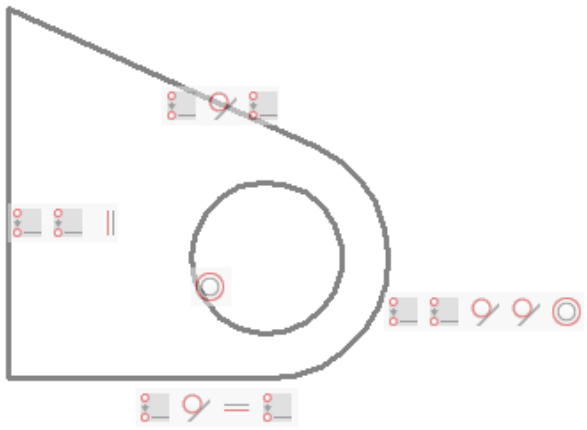
6.6.1 Creating geometric constraints

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.

Uwaga:

- 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:

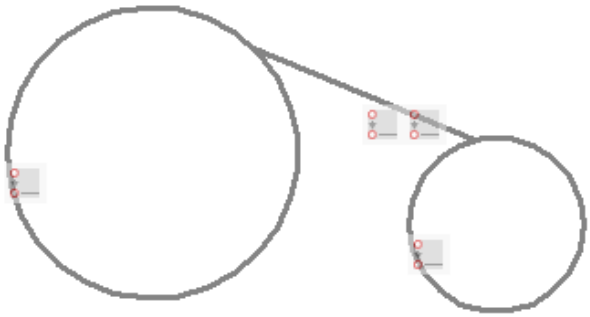
- Click the**Delete 2D Constraints**tool.
- Launch the DELCONSTRAINT command.

You are prompted: Select entities to delete all constraints:

2 Selectthe entities you want to delete the constraints from.

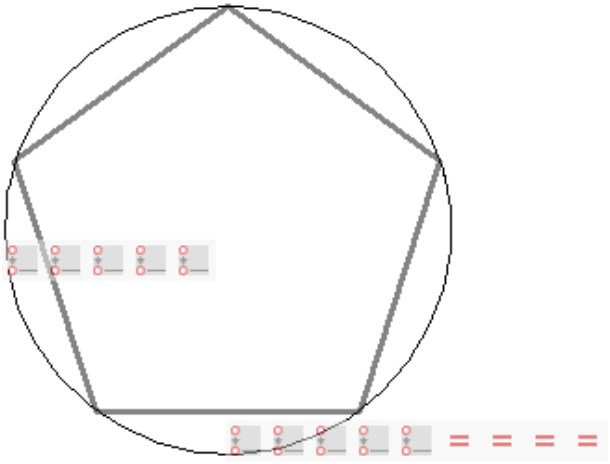
3 Right click to stop selecting entities and delete all constraints from the selection set.

6.6.6 Some examples

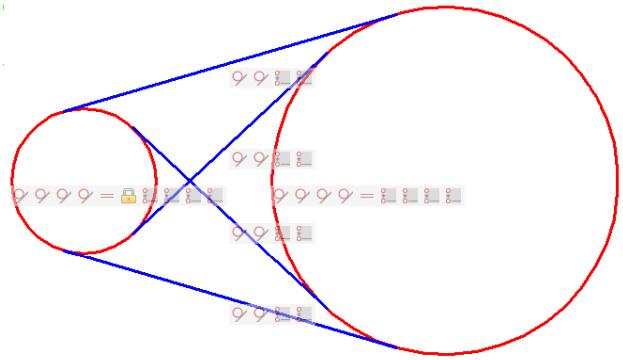


Coincident constraints between the endpoint of a line and two circles.

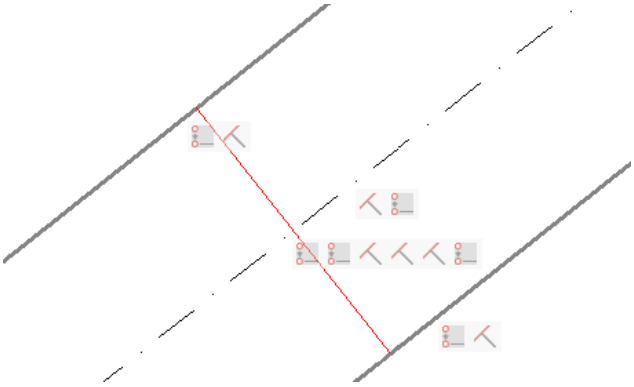
If the endpoints or center points of entities already coincide, the**Autoconstraint**option of theGCOINCIDENTcommand automatically applies coincident constraints to such points.



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

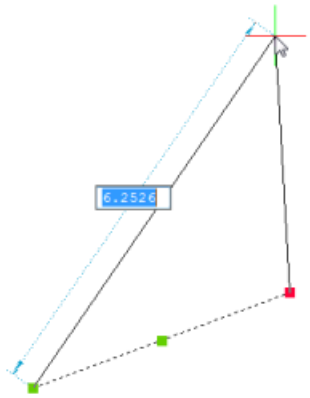
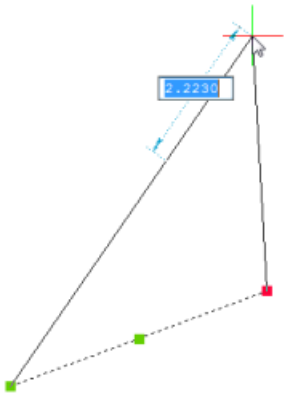
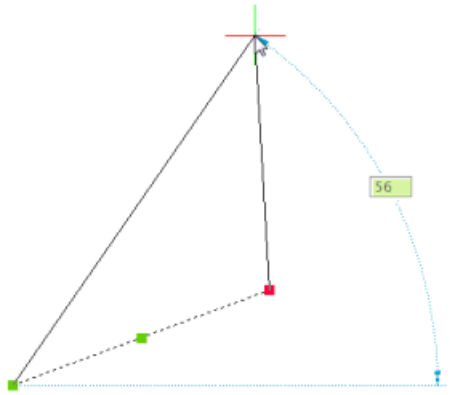
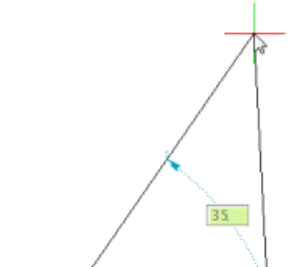
Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such, dynamic dimensions help you to focus on the drawing area.

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.

Uwaga:

- 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.

Uwaga:

- 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.

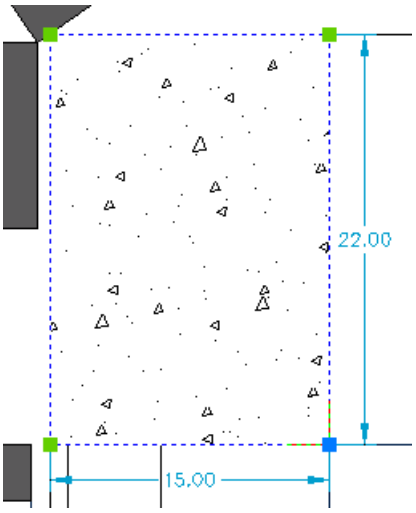
1 Select the entity or entities.

All grips on the selected entities display.

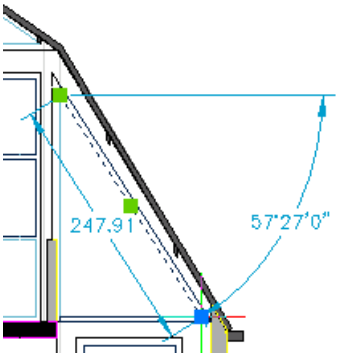
2 Hover the cursor over a grip.

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

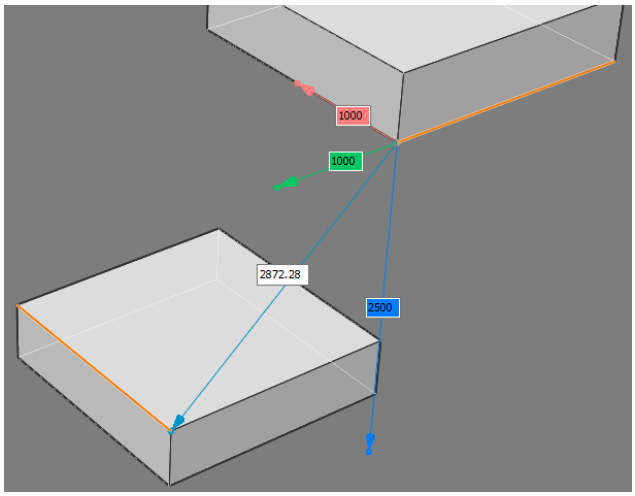
If the**Tracking dynamic dimensions**option of theDYNMODEsystem variable is set, dynamic dimensions display whenentity snap tracking pointsortemporary tracking pointsare acquired.

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.

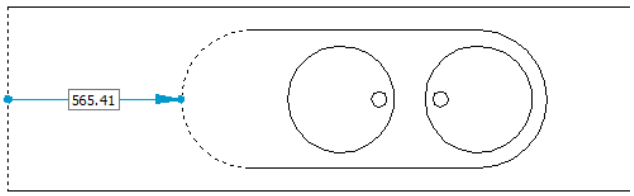
4 (option) Click a distance field and type a value.

The second entity is moved.

* Press and hold the Ctrl-key to select a subentity.

You can use the Nearest Distance feature to:

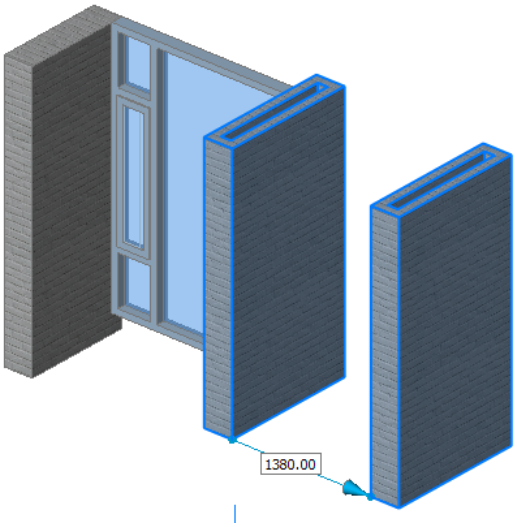
- Measure the distance between two entities or subentities.
- Adjust the distance between a (sub)entity and a block.



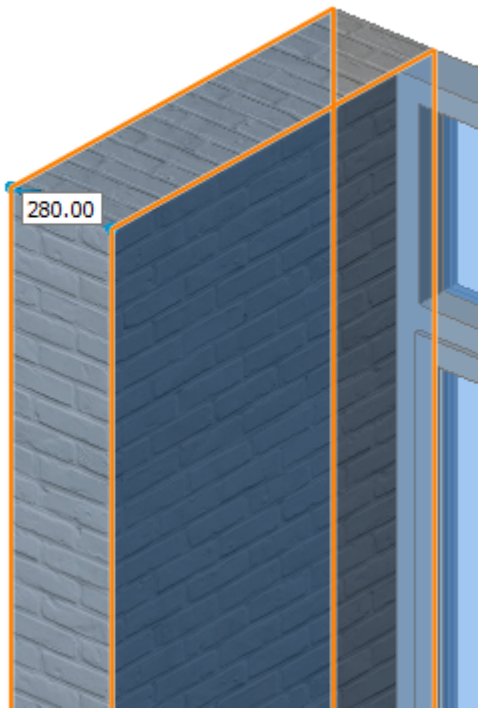
- Modify the dimensions of a rectangle.



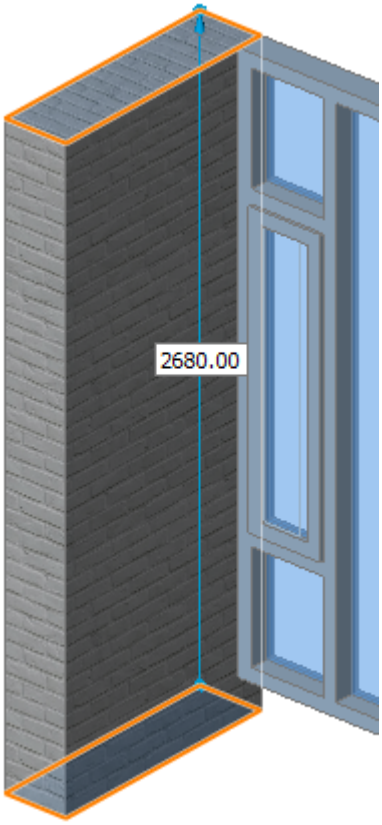
- Adjust the distance between 3D solids.



- 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**.

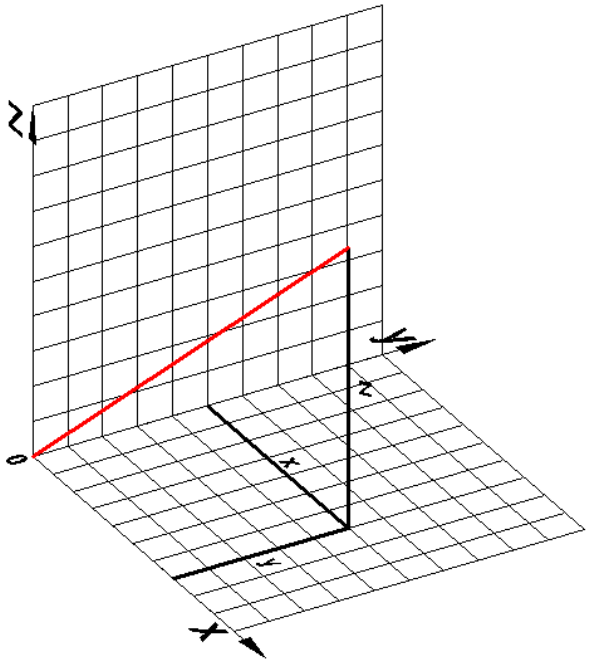
WhenDynamic Dimensionsare active and you type a value in the**Length**field and add a comma (,) the content of the**Length**field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the 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**.

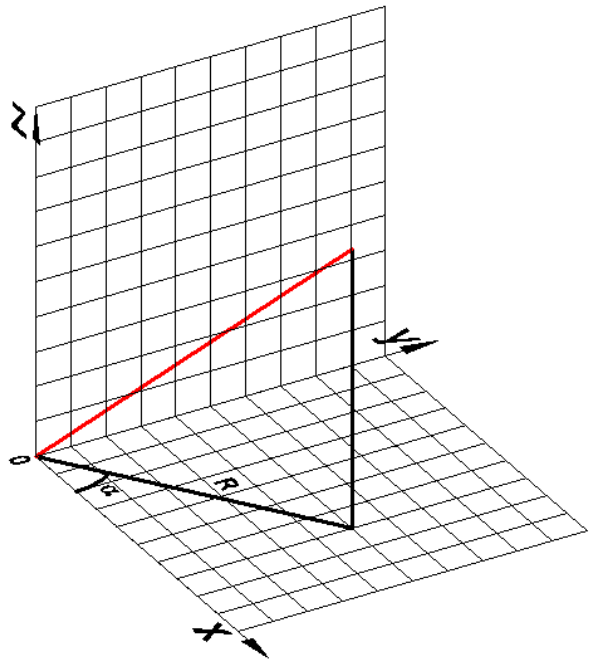
6.9.4 Working with cylindrical coordinates

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.

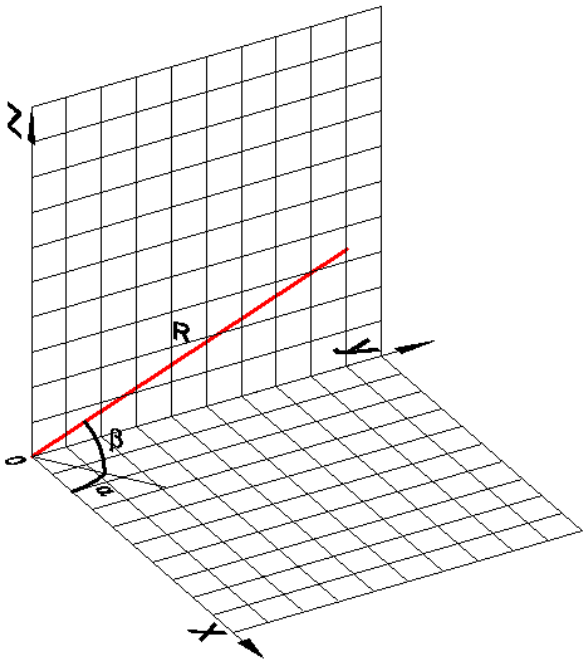


6.9.5 Working with spherical coordinates

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)

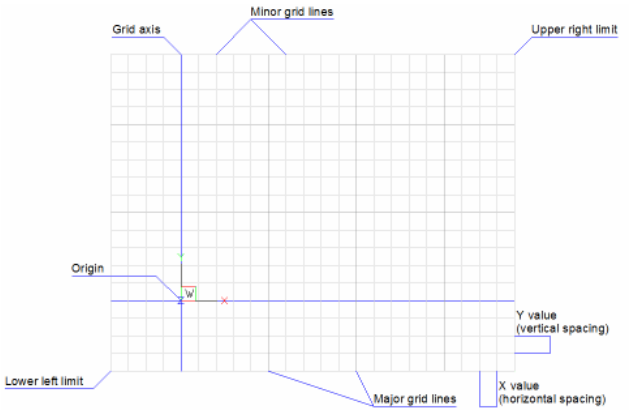


6.10 Snap and grid

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.

Uwaga: 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

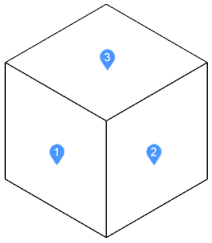
- Execute the LIMITScommand to set the drawing limits.
- Click the**Drawing Limits**item () in the Settings menu to toggle theLIMCHECKsystem variable on/off.

  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.

Uwaga: 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.

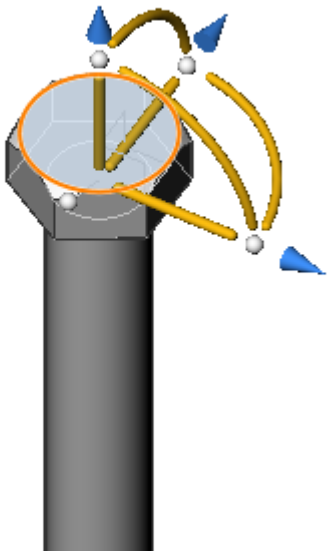
The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active ifSNAPTYPE = 2. This system variable overrides theSNAPMODE system variable.

The AdaptiveGridStepSize user preference controls the smallest available step size; default = 4 pixels.

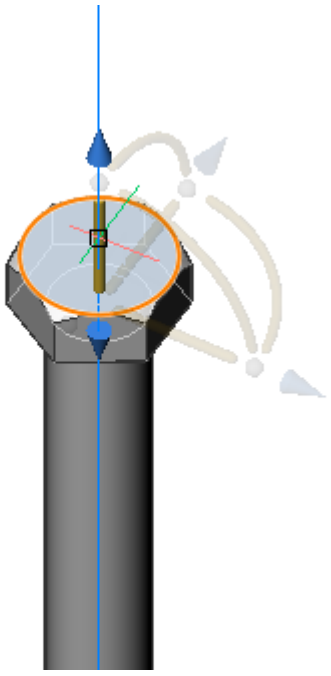
6.11.1 Using adaptive grid snap with the Manipulator

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.

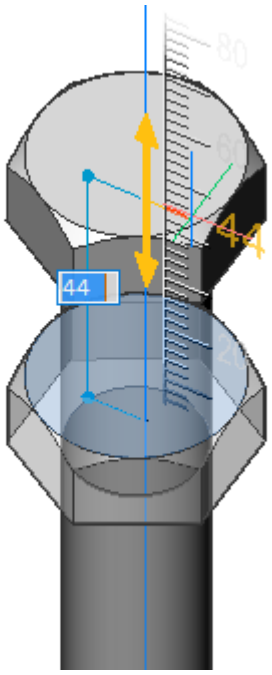


3 Click to select the axis, then move the cursor.

The Ruler displays.

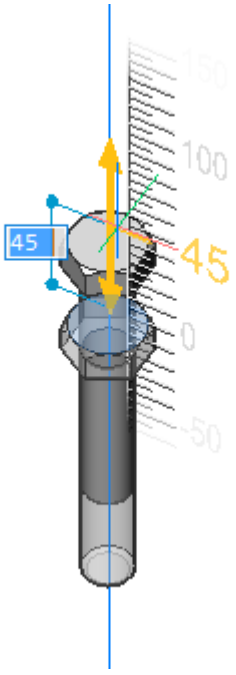
The current displacement distance displays in a dynamic field.

The grid snap resolution automatically adapts according to the zoom factor.



Rysunek 2: Ruler divisions at 2 mm

Rysunek 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.

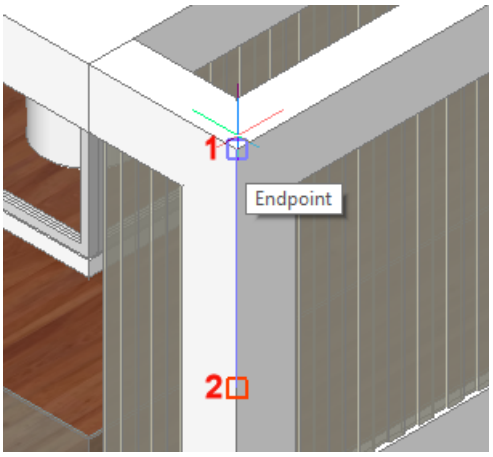
- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

Uwaga: 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).

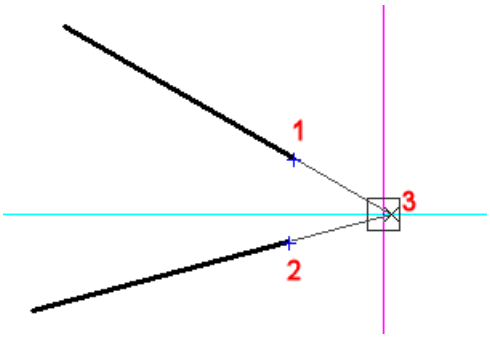
A small red cross indicates the entity is marked for extension.

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.

Uwaga: 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.

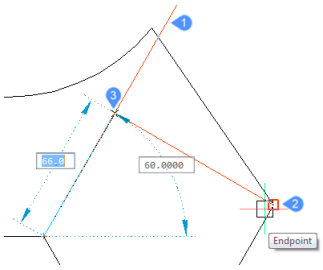
If bitcode 2 of the POLARMODE system variable is checked, the polar angles used inPolar Tracking(interval and additional angles) are also used inEntity Snap Tracking.

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.

Uwaga: 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.

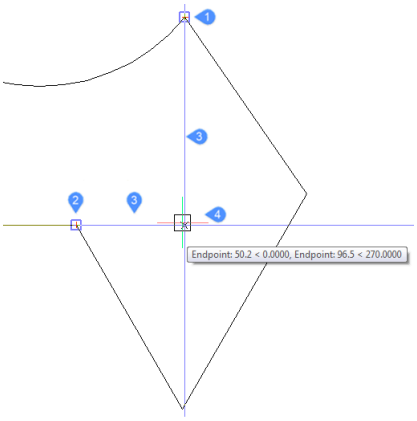
Prompts you: Start of line or [Follow] <Last point>:

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.

Uwaga:

- 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).

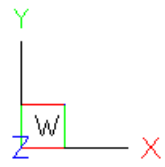
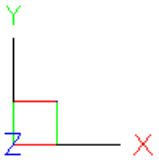
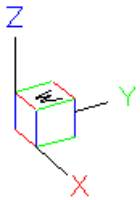
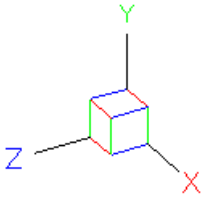
6.17.1 Controlling the display of the coordinate system icon

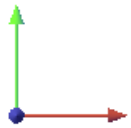
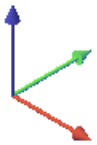
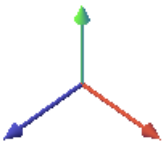
TheUCSICON and UCSICONPOS system variables control the display and location of the UCS icon:

- 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.

Uwaga: 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.

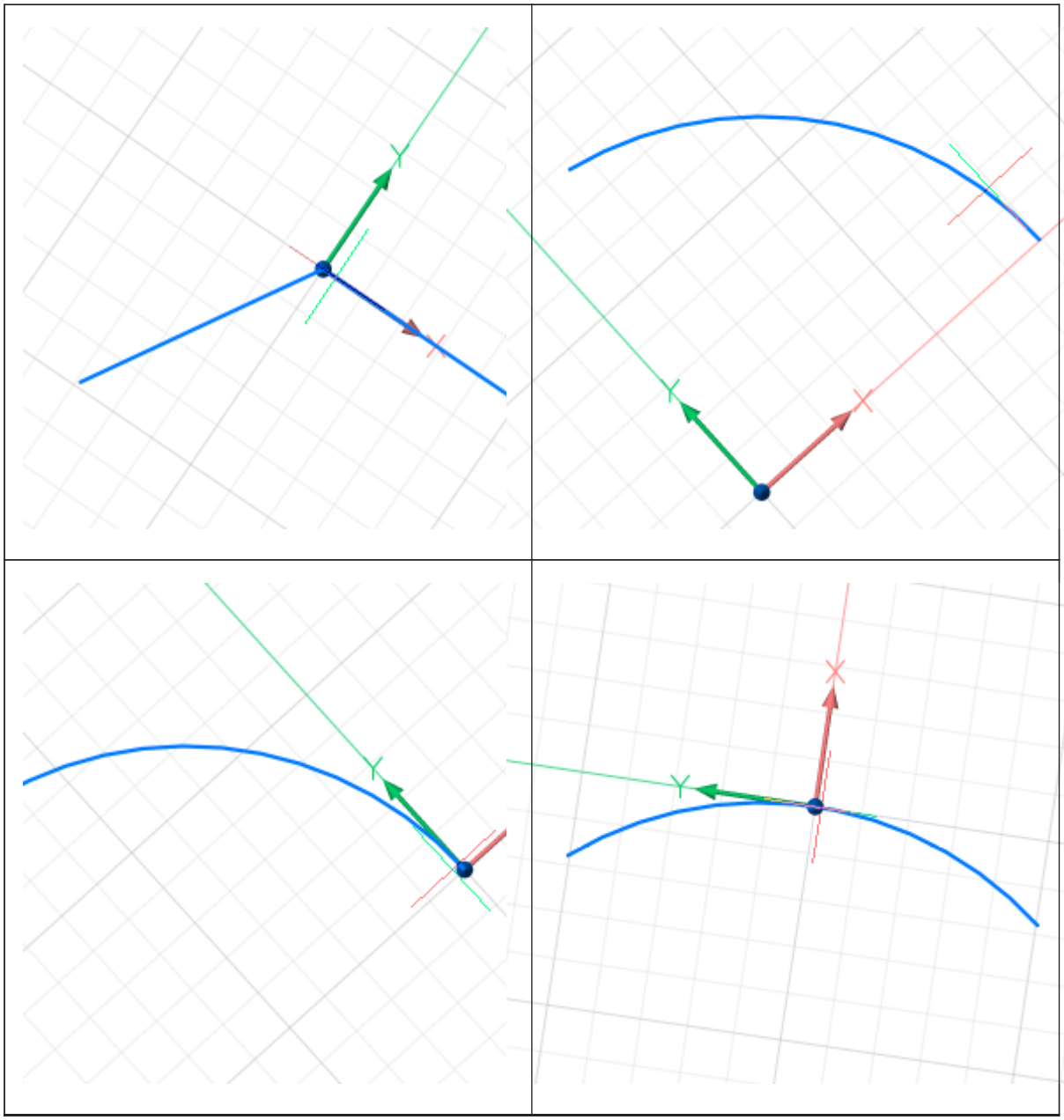
The entity or face highlights.

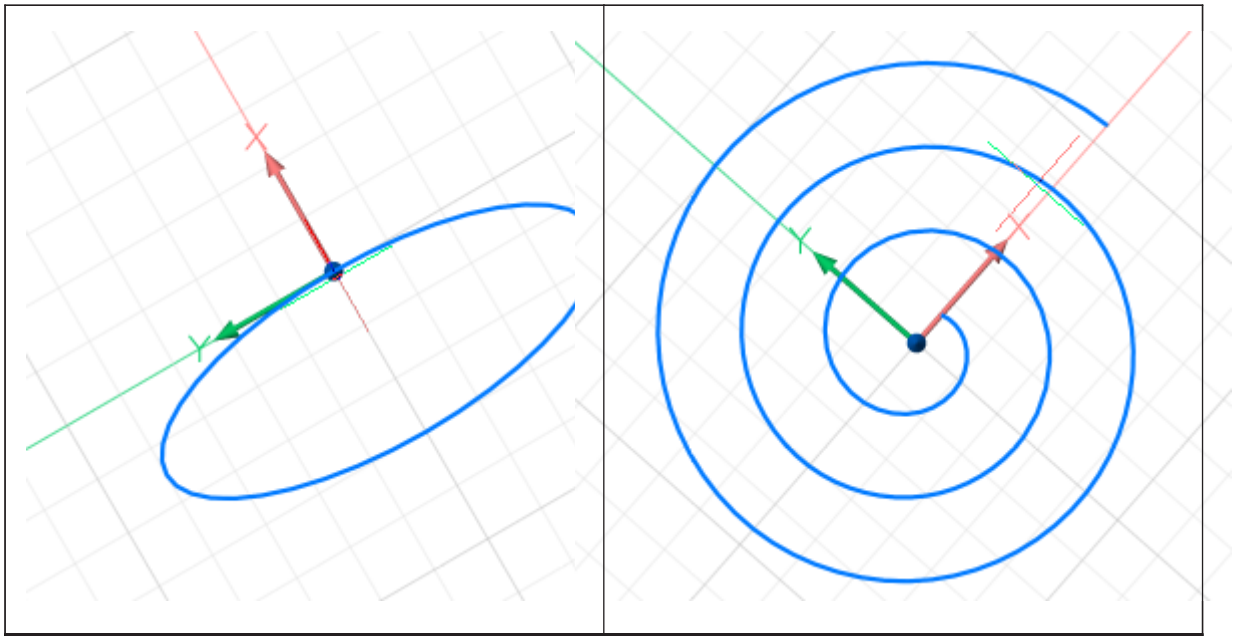
If GRIDMODE is On, the grid is aligned with the hovered entity or face.

3 IfUCSICON= 3 (On at Origin), the UCS icon displays.

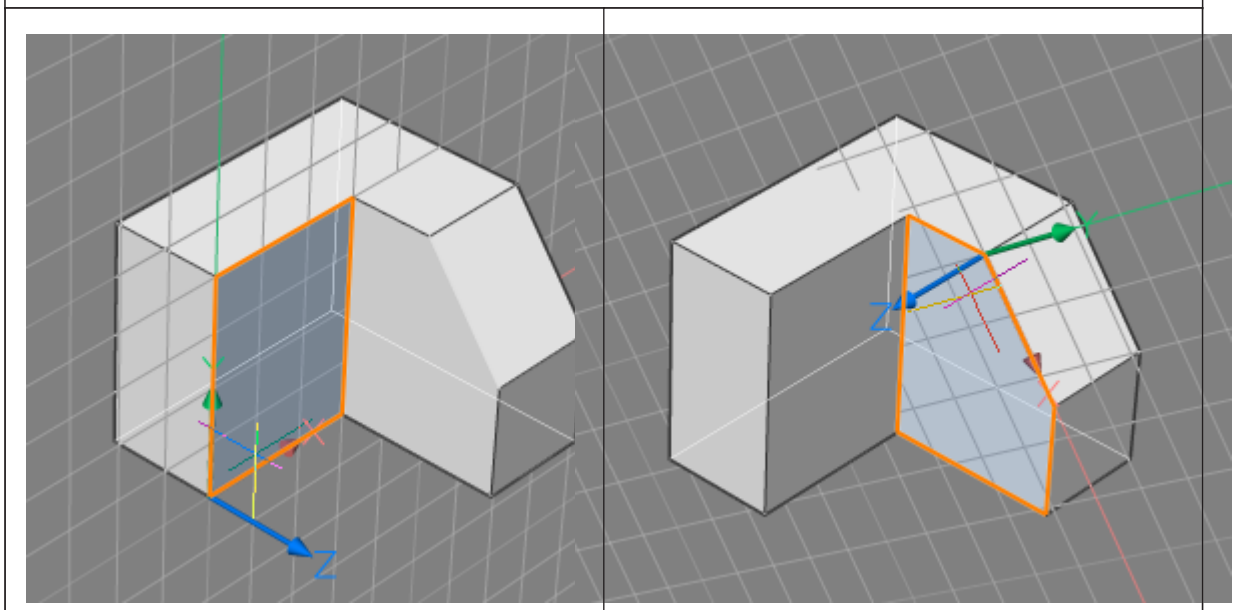
4

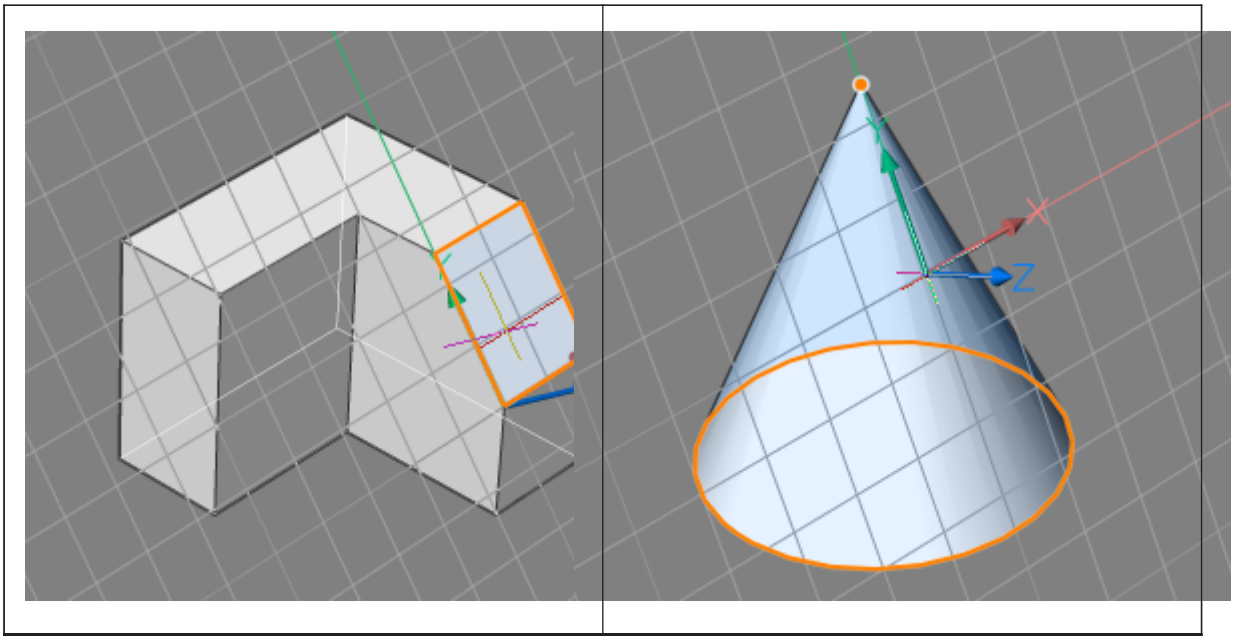
Dynamic UCS on other entities





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.

Uwaga: 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.

Uwaga: 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.

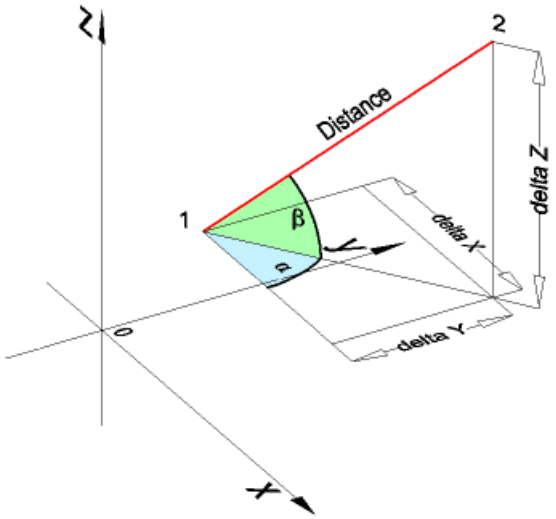
When Dynamic Input (DYN) is On, the distance displays in thedynamic dimensionfield.

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.

Uwaga: 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.

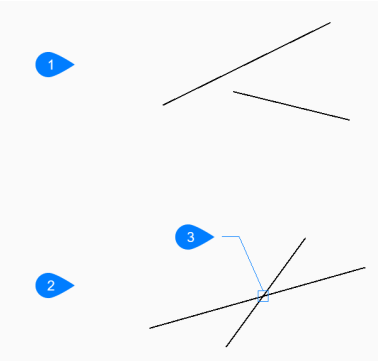
BricsCAD® reports in the command line:

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.

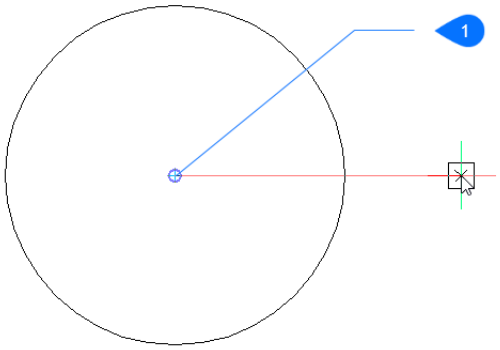
6.20.2 Center entity snap

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

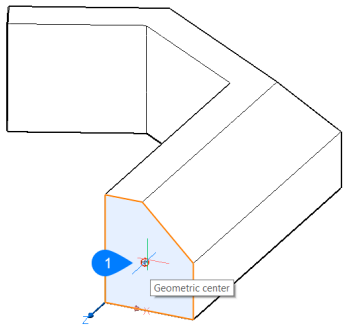
- Elliptical 3D faces



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.



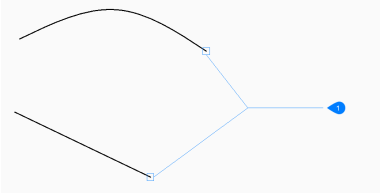
1 Geometric center entity snap.

6.20.4 Przyciąganie do punktu końcowego

Przyciąganie elementu Koniec umożliwia przyciąganie kursora do punktów końcowych elementów.

Łącznie z:

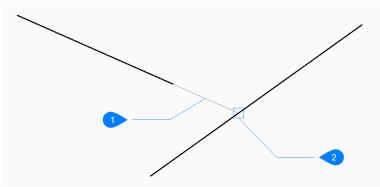
- Linie
- Łuki
- Segmenty polilinii
- Otwórz splajn
- Krawędzie 3D



1 Punkt końcowy elementu środkowego przyciągania.

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

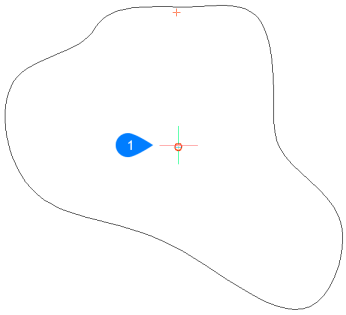
2 Extension entity snap.

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:

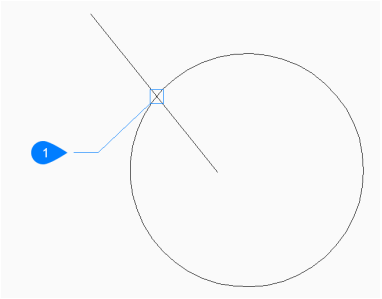
- Text
- Mtext
- Tables
- Attribute text
- Blocks
- External References
- PDF References
- Raster images



1 Insertion entity snap.

6.20.8 Przyciąganie do elementu przecięcia

Przyciąganie do elementów Przecięcie umożliwia przyciąganie kursora do punktu przecięcia elementów, które fizycznie się przecinają.



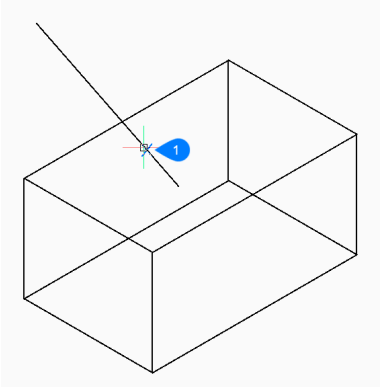
1 Przyciąganie do elementu przecięcia.

6.20.9 Intersection 3D entity snap

The Intersection 3D entity snap enables the cursor to snap to the intersection of faces with linear entities.

Including:

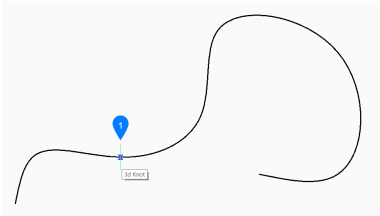
- Lines
- Polyline segments
- 3D edges
- Polar tracking lines
- Entity snap tracking lines



1 3D intersection entity snap.

6.20.10 Knot 3D entity snap

The Knot 3D entity snap enables the cursor to snap to knots on splines.



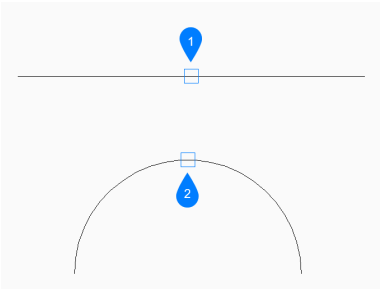
1 Knot 3D entity snap.

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

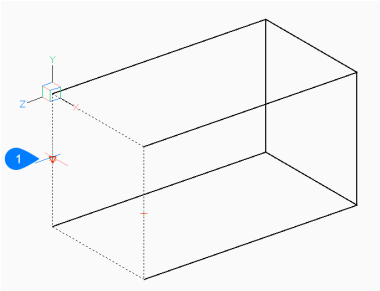


1 Midpoint (of line).

2 Midpoint (of arc).

6.20.12 Midpoint 3D entity snap

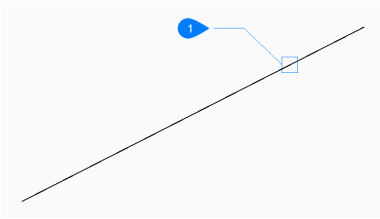
The Midpoint 3D entity snap enables the cursor to snap to the midpoint of 3D edges.



1 Midpoint 3D entity snap.

6.20.13 Nearest entity snap

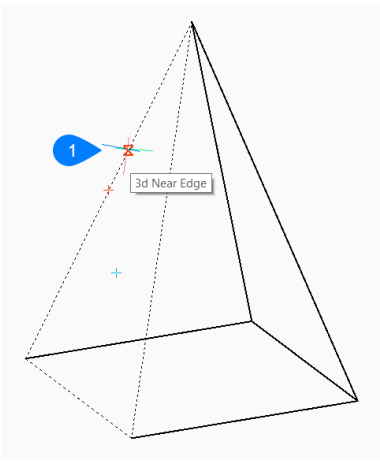
The Nearest entity snap enables the cursor to snap to 2D or 3D geometry nearest to the cursor.



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.



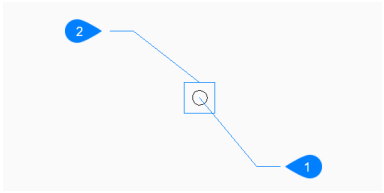
1 Nearest 3D entity snap.

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

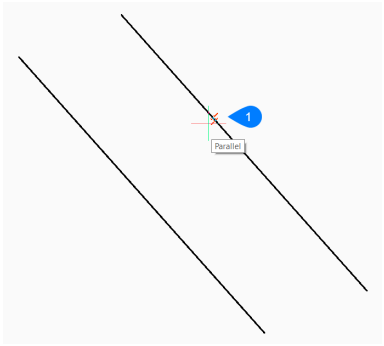


1 Point.

2 Node entity snap.

6.20.16 Parallel entity snap

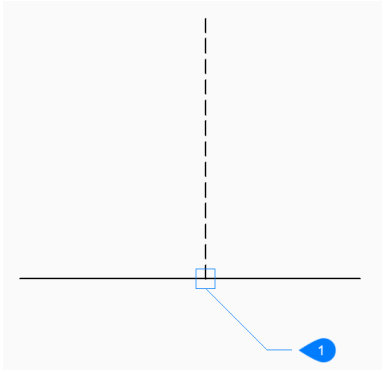
The Parallel entity snap enables the cursor to snap to a point parallel to an existing entity.



1 Parallel entity snap.

6.20.17 Perpendicular entity snap

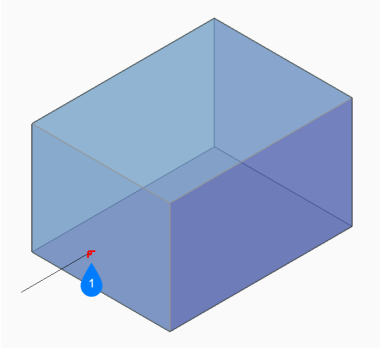
The Perpendicular entity snap enables the cursor to snap to a point perpendicular to an existing entity.



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.



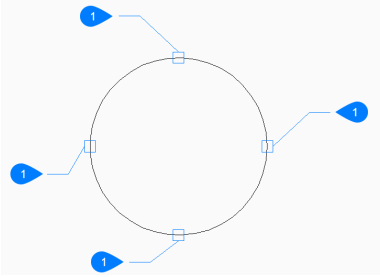
1 Perpendicular to a face 3D entity snap.

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.

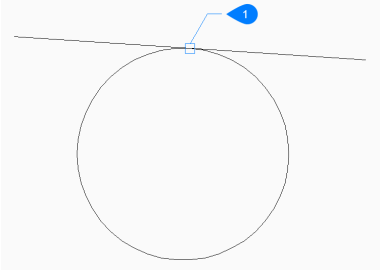
6.20.20 Tangent entity snap

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

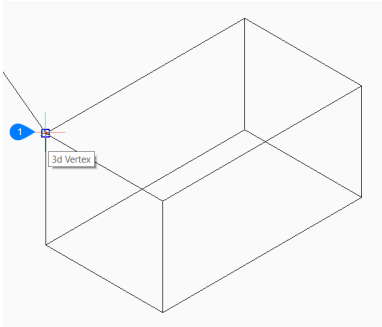
- Circular 3D faces
- Elliptical 3D faces



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. Przeglądanie rysunku

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.

Uwaga: 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.

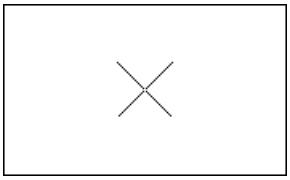
1 Launch the ZOOM command.

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.

Uwaga:

- 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

Uwaga: 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.

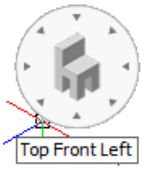
7.9.4 Using the LookFrom control in Isometric mode

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.

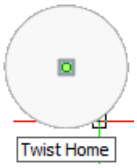
2 Move the cursor to the edge of the **LookFrom** control, on the left half for counterclockwise, on the right half for clockwise rotation. 15°, 30°,45°, 60°, 90°, 120°, 135° and 180° angles are available.



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.

Uwaga: 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.

Uwaga:

- 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.

Uwaga: 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.

Uwaga: 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.

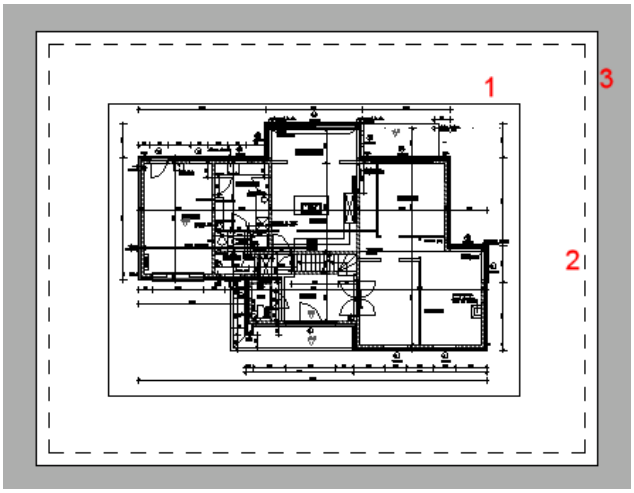
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space in layout, click any layout viewport to make it the current viewport, 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.

Uwaga:

- 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.

Uwaga: 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.

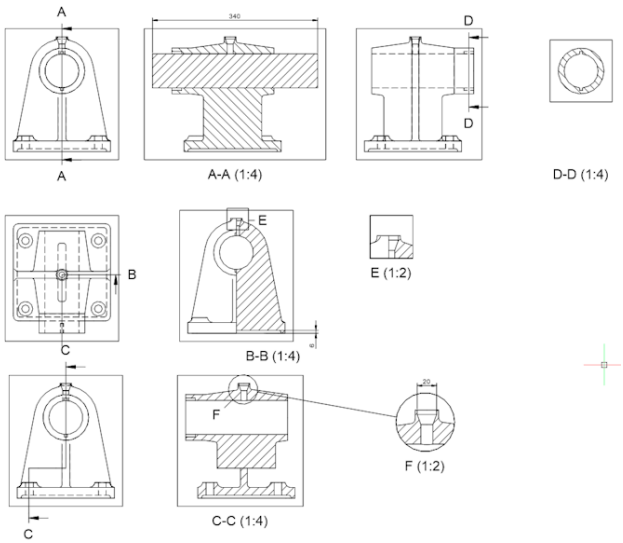
2 In the **Layout Manager** dialog box you can:

- 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.

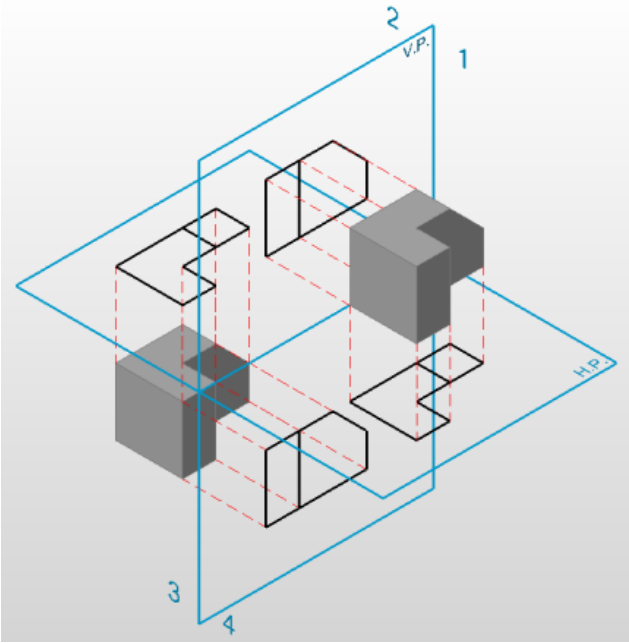
To generate the standard views of your 3D solid model, run the ViewBase command from Model Space.

You can choose between first angle (European) projection or third angle (American) projection.

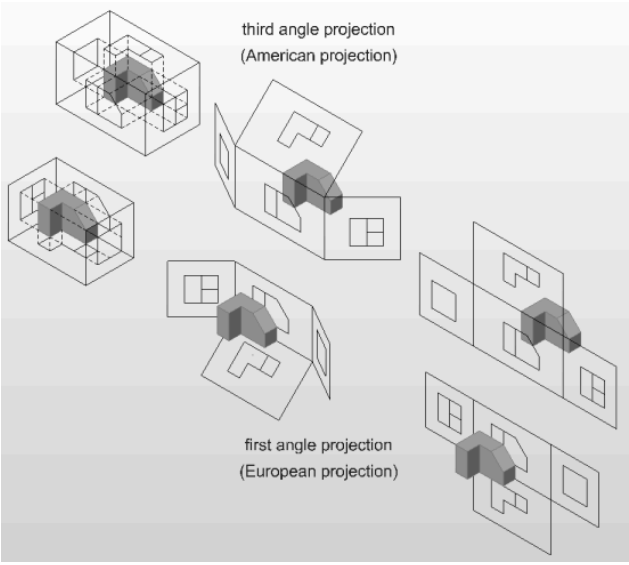
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the 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.

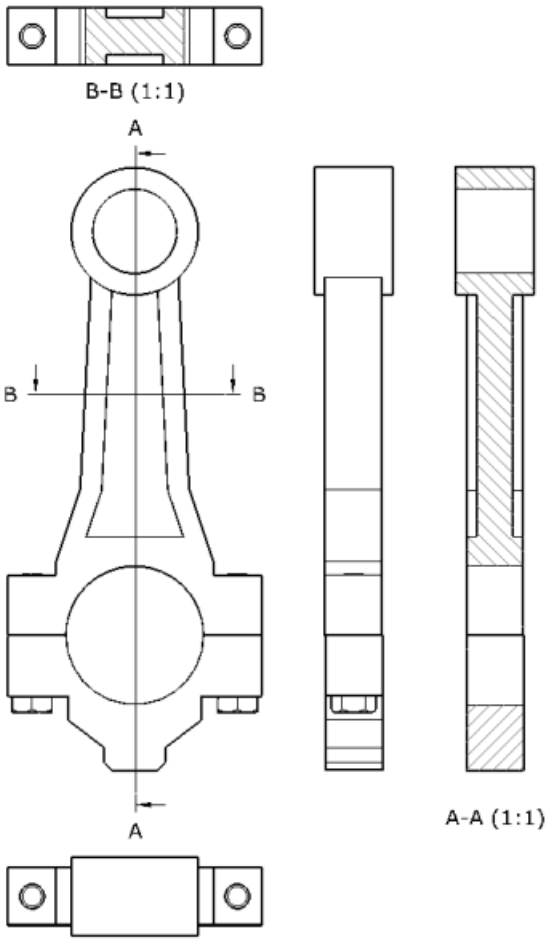
The display of the BM_Isometric_Hidden layer is turned off by default.

Additional projected views can be added using the VIEWPROJ command. A projected view inherits the scale, display settings and alignment from the parent view.

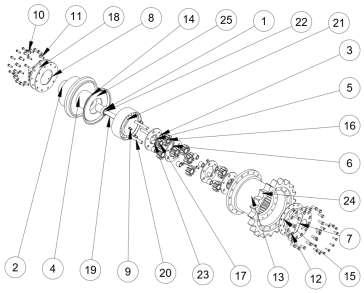
7.19.2 Generating section views

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.

Uwaga:

- 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. Rysowanie 2D

8.1 Tworzenie podmiotów

Na rysunku można tworzyć różne typy elementów. Elementy rysunkowe mogą być bardzo proste, takie jak linie, okręgi, łuki, punkty i promienie, lub złożone, takie jak polilinie, splajny i płaszczyzny. Oprócz elementów 2D można również tworzyć elementy powierzchni 3D lub bryły 3D w **BricsCAD®** .

Aby jeszcze bardziej uzupełnić rysunek, BricsCAD zapewnia szereg narzędzi do dodawania tekstu, kreskowania i wymiarów.

8.1.1 Ogólna procedura tworzenia nowych podmiotów

Nowe elementy są tworzone przy użyciu bieżących ustawień tworzenia elementów na bieżącej warstwie w płaszczyźnie XY bieżącego układu współrzędnych, domyślnie jest to Światowy Układ Współrzędnych (GUW).

1 (opcjonalnie) Ustaw bieżącą warstwę.

2 (opcjonalnie) Dostosuj właściwości tworzenia encji w **Ogólne** sekcja na **Właściwości** płyta.

3 Uruchom polecenie.

4 Wykonaj polecenie zgodnie z monitami poleceń.

8.2 Ustawienia tworzenia jednostek

Domyślne ustawienia i preferencje tworzenia elementów rysunku są określone przez zmienne systemowe.

Możesz przeglądać i modyfikować te zmienne:

- W oknie dialogowym Ustawienia. Rozwiń **Rysowanie** > **Tworzenie Obiektów**. Na górze listy wyświetlane są ogólne właściwości, takie jak kolor, rodzaj linii i warstwa. Poniżej wyświetlane są ustawienia właściwości specyficznych dla jednostki.
- w **Ogólne** w panelu Właściwości, gdy nic nie jest zaznaczone.

ten **Ogólne** ustawienia to:

- **Kolor**: Ustawia kolor jednostki. Domyślnie = JakWarstwa.
- **WArstwa**: Ustawia bieżącą warstwę.
- **Rodzaj linii**: Ustawia bieżący rodzaj linii. Domyślnie = JakWarstwa.
- **Skala Rodzaju Linii**: Ustawia bieżącą skalę rodzaju linii. Domyślnie = 1.
- **SzerokośćLinii**:: Ustawia bieżącą szerokość linii. Domyślnie = JakWarstwa.
- **Przezroczystość**: Ustawia bieżącą przezroczystość. Domyślnie = JakWarstwa.
- **Elewacja**: Określa wysokość (wartość Z) nad płaszczyzną XY bieżącego układu współrzędnych (GUW lub LUW). Domyślnie = 0.

Uwaga: Większość ustawień tworzenia elementów jest zapisywana w rysunku, więc mają zastosowanie tylko do bieżącego rysunku. Jeśli chcesz dostosować te ustawienia dla wszystkich przyszłych rysunków, edytuj domyślne rysunki szablonu i dostosuj tam ustawienia. Rysunki szablonów najlepiej jest zapisać w

folderze zapisanym za pomocą ustawienia ŚCIEŻKA SZABLONU. Zmienna systemowa BASEFILE określa ścieżkę i nazwę domyślnego pliku szablonu.

8.3 Working with layers

You can create and manage layers in the **Layers** section of the Drawing explorer dialog box (EXPLAYERS command) or in the Layers panel (LAYERSPANELOPEN command).

The Drawing Explorerdialog box cannot be opened while you use other commands, while the Layers panel can. You can move and resize it as desired, or even move it to a second monitor.

8.3.1 Current layer

New entities are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

- The current layer is saved through the CLAYER system variable.
- The CENTERLAYER* system variable controls the layer where centerlines and centermarks created by theDIMCENTERcommand are placed on.
- The DIMLAYER* system variable controls the layer for dimension entities created by the dimensioning commands.
- Entities created by theINTERFEREcommand are created on the layer which is specified by the INTERFERELAYER system variable.

* The default value is <Use Current>

To set the current layer, do one of the following:

- With no entity selected, select**Layer**on the Properties panel, then choose the new current layer in the drop down list.
- On the **Entity Properties** toolbar, click the down arrow next to the**Layer**field, then choose a layer in the drop down list.
- On the **Layers** panel on the Ribbon, click the down arrow and choose a layer in the drop down list.
- Right click on the**Layer**field in theStatus Bar, then choose a layer in the context menu.

Uwaga:

- It is possible to turn the display of the current layer off ( ). As a result, everything you draw will be invisible until the display of the current layer is turned on ( ) again. A warning displays if you try to turn off the current layer.
- It is not possible the freeze ( ) the current layer.

8.3.2 Layer states

LAYERSTATE command.

Layer states are named content stored in a drawing.

Layer states allow you to save and restore configurations of layer properties and states. You might want a layer to display in blue sometimes and green at other times, or you need some layers to be off or frozen

or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings, you can use layer states to save your settings. You can save multiple layer states in each drawing. You can copy layer states between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

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):

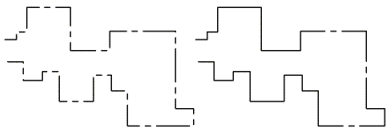
Drawing Unit	LTSCALE
Imperial (MEASUREMENT = OFF)	
Inch	1
Foot	1/12

Drawing Unit	LTSCALE
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.

Uwaga: The Entity Linetype Scale applies to new entities only.

8.5 Entity color

The color determines how an entity displays on screen and how it prints. Entities are created in the current color, as defined by the CECOLOR system variable.

You can choose between 255**index colors**or define a**true color**.

8.5.1 Index colors

Index color is the specification of the color of a pixel on a display screen using a an 8-bit color value, allowing up to 256 possible colors.

Each of the Index Colors has a unique number from 1 to 255. Seven of the index colors can also be referred to by name: red (1), yellow (2), green (3), cyan (4), blue (5), magenta (6) and white/black (7). Index color 7 displays white on a black screen background and black on a white screen background. Index color 7 always prints in black.

The two additional color properties are ByLayer and ByBlock. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. ByLayer is color number 256, and ByBlock is color number 0. In all commands where you would use a color, you can indicate ByLayer and ByBlock as well as by numbers 256 and 0, respectively.

The options to set the Color are:

- ByLayer: Entities adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

- ByBlock: Entities are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block when you insert the block into a drawing.
- Index Color
- True Color

Uwaga: Whether entities using an index color will print in this color, depends on the**Color Table**(CTB) or**Style Table**(STB) that is used for printing. Only if the**Color**setting in the CTB or STB definition file is set to**Use Object Color**, the printed color matches the entity color.

8.5.2 True colors

True color is the specification of the color of a pixel on a display screen using a 24-bit value, which allows up to 16 777 216 possible colors.

The number of bits used to define a pixel's color shade is itsbit-depth. True color is also known as24-bit color. Some new color display systems offer a 32-bit color mode. The extra byte, called thealpha channel, is used for control and special effects information.

True colors use a**RGB**(**R**ed,**G**reen,**Bl**ue) color definition. Each of these parameters has a range from 0 to 255. The RGB definition for black is (0,0,0), the RGB definition for white is (255,255,255).

8.5.3 Setting the current entity color

1 Do one of the following:

- Launch the COLOR command.
- Click the**Select Color**tool button () on the Settings toolbar.
- In theProperties panel, select**Color**, then choose**Select Color**in the drop down list.
- Double click the**Color Field**in theStatus bar.

The Select color dialog box displays.

2 (option) Click the**Index Color** tab, then do one of the following:

- Click one of the colored tiles.
- Type the color number in the**Color Number**field.

3 (option) Click the**True Color**tab, then do one of the following:

- Click the in**Color Selection**pane.
- Set the color parameters in the**Color Parameter**fields.

4 Click the**OK**button to confirm.

8.6 Lineweight

Lineweights determine how thick or thin entities appear on the screen and / or when printed.

The lineweight can be set ByLayer, ByBlock, Default or to a specific value.

Lineweights are defined in millimeters or inches, depending on the LWUNITS system variable.

You cannot assign lineweights to planes, points, TrueType fonts, and raster images.

The options to set the Transparency are:

- ByLayer: The lineweight is determined by the Lineweight property of the current layer. Changing the Lineweight property of a layer will affect all entities that have their lineweight set ByLayer on this layer.
- ByBlock: Entities created using the lineweight ByBlock, are get the Default lineweight until they are included in a block definition. The entities then inherit the lineweight of the block when you insert the block into a drawing.
- Default: The default lineweight is saved through the LWDEFAULT system variable. Changing the Default lineweight affects all the entities in the entire drawing that have the lineweight set to Default.

8.6.1 System variables

The CELWEIGHT system variable defines the lineweight for new entities.

The LWDEFAULT system variable sets the value of the Default lineweight.

The LWDISPLAY system variable controls the display of lineweights. Click the LWT field in the Status bar to toggle the lineweight display On/Off.

The LWDISPLAYSCALE system variable controls the display scale of lineweights in the Model tab. Values between 0 and 1 are accepted.

8.6.2 Printing lineweights

Entities using anindex color will use lineweights when printed only if the**Color Table**(CTB) or**Plot Style**(STB) is defined as **Use object lineweight**. Otherwise the lineweight is defined by the entity color or plot style.

Entities using atrue coloralways use the object line weight when printed.

8.7 Tryb wypełnienia

Polecenie WYPEŁNIJ przełącza wyświetlanie obszarów wypełnionych i zakreskowanych oraz określa, czy szerokie polilinie, ślady, bryły, kreskowania (w tym pełne wypełnienie) i szerokie polilinie są wypełniane. Przełącza wartość zmiennej systemowej FILLMODE.

Gdy TRYB WYPEŁNIENIA jest wyłączony, wszystkie wypełnione elementy są wyświetlane i drukowane jako kontury, co skraca czas wyświetlania lub drukowania rysunku.

	Polilinia	Pierścień	Znak	Płaszczyzna (Bryła)
Wypełnij włączone				

	Polilinia	Pierścień	Znak	Płaszczyzna (Bryła)
Wypełnij				

8.7.1 Ustawianie trybu wypełniania

Wykonaj jedną z następujących czynności:

- Uruchom polecenie WYPEŁNIJ.
- Ustaw wartość zmiennej systemowej FILLMODE.

Uruchom polecenie REGEN lub REGENALL, aby ponownie wygenerować rzutnie.

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 Rysowanie jednostek liniowych

Polecenia rysowania elementów liniowych to:

	Rysuje polilinie o współrzędnych x,y,z.
	Rysuje łuki.
	Tworzy polilinię, która tworzy granicę wokół zamkniętych obszarów.
	Rysuje koła.
	Rysuje szerokie okręgi z polilinii.
	Rysuje elipsy i łuki eliptyczne.
	Rysuje spirale 2D lub helisy 3D.
	Rysuje segmenty linii.
	Rysuje multilinie.
	Rysuje połączone linie i łuki.
	Rysuje regularne wielokąty z polilinii.
	Rysuje półnieskończone linie.

	Rysuje prostokąty i kwadraty.
	Rysuje chmurki wersji.
	Rysuje naszkicowane (odręczne) linie.
	Rysuje otwarte i zamknięte splajny w 2D i 3D.
	Rysuje szerokie segmenty linii.
	Rysuje nieskończone linie.

8.11 Drawing lines

A line consists of two points: a start point and an endpoint. Using theLINEcommand you can draw single lines or a series of connected lines, but each line is considered a separate Line entity.



The Line tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

8.11.1 General procedure

1 Launch the LINE command.

Prompts you: Start of line or [Follow] <Last point>:

2 Specify the start point.

The line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Undo]:

3 Specify the end point.

The line is created. A new line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Undo]:

4 Specify the end point.

The line is created. A new line displays dynamically.

Prompts you: Set end point or [Angle/Length/Follow/Close/Undo]:

5 Do one of the following:

- Keep specifying end points for each new line segment.
- Choose the **Close** option to connect to the start point of the first line.

- Press Enter or right click to stop.

8.11.2 Specifying the angle and the length

The options are:

- Use the Angle and Length command options.
- Use dynamic dimensions (DYNDIVIS = 1 or 2).

  Press the TAB key to switch between the Angle and Length fields.

8.11.3 Specifying the angle to another linear entity

Right click the DUCS field in the Status bar, then choose DUCS on other entities to set the **Enable for 2D Entities** option of the UCSDETECT system variable.

1 Launch the LINE command.

2 Hover over the reference line or linear polyline segment.

  A temporary UCS aligns with line or polyline segment.

  The UCS icon displays at the nearest midpoint or endpoint of the entity under the cursor.

3 Press the Shift-key to lock the temporary UCS.

4 Specify the start point of the line.

  Angles are now relative to the selected line or linear polyline segment.

Uwaga: The WCS is restored when you conclude the LINE command.

8.12 Rysowanie promieni

Promień to linia, która zaczyna się w punkcie i rozciąga się w nieskończoność. Ponieważ promienie rozciągają się do nieskończoności, nie są obliczane jako część zakresu rysunku.

Domyślną metodą rysowania promienia jest wybranie punktu początkowego promienia, a następnie określenie jego kierunku.

Ponieważ są one często używane jako odniesienia do tworzenia innych elementów, promienie i nieskończone linie są czasami nazywane liniami konstrukcyjnymi.



Narzędzie Promień jest dostępne na:

- Menu Rysuj.
- Pasek narzędzi Rysuj.
- Panel wstążki Rysuj.

8.12.1 Generalna procedura

1 Uruchom polecenie PROMIEŃ.

  Monituje: Promień nieskończony: Początek promienia lub [Poziomo/Pionowo/Kąt/Dwusieczna/Równoległy]:

2 Określ punkt początkowy promienia.

Promień wyświetla się dynamicznie.

3 Określ kierunek promienia.

Następny promień jest wyświetlany dynamicznie.

4 (opcjonalnie) Kontynuuj określanie kierunków, aby narysować serię promieni, zaczynając od tego samego punktu.

5 Kliknij prawym przyciskiem myszy, aby zatrzymać rysowanie promieni.

8.13 Rysowanie nieskończonych linii

Linia nieskończona (XLINE) to linia przechodząca przez dany punkt, zorientowana pod określonym kątem i rozciągająca się do nieskończoności w obu kierunkach. Ponieważ nieskończone linie rozciągają się do nieskończoności, nie są obliczane jako część zakresu rysunku.

Domyślną metodą rysowania linii nieskończonej jest wybranie punktu, a następnie określenie jego kierunku.

Ponieważ są one często używane jako odniesienia do tworzenia innych elementów, promienie i nieskończone linie są czasami nazywane liniami konstrukcyjnymi.



Narzędzie Linia nieskończona jest dostępne na:

- Menu Rysuj.
- Pasek narzędzi Rysuj.
- Panel wstążki Rysuj.

8.13.1 Generalna procedura

1 Uruchom polecenie XLINE.

Prosi cię:

Linia nieskończona: Ustaw punkt wzdłuż linii lub [Poziomo/Pionowo/Kąt/Dwusieczna/Równoległy]:

2 Określ punkt początkowy nieskończonej linii.

Nieskończona linia wyświetla się dynamicznie.

3 Określ kierunek nieskończonej linii.

Następna nieskończona linia wyświetla się dynamicznie.

4 (opcjonalnie) Kontynuuj określanie kierunków, aby narysować serię nieskończonych linii, zaczynając od tego samego punktu.

5 Kliknij prawym przyciskiem, aby zatrzymać.

8.14 Drawing polylines

A polyline is an open or closed sequence of connected line and/or arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment.

The PLINE command interactively creates open and closed polylines entering points.

The BOUNDARY and -BOUNDARY commands create closed polylines from an enclosed area, defined by other entities.

The JOIN command creates polylines from a series of connected lines, polylines and arcs.

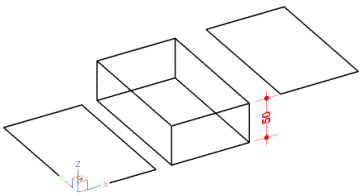
The **Polyline** tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

The**Thickness** property specifies the height of a polyline. IfThickness= 0 (zero) there is no visual difference between polylines and a series of lines and/or arcs.

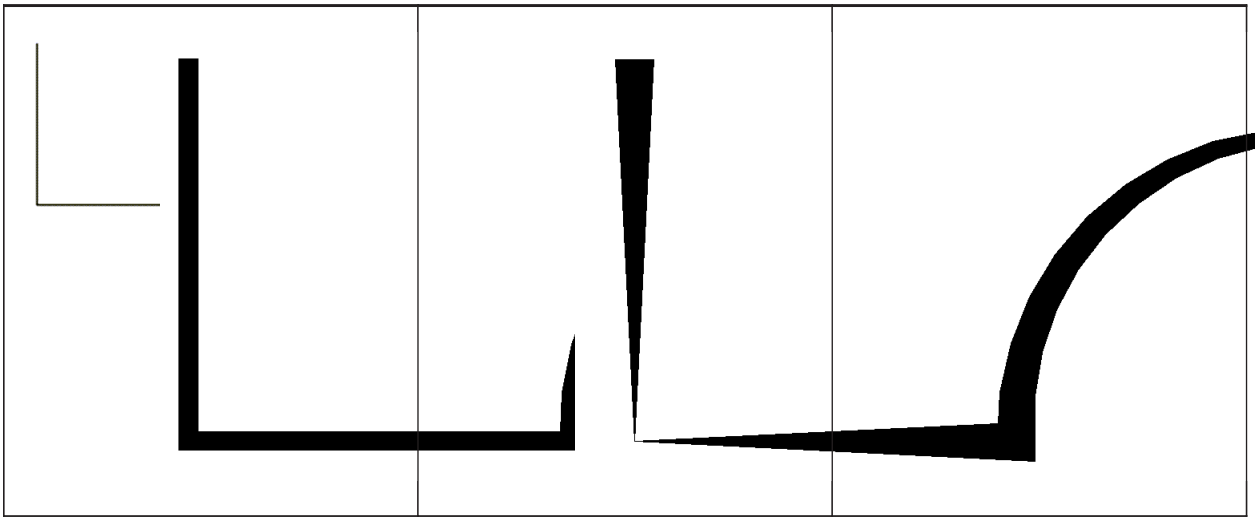
The default thickness is saved through the THICKNESSsystem variable.

The **Elevation** property specifies the distance between the plane of the polyline and the XY-plane of the WCS (World coordinate system).



THICKNESS= 0 (left),**THICKNESS**= 50 (middle),**ELEVATION**= 50 (right)

When a polyline is edited, you can modify the entire polyline or change individual segments and you can add or remove vertices.



Zero width polyline	Constant width polyline	Tapered width polyline

Uwaga:

- WhenFILLMODE = 0, only the outline of all filled entities, such as wide polylines and planes, displays.
- Depending on the value of theGRIPSsystem variable polylines have grips on their endpoints and vertices (GRIPS = 1) or endpoints, vertices and vertex midpoints (GRIPS=2)
- Polylines have a direction, which can be modified by the**Reverse Direction**option of thePEDIT(polyline edit) command.

8.14.1 General procedure

1 Launch the PLINE command.

Prompts you: Select start of polyline or [Follow] <Last point>:

2 Specify a point.

Prompts you: Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]:

3 Specify a point.

Prompts you: Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width/Undo]:

4 Specify a point.

Prompts you:

Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]:

5 Keep specifying points.

6 Do one of the following:

- Choose the **Close** option to connect to the start point and create a closed polyline.
- Choose the **Undo** option to remove the previous vertex.
- Right click or press Enter to stop.

8.14.2 Creating boundary polylines

1 Launch the BOUNDARY command.

The **Boundary** dialog box displays.

2 Click the**Pick points in boundaries**button.

The dialog box closes.

3 Click in the area where you want to create the boundary polyline.

The boundary polyline highlights.

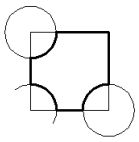
4 (option) Keep picking points to create more boundary polylines.

5 Right click to accept the polyline(s).

The **Boundary** dialog box opens again.

6 Click the**OK**button.

The polyline(s) is (are) created.



8.15 Rysowanie multilinii

Multilinie składają się z dwóch lub więcej równoległych linii, zgodnie ze stylem utworzonym za pomocą polecenia MLSTYL.

Narzędzie Multiline jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** pasek narzędzi
- ten **Rysuj** panel wstążki

8.15.1 Generalna procedura

1 Uruchom polecenie MLINIA.

1 Bieżące ustawienia: Wyrównanie = Góra, Skala = 20,00, Styl = Standardowy

Zostanie wyświetlony monit: Określ punkt początkowy lub [Uzasadnienie / Skala / Styl]:

2 Określ punkt.

3 **Pasek poleceń wyświetli** : Ustaw punkt końcowy lub [Kąt/Długość/Cofnij]:

4 Określ punkt.

5 **Pasek poleceń wyświetli** : Ustaw punkt końcowy lub [Kąt/Długość/Śledź/Cofnij]:

6 Określ punkt.

7 **Pasek poleceń wyświetli** : Ustaw punkt końcowy lub [Kąt/Długość/Śledź/Zamknij/Cofnij]:

8 Wykonaj jedną z następujących czynności:

- Określ więcej punktów.
- Wybierz **Zamknij** możliwość utworzenia zamkniętej pętli.
- Kliknij prawym przyciskiem myszy, aby zakończyć.

Uwagi:

Multilinie są tworzone przy użyciu bieżącego stylu multilinii zdefiniowanego przez zmienną systemową CMLSTYLE.

Zmienna systemowa CMLJUST ustawia wyrównanie multilinii względem określonych punktów. Dostępne opcje to:

- **Góra**: W górnej linii stylu multilinii.
- **Zero**: Przy odsunięciu zerowym definicji stylu multilinii.
- **Dół**: W dolnej linii stylu multilinii.

Zmienna systemowa CMLSCALE ustawia skalę multilinii.

Właściwości multilinii można edytować w **Właściwości** płyta.

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 Drawing freehand sketches

A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline.

A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

The SKPOLY system variable defines whether a chain(s) of connected lines or a polyline(s) is created.

The SKETCHINC system variable sets the length or increment of each line or polyline segment. The smaller the increment, the more accurate your sketch is, but a small value increases the file size.



The Sketch tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.

8.17.1 General procedure

1 Launch the SKETCH command.

Prompts you: Length of segments <current length>:

2 Do one of the following:

- Press Enter to accept the current length.
- Type a new value in the Command line.
- Click two points to define the length graphically.

3 Click to start sketching (pen down).

The crosshairs are replaced by thePenciltool ( )

Prompts you:

Press ENTER or click to toggle [Erase/Connect/save and eXit/Quit no save/Record (save)]:

4 Click to start sketching: as you move the cursor the sketched line displays in light green.

5 (option) Click to stop sketching (pen up), then click again (pen down) to resume.

6 (option) Choose the**Erase**option.

The cursor changes to the**Erase**tool ( ).

Move the cursor to the end of a sketch line to start erasing.

Click to stop erasing and resume sketching.

7 (option) Choose the**Record (save)**option to save the sketching line and keep sketching.

8 Choose the**save and eXit**option to save the sketch lines and exit the sketch command.

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 Rysowanie łuków

Domyślną metodą rysowania łuków kołowych jest określenie trzech punktów: punktu początkowego, drugiego i końcowego.

Inne metody rysowania łuków można znaleźć w podmenu Łuk w menu Rysuj oraz na pasku narzędzi Łuki.

Narzędzie Arc jest dostępne na:

- Menu Rysuj.

- Pasek narzędzi Rysuj.
- Panel wstążki Rysuj.
- Zakładka Rysuj w Kwadracie Bez zaznaczenia.

8.19.1 Generalna procedura

1 Uruchom polecenie ARC.

Monituje: Określ punkt początkowy łuku.

2 Określ punkt.

Monituje: Podaj drugi punkt lub [Kąt/Środek/Kierunek/Koniec/Promień]:

3 Określ punkt.

Prosi Cię: Punkt końcowy:

4 Określ punkt.

8.19.2 Edycja łuku

1 Wybierz łuk.

Właściwości łuku są wyświetlane w panelu Właściwości.

2 Wpisz nową wartość w**Środek**,**Promień**,**Kąt początkowy**lub**Kąt kąńcowy**, a następnie naciśnij klawisz Enter.

3 Naciśnij klawisz Esc, aby zatrzymać edycję łuku.

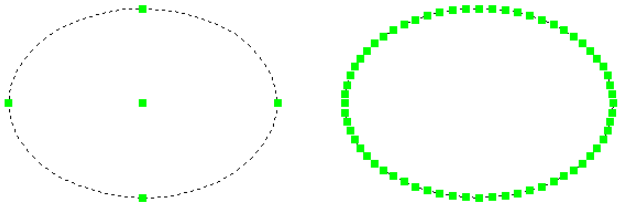
8.20 Drawing ellipses

The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. Half the major axis is called the major radius, half the minor axis is the minor radius. The ratio (minor radius divided by the major radius) defines the eccentricity of the ellipse. A circle is an ellipse with an eccentricity of 1.

Other methods to draw ellipses can be found in theEllipsesubmenu of the Draw menu and in the Ellipses toolbar.

The PELLIPSE system variable controls whether real ellipses or a polyline representation is created.

The difference between a real ellipse and a polyline representation of an ellipse is visible when you select the ellipse. A real ellipse has five handles: center point and the endpoints of the axes. A polyline representation of an ellipse is a closed polyline composed of arc segments.



real ellipse (left) and the polyline representation of an ellipse (right)

The Ellipse tool is available on:

- The Draw menu.
- The Draw toolbar.
- The Draw ribbon panel.
- The Draw tab of the No Selection Quad.

8.20.1 General procedure to create an ellipse

1 Launch the ELLIPSE command.

Prompts you: First end of ellipse axis or [Arc/Center]:

2 Specify a point:

Prompts you: Second end of axis:

3 Specify a point:

Prompts you: Set Other axis or [Rotation]:

4 Specify a point.

8.20.2 General procedure to create an elliptical arc

An elliptical arc is a portion of an ellipse.

1 Launch the ELLIPSE command.

Prompts you: First end of ellipse axis or [Arc/Center]:

2 Select the Arc option.

Prompts you:

3 Specify a point:

Prompts you: Second end of axis:

4 Specify a point:

Prompts you: Set Other axis or [Rotation]:

5 Specify a point.

Prompts you: Set start angle of arc or [Parameter]:

6 Do one of the following:

- Type a value in the dynamic field.

- Click to specify the arc graphically.

Prompts you: [Parameter/Included angle] <End angle>:

7 Click a point.

8.20.3 Editing an ellipse or an elliptical arc

1 Select the ellipse or elliptical arc..

The properties of the ellipse display in theProperties panel.

2 Type a new value in the**Center**,**Major radius**,**Minor radius**,**Radius ratio**,**Start angle**or**End angle**field, then press Enter.

3 Press the Esc key to stop editing.

Uwaga: Editing the Start angleorEnd anglefields of an ellipse turns the ellipse into an elliptical arc. Setting both properties to 0 (zero) turns an elliptical arc into an ellipse.

8.21 Rysowanie prostokątów

Prostokąt jest tworzony jako zamknięta, czteroboczna polilinia.

Prostokąt jest rysowany przez określenie dwóch przeciwległych narożników. Wymiary dynamiczne pozwalają określić szerokość i wysokość prostokąta.

Dopóki **Obrócony** wybrana jest opcja, boki prostokąta są zawsze równoległe do osi x i osi y bieżącego układu współrzędnych (GUW lub LUW).



Narzędzie Prostokąt jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** pasek narzędzi
- ten **Rysuj** panel wstążki
- ten **Rysuj** zakładka **Brak wyboru Quad**

8.21.1 Generalna procedura

1 Uruchom polecenie PROSTOK.

1 **Prosi cię:**

Wybierz pierwszy narożnik prostokąta lub [Fazowanie/Zaokrąglenie/Obrócony/Kwadrat/Rzędna/Grubość/Szerokość linii/Obszar/Wymiary]:

2 Określ punkt.

3 Prostokąt jest wyświetlany dynamicznie.

Monituje : Inny róg prostokąta:

4 Wykonaj jedną z następujących czynności:

- Określ punkt.
- Wpisz wartość w polu wymiaru dynamicznego.
- Naciśnij klawisz TAB, aby przełączyć. Naciśnij Enter, aby potwierdzić.

Uwaga:

Ponieważ prostokąt jest tworzony jako zamknięta polilinia, prostokąty mają kierunek. Kierunek zamkniętej polilinii jest dodatni, jeśli jest rysowana przeciwnie do ruchu wskazówek zegara i ujemny, jeśli jest rysowana zgodnie z ruchem wskazówek zegara. Prostokąty mają kierunek przeciwny do ruchu wskazówek zegara, jeśli współrzędne X i Y drugiego punktu są wyższe lub niższe niż współrzędne pierwszego punktu (= ruch kursora jest dodatni lub ujemny wzdłuż osi X i Y) . Prostokąty mają kierunek zgodny z ruchem wskazówek zegara, jeśli współrzędna X lub Y jest wyższa lub niższa niż współrzędna X lub Y pierwszego punktu (= ruch kursora jest dodatni wzdłuż osi X i ujemny wzdłuż Y- oś lub odwrotnie).

ten Odwróć opcja polecenia EDPLIN (Edytuj polilinię) zmienia kierunek polilinii.

8.22 Rysowanie wielokątów

Wieloboki to zamknięte polilinie składające się z co najmniej trzech, a maksymalnie 1024 boków o równej długości.

Wielokąty można rysować za pomocą następujących metod:

- **Środek - Wierzchołek** : najpierw zdefiniuj punkt środkowy, a następnie wierzchołek (= promień opisanego okręgu).
- **Środek - bok** : najpierw zdefiniuj punkt środkowy, a następnie punkt środkowy boku (= promień metody okręgu wpisanego).
- **Krawędź** : zdefiniuj długość boku wielokąta.



Narzędzie Wielokąt jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** pasek narzędzi
- ten **Rysuj** panel wstążki
- ten **Rysuj** zakładka **Brak wyboru Quad**

8.22.1 Generalna procedura

1 Uruchom polecenie WIELOKĄT.

1 **Monituje** : Wielokąt: Ustaw liczbę boków lub [Szerokość linii/Wiele wielokątów] <4>:

2 Wykonaj jedną z następujących czynności:

- Naciśnij Enter lub kliknij prawym przyciskiem myszy, aby zaakceptować aktualną liczbę boków.
- Wpisz wartość, a następnie naciśnij Enter.

Monituje o : Określ przez: Ustaw środek wieloboku lub [określ według krawędzi]:

3 Określ punkt.

4 **Monituje o** : Określ przez: Wybierz punkt środkowy boku lub [określ przez Wierzchołek]:

5 Określ punkt.

Uwaga:

- Opcja wybrana w kroku 3 (punkt środkowy boku lub określenie przez wierzchołek) zostanie zapamiętana następnym razem.

- Liczba boków jest ustawiana przez zmienną systemową POLYSIDES, ale wartość nie jest zapisywana między sesjami. Przy pierwszym uruchomieniu polecenia WIELOKĄT w sesji BricsCAD POLYSIDES = 4.

8.23 Rysowanie śladów

Ślady są czterostronnie wypełnionymi elementami, utworzonymi z dwóch równoległych boków. Kiedy tworzysz serię śladów, są one połączone ze sobą L.



Narzędzie śledzenia jest dostępne na

- ten **Rysuj** menu.
- ten **Rysuj** panel wstążki

Domyślna szerokość śledzenia jest zapisywana przez zmienną systemową TRACEWID.

8.23.1 Generalna procedura

1 Uruchom polecenie ŚLEDZENIE.

1 **Podpowiada:** Szerokość śladu <bieżąca szerokość>:

2 Wykonaj jedną z następujących czynności:
   - Kliknij prawym przyciskiem, aby zaakceptować aktualną szerokość.
   - Wpisz wartość, a następnie naciśnij Enter.
   - Zdefiniuj graficznie nową bieżącą szerokość, klikając dwa punkty.

   Prosi Cię: Początek śledzenia:

3 Określ punkt początkowy śladu.

4 **Prosi Cię:** Następny punkt:

5 Określ punkt końcowy pierwszego śladu.

6 Linia wskazuje kierunek pierwszego śladu.

   Kierunek drugiego śladu jest wyświetlany dynamicznie.

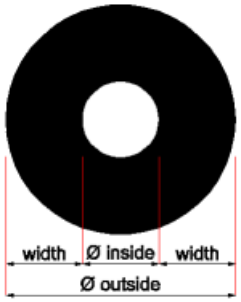
7 Wykonaj jedną z następujących czynności:
   - Kliknij prawym przyciskiem myszy, aby utworzyć ślad.
   - Określ punkt końcowy następnego śledzenia.

8 Wykonaj jedną z następujących czynności:
   - Powtórz krok 5, aby nadal dodawać ślady.
   - Kliknij prawym przyciskiem, aby zatrzymać.

Uwaga: Gdy zmienna systemowa FILLMODE jest wyłączona, wszystkie wypełnione elementy, takie jak bryły, są wyświetlane i drukowane jako kontury.

8.24 Rysowanie pączków

Pierścienie to pełne, wypełnione koła lub pierścienie utworzone jako zamknięte, dwusegmentowe, szerokie polilinie.



Domyślną metodą rysowania pierścieni jest określenie jego średnicy wewnętrznej i zewnętrznej, a następnie określenie jego środka. Jeśli średnica wewnętrzna wynosi zero, tworzony jest wypełniony okrąg.



Narzędzie Donut jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** panel wstążki

8.24.1 Generalna procedura

1 Uruchom polecenie PĄK.

1 **Monituje** : Średnica wewnętrzna pierścienia [2 punkty/3 punkty/styczny promień styczny] < bieżąca średnica wewnętrzna >:

2 Wykonaj jedną z następujących czynności:

- Kliknij prawym przyciskiem myszy, aby zaakceptować aktualną średnicę wewnętrzną.
- Wpisz nową wartość, a następnie naciśnij Enter.
- Zdefiniuj graficznie nową średnicę wewnętrzną, klikając dwa punkty.

Podpowiada : Średnica zewnętrzna pączka <aktualna średnica zewnętrzna>:

3 Wykonaj jedną z następujących czynności:

- Kliknij prawym przyciskiem myszy, aby zaakceptować aktualną średnicę zewnętrzną.
- Wpisz nową wartość, a następnie naciśnij Enter.
- Zdefiniuj graficznie nową średnicę zewnętrzną, klikając dwa punkty.

Podpowiada : Środek pączka:

Określ punkt.

Podpowiada : Środek pączka:

4 Wykonaj jedną z następujących czynności:

- Kontynuuj tworzenie pączków.
- Kliknij prawym przyciskiem, aby zatrzymać.

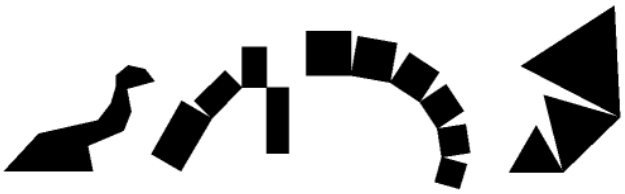
Uwaga:

- Gdy zmienna systemowa FILLMODE jest wyłączona, wszystkie wypełnione elementy, takie jak bryły, są wyświetlane i drukowane jako kontury.

- Bieżąca wartość średnicy wewnętrznej i zewnętrznej jest zapisywana przez zmienne systemowe DONUTID i DONUTOD. Te wartości nie są zapisywane między sesjami. Przy pierwszym uruchomieniu polecenia DONUT w sesji BricsCAD DONUTID=0.5 i DONUTOD=1.

8.25 Rysowanie brył

Bryły 2D to prostokątne, kwadratowe, trójkątne lub czworokątne obszary wypełnione jednolitym kolorem.



Od lewej do prawej: bryły czworokątne, prostokątne, kwadratowe i trójkątne

Narzędzie Bryła jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** pasek narzędzi
- ten **Rysuj** panel wstążki

8.25.1 Generalna procedura

1 Uruchom polecenie SOLID.

1 **Monituje** : Ustaw pierwszy punkt płaszczyzny lub [Prostokąt/Kwadrat/Trójkąt]:

2 Określ punkt.

3 **Powiadamia** : Drugi punkt:

4 Określ punkt.

5 **Podpowiada** : Trzeci punkt płaszczyzny:

6 Określ punkt.

7 **Podpowiada** : Czwarty punkt:

8 Określ punkt.

9 Powstaje bryła.

Podpowiada : Trzeci punkt płaszczyzny:

10 Wykonaj jedną z następujących czynności:

- Kontynuuj określanie punktów, aby utworzyć więcej brył.
- Kliknij prawym przyciskiem, aby zatrzymać.

Uwaga: Gdy zmienna systemowa FILLMODE jest wyłączona, wszystkie wypełnione elementy, takie jak bryły, są wyświetlane i drukowane jako kontury.

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).

Uwaga: 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.

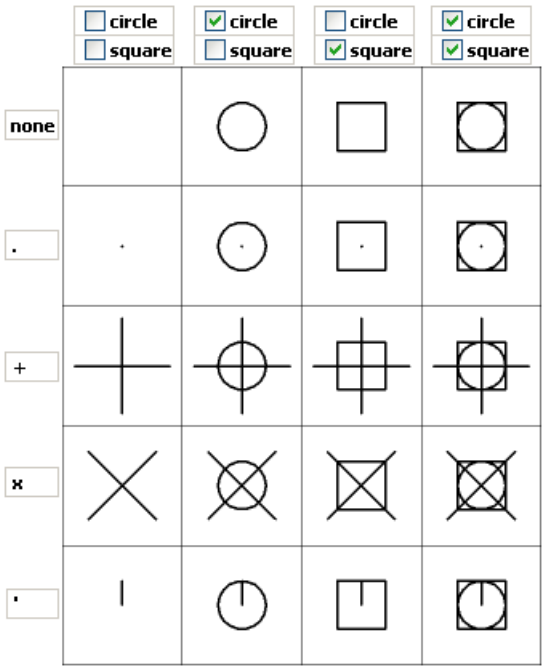
5 BricsCAD reports in the command line: Revision cloud finished.

Uwaga:

- 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 Rysowanie punktów

Ponieważ punkty są bezwymiarowe, pojedynczy piksel powinien być (prawie) poprawną reprezentacją na ekranie. Ale jest to ledwo widoczne, zwłaszcza na złożonym rysunku. W związku z tym można wybierać spośród wielu możliwych stylów wyświetlania, ustawianych przez zmienną systemową PDMODE (Tryb wyświetlania punktu):



Rozmiar stylu reprezentacji punktu jest kontrolowany przez zmienną systemową PDSIZE (Rozmiar wyświetlania punktu). Gdy PDSIZE=0, rozmiar wyświetlania w punktach wynosi 5% rozmiaru ekranu.

Uwaga: Jeśli PDSIZE=0 (zero), użyj polecenia REGEN, aby zmienić rozmiar wyświetlania punktów do 5% rozmiaru ekranu po powiększeniu lub pomniejszeniu.

Narzędzie Punkt jest dostępne na:

- ten **Rysuj** menu
- ten **Rysuj** pasek narzędzi
- ten **Rysuj** panel wstążki

8.27.1 Generalna procedura

1 Uruchom polecenie PUNKT.

1 **Monituje** : Położenie punktu lub [Ustawienia punktu.../Wiele punktów]:

2 Określ punkt.

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.

Uwaga: 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 Drawing helixes

A helix is a curve turning about an axis on the surface of a cylinder (cylindrical helix) or cone (conical helix) while rising at a constant upward angle from a base.

A spiral is a planar curve that winds around a fixed center point at a continuously increasing distance from that point.

The HELIX command can create both 3D helixes and 2D spirals.



The Helix tool is available on:

- the **Draw** menu
- the **Polylines** toolbar
- the **Draw** ribbon panel

8.29.1 General procedure to draw a spiral

1 Launch the HELIX command.

1 Number of turns = 3 Twist = CCW

You are prompted: Specify center point of base:

2 Specify a point.

3 Specify base radius or [Diameter] <1>:

4 Do one of the following:

- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify top radius or [Diameter] <base radius>:

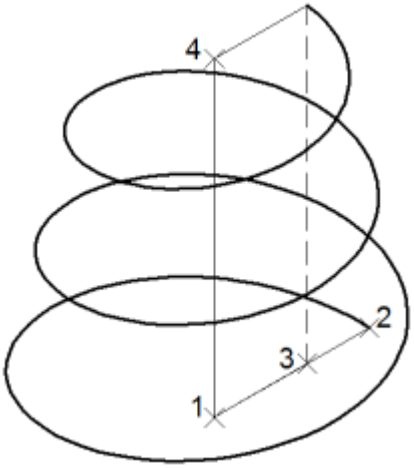
5 Do one of the following:

- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>:

6 Type 0 (zero), then press Enter

8.29.2 General procedure to draw a helix



1 Launch the HELIX command.

1 Number of turns = 3 Twist = CCW

You are prompted: Specify center point of base:

2 Specify a point (1).

3 **You are prompted**: Specify base radius or [Diameter] <1>:

4 Do one of the following (2):

- Specify a point.
- Type a value, then press Enter.

5 Do one of the following (3):

- Press Enter to set the top radius equal to the base radius, thus creating a cylindrical helix.
- Specify a point.
- Type a value, then press Enter.

You are prompted: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1>:

6 Do one of the following (4):

- Specify a point.
- Type a value, then press Enter.

8.29.3 Editing a helix

1 Select the helix.

1 The helix properties display in the **Geometry** category in the **Properties** panel.

2 Edit the properties:

- **Constrain**: constrains either the height, turn height or number of turns. The unconstrained properties are adjusted automatically.
- **Height**: Overall height of the helix. height = turn height * turns.
- **Turns**: Number of turns.

- **Turn height**: Height of one turn.
- **Base radius**: start radius.
- **Top radius**: end radius.
- **Twist**: direction CW (Clockwise) or CCW (Counterclockwise).

Uwaga:

- Set the **Top radius** = **Base radius** for cylindrical helixes.
- Set **Height** or **Turn height** to zero for spirals.
- The EXPLODE command turns a helix into a spline.

8.30 Rysowanie polilinii 3D

Polilinia 3D składa się z segmentów liniowych w przestrzeni 3D.



Narzędzie 3DPOLY jest dostępne na:

- ten **Rysuj** menu
- ten **Polilinie** pasek narzędzi
- ten **Rysuj** panel wstążki

8.30.1 Generalna procedura

1 Uruchom polecenie 3DPOLY.

1 **Zostaniesz poproszony** : Wybierz początek polilinii lub [Podążaj] <Ostatni punkt>:

2 Określ punkt.

3 **Zostaniesz poproszony** : Następny punkt:

4 Określ punkt.

5 **Zostaniesz poproszony** : Następny punkt lub [Zamknij/Cofnij]:

6 Wykonaj jedną z następujących czynności:
   - Wybierz **Zamknij** możliwość połączenia z punktem początkowym i utworzenia płaskiej trójkątnej polilinii 3D.
   - Określ punkt.

   Zostaniesz poproszony : Ustaw następny punkt lub [Zamknij/Cofnij]:

7 Wykonaj jedną z następujących czynności:
   - Wybierz **Zamknij** możliwość połączenia z punktem początkowym.
   - Określ punkt.

   Zostaniesz poproszony : Ustaw następny punkt lub [Zamknij/Cofnij]:

8 Kontynuuj określanie punktów lub naciśnij klawisz Enter, aby zatrzymać.

8.30.2 Edycja polilinii 3D

1 Wybierz polilinię 3D.

1 Właściwości są wyświetlane w panelu Właściwości.

2 PONiżej **Misc**:

- **Dopasuj/Wygładź**: Wybierz **Żaden**, **Sześcienny** lub **Kwadratowy**.

  ten **Sześcienny** oraz **Kwadratowy** opcje tworzą reprezentację splajnu polilinii 3D.

- **Zamknięty**: Po ustawieniu na **Nie**, usuwa segment zamykający.

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.

Uwaga:

- 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.

Uwaga:

- 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. 3D Modeling

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.

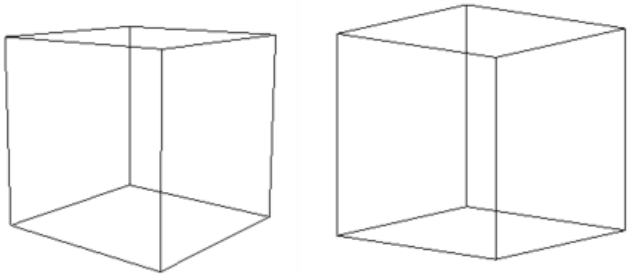
Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

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.

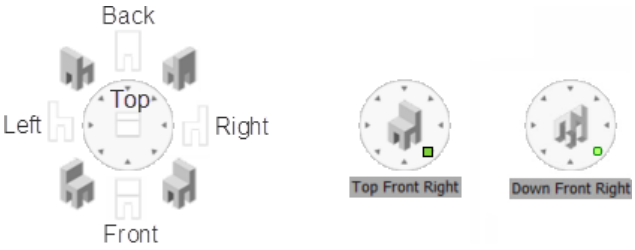
- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode. In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

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. ***Uwaga:*** 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

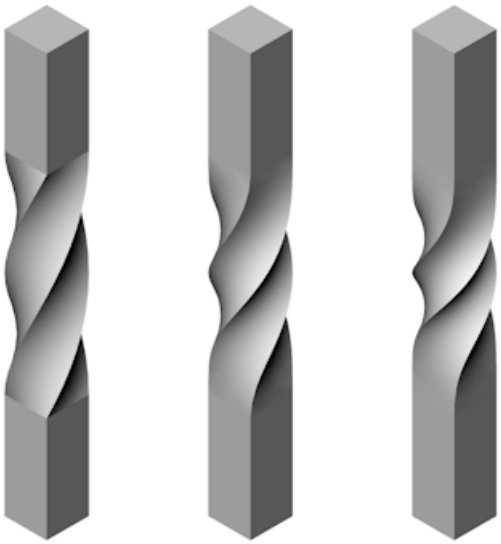
Since entities are created in the XY-plane of the current coordinate system (UCS), it is necessary to align the UCS with the 3D solid face. You have two options here:

- 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.

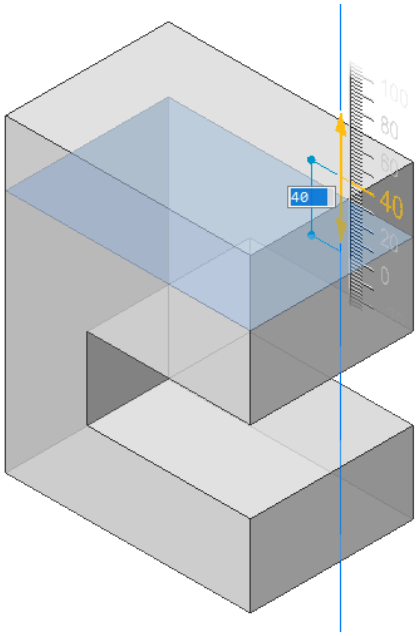
Make sure the **Quad** and **Dynamic Dimensions** are switched ON.

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.

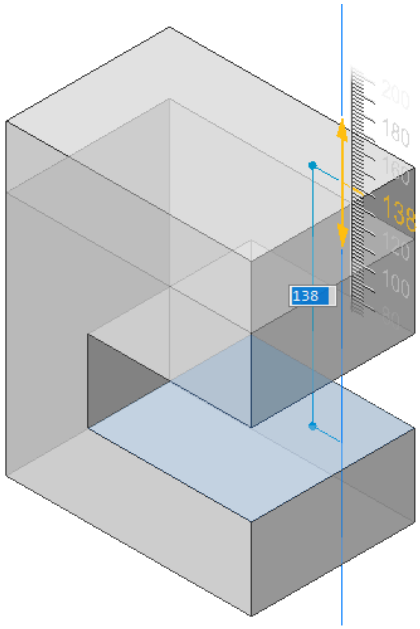
The selected face moves with the cursor and a dynamic dimension field displays the relative distance along with a ruler.



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.

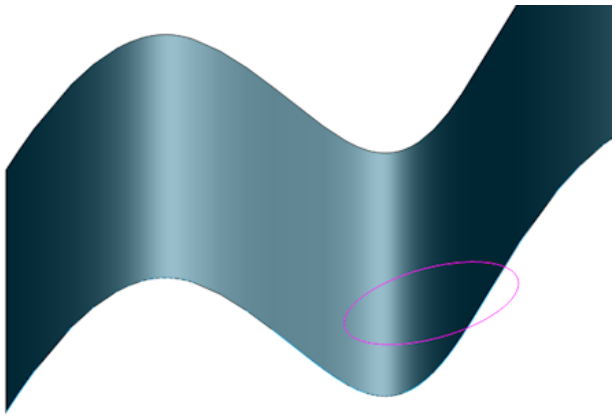
2 Place the cursor over the outline, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

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.

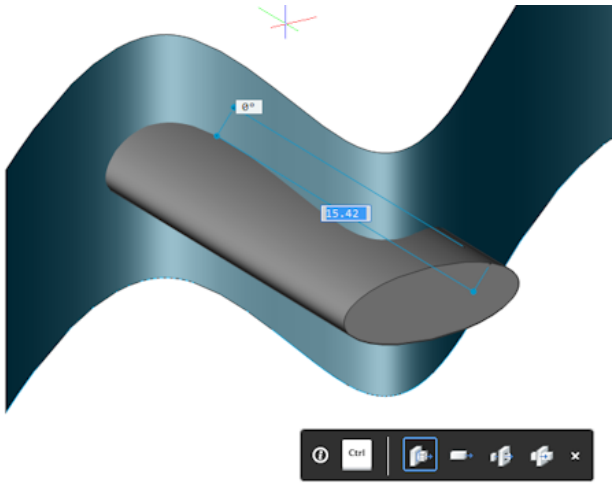


2 Place the cursor over the 2D closed entity, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

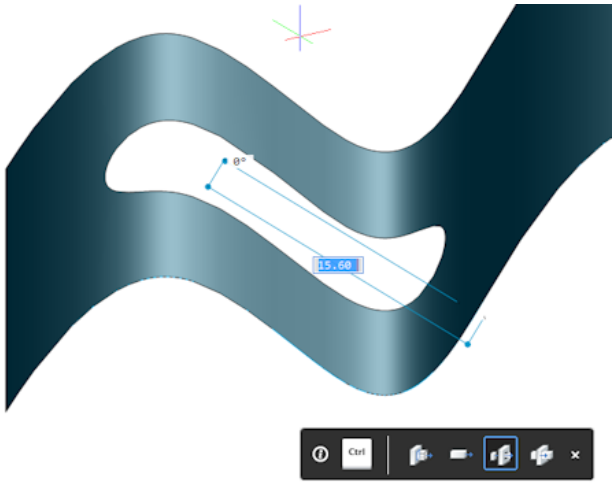
3 Extrude the 2D entity in the direction of the surface.

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.

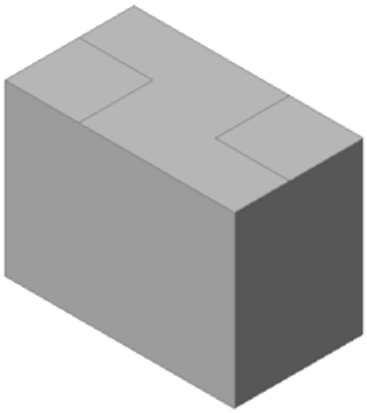
2 Select one or more faces of the feature.

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

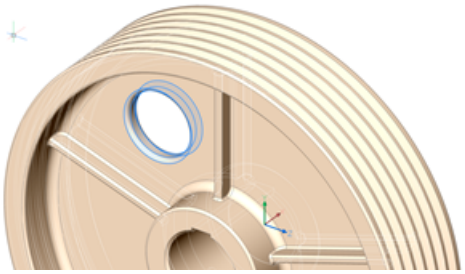
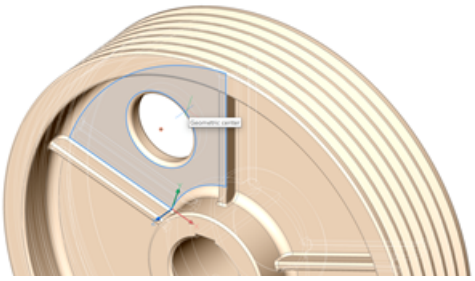
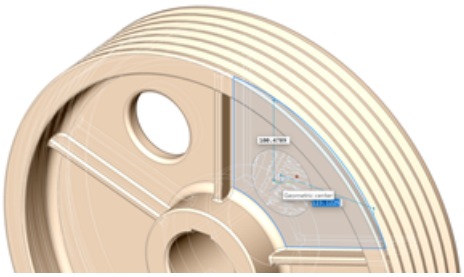
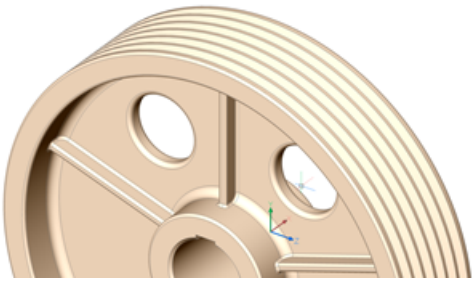
3 Specify the base point.

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.

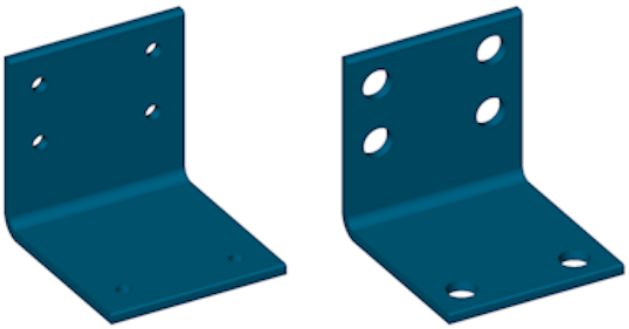
For a twist deformation, it is possible to specify the continuity between the deformed and the fixed portion of the entities.

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.

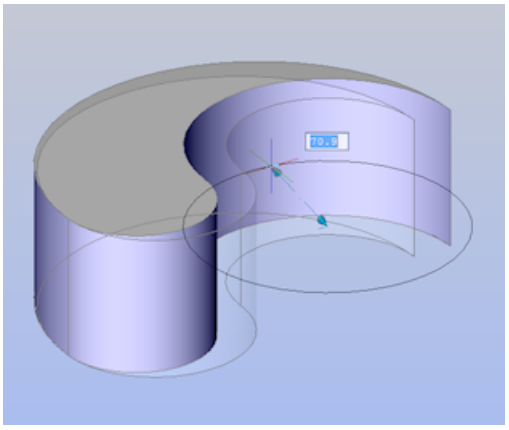
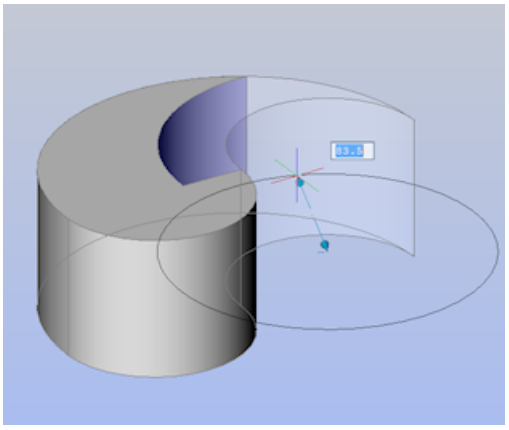
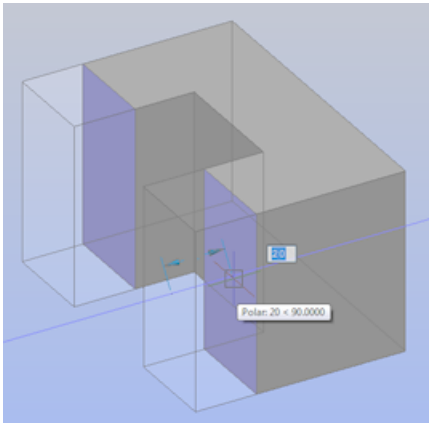
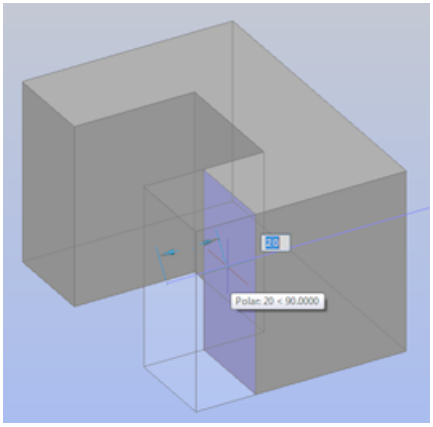
- tangent surfaces (planes, cylinders and cones) ()
- coincident planes ()
- parallel planes ()
- perpendicular planes ()
- cylinders perpendicular to planes ()
- coaxial surfaces (cylinders and cones) ()
- surfaces of equal radius (cylinders and spheres) ()
- vertices between 4 or more faces ()
- edges between coincident faces ()
- switch off/on all ()

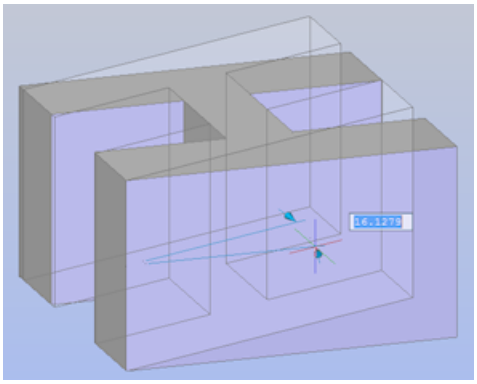
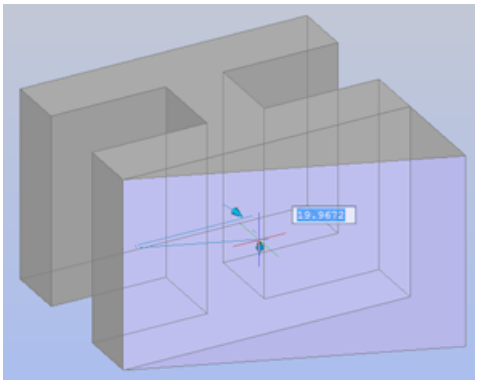
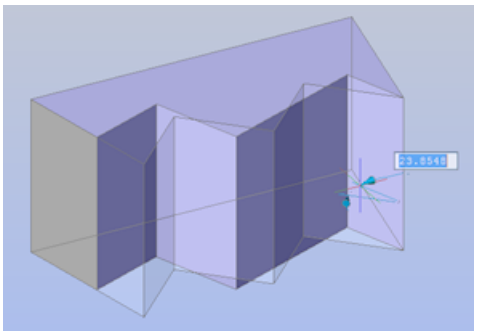
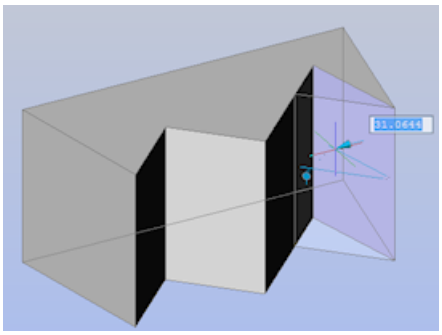
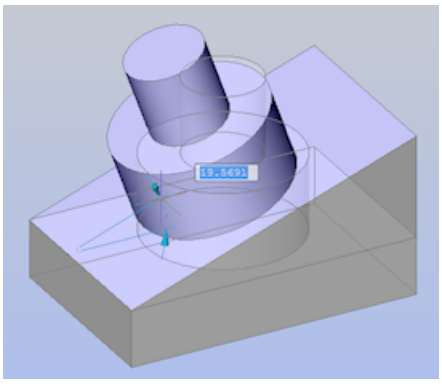
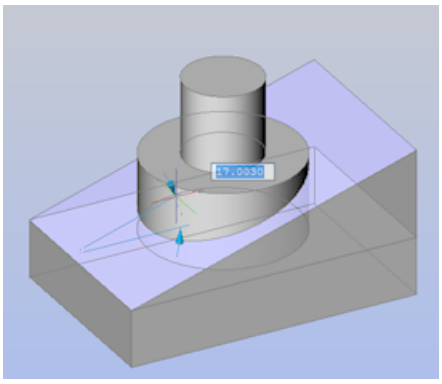
The design intent options can also be set by editing the DMRECOGNIZE system variable in the **Settings** dialog box.

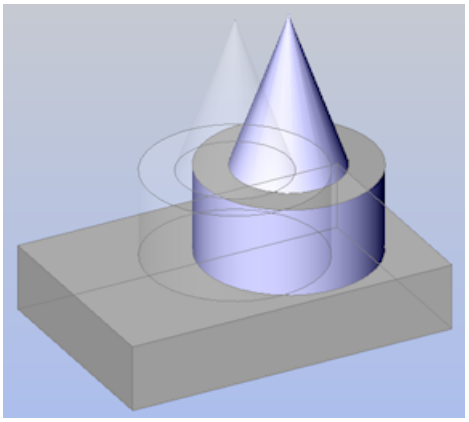
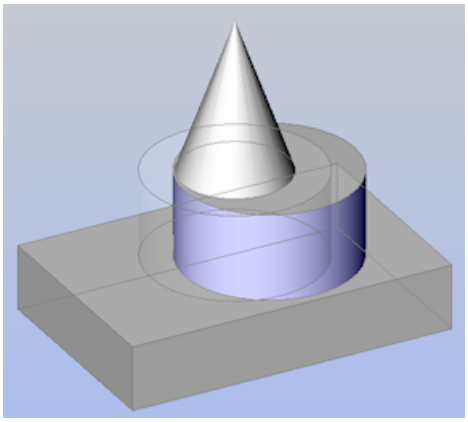
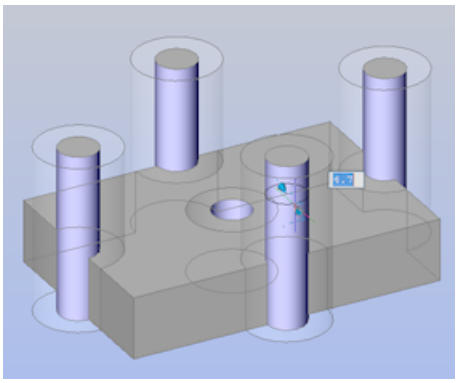
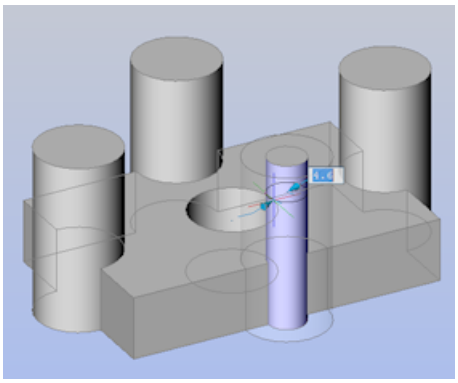
Click the **Options** tool button ( ) on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly.

Uwaga: 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.

Uwaga:

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

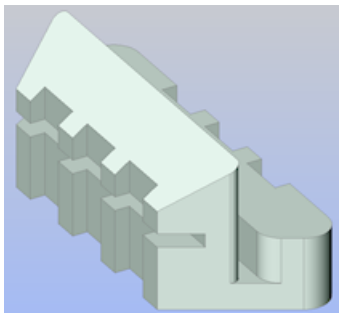
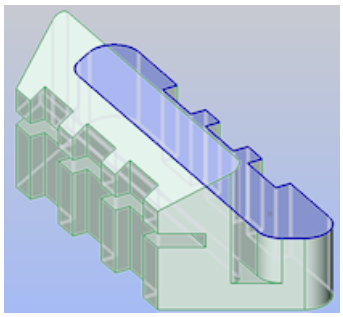
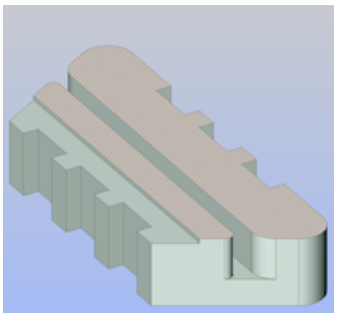
1 Launch SECTIONPLANE command.

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

Uwaga: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

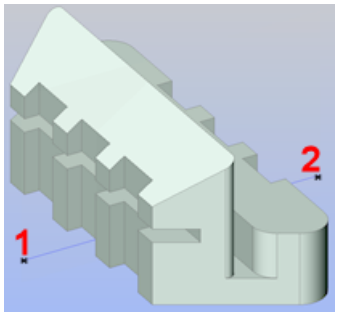
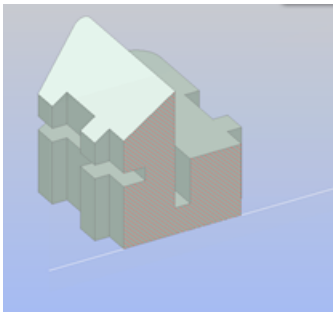
2 Specify two points.

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.

Uwaga: If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

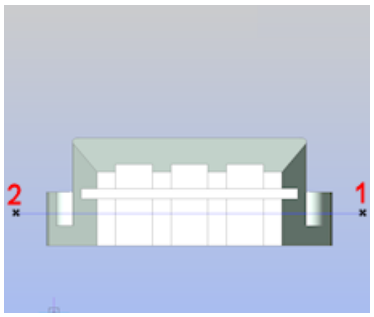
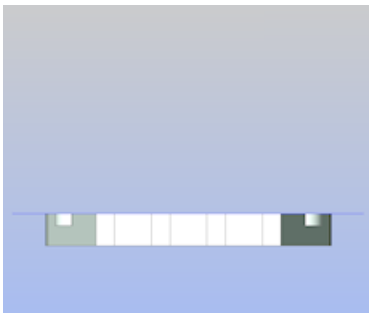
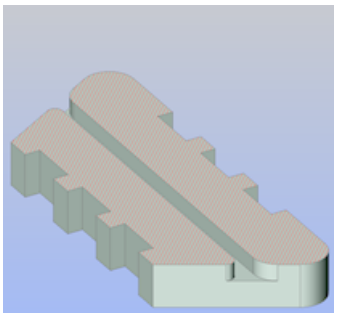
5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

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

Uwaga: 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:

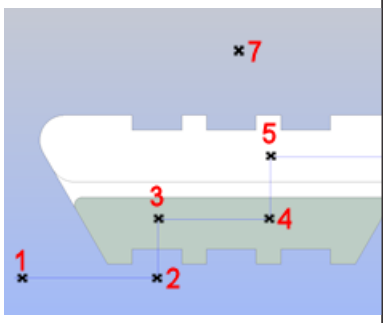
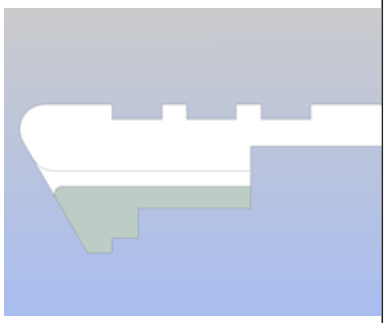
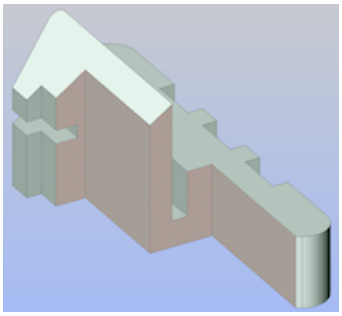
7 Specify the other points (3 - 6), then right click or press Enter to stop.

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.

Uwaga: 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:

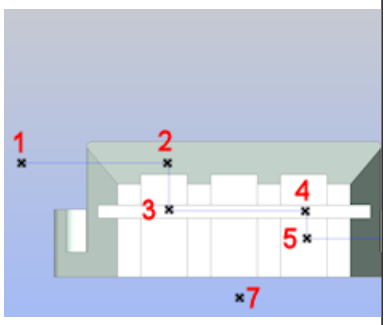
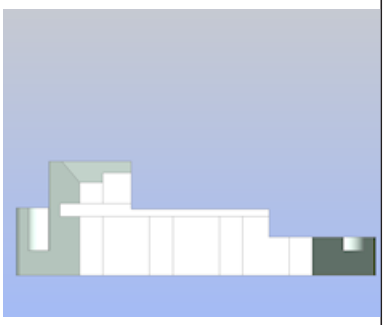
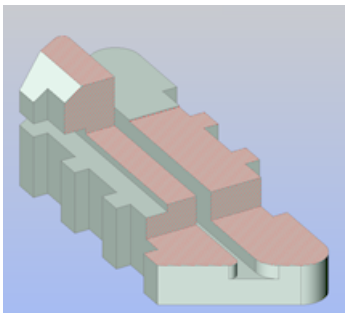
11 Specify the other points (3 - 6) , the right click or press Enter to stop.

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

Uwaga: 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.

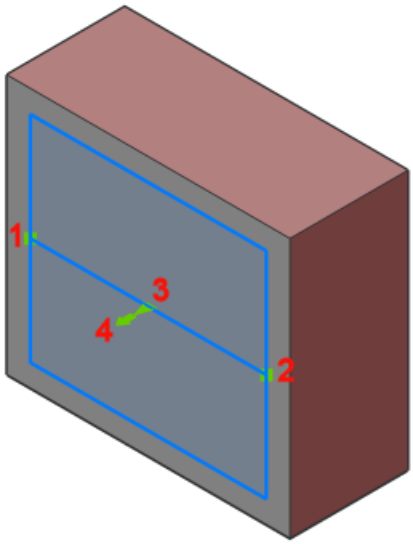
9.6.8 To modify the size and position of a section plane

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.

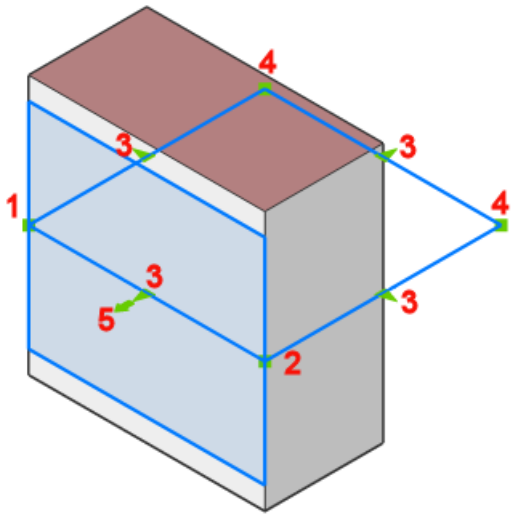
3 (option) Set the **State** property to **Plane**:



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.

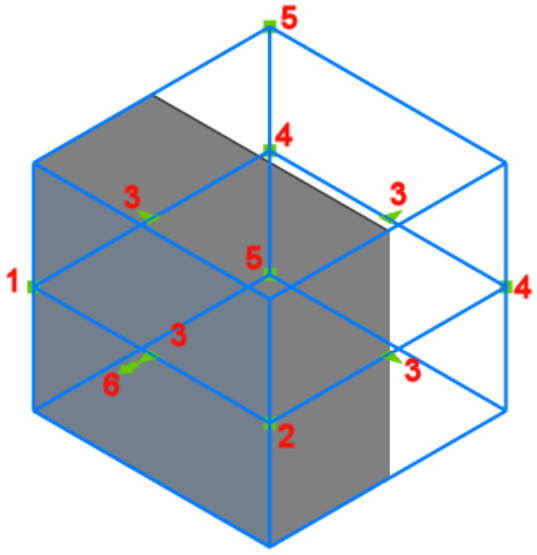
4 (option) Set the **State** property to **Boundary**:



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

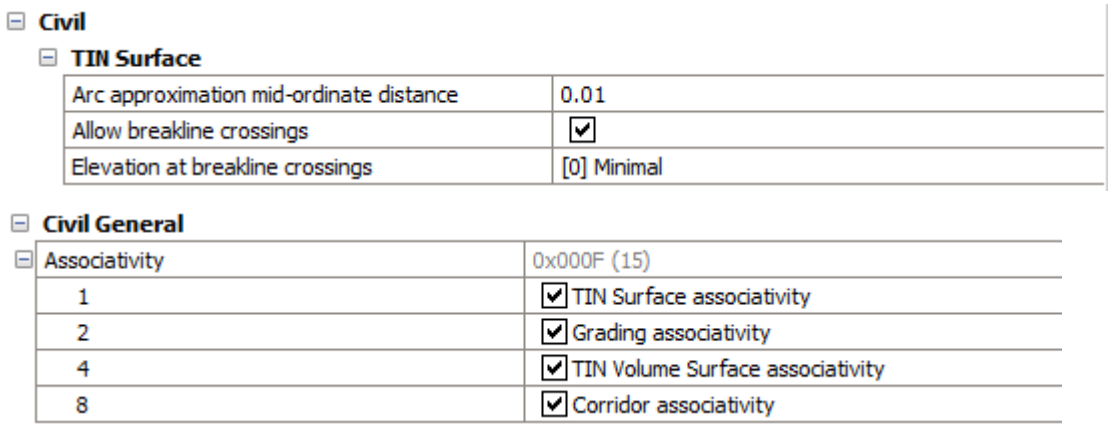
In BricsCAD, topographical surfaces are created using the TIN Surface tool (triangulated irregular network) or the TIN command.

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

Uwaga: 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.

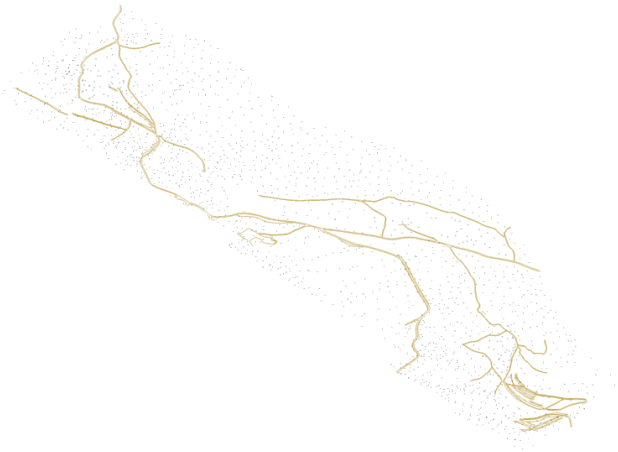
Maximum elevation
Displays the maximum elevation value found in the TIN surface.

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.

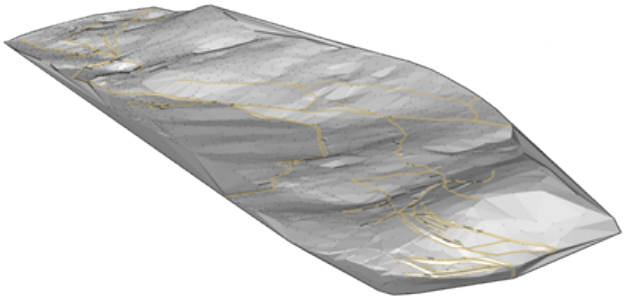


2 Launch the TIN command.

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.

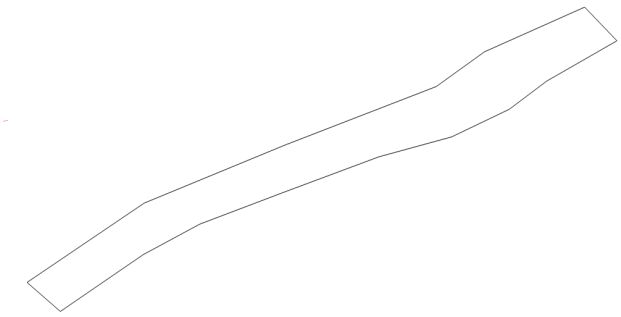
3 Choose the point data file you want to open or choose multiple files to import all at once.

4 Click Open to import the toposurface.

Uwaga: 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.

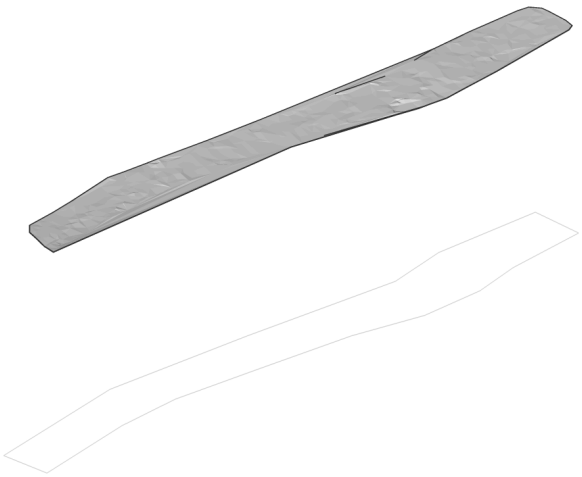
4 Select the closed polyline to use.

5 Specify if the crossed breaklines should be removed.

Uwaga: 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**.

Uwaga: 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.

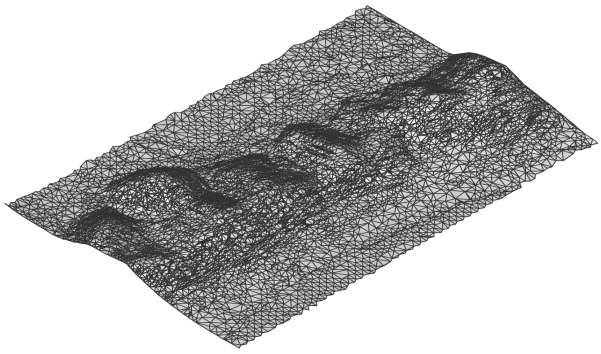
4 Pick a point where you want to start creating a TIN surface.

Specify the elevation value of each of those points, then press ENTER to accept.

Uwaga: 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.

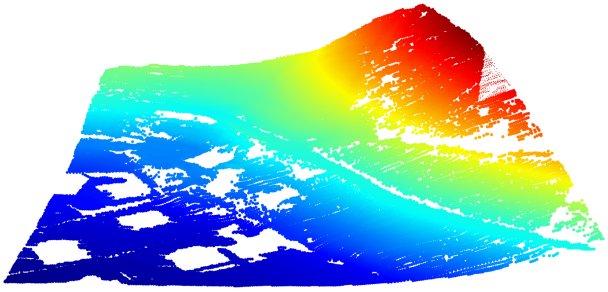
3 Select the **create from Faces** option in the Command line.

4 Select desired 3D faces.

5 Press Enter.

9.7.10 Procedure: Creating a TIN Surface from a Point Cloud

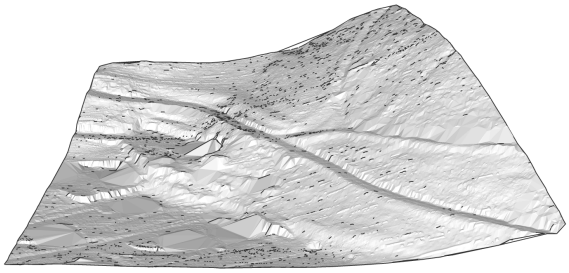
1 Open the drawing file that contains a point cloud.



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**.

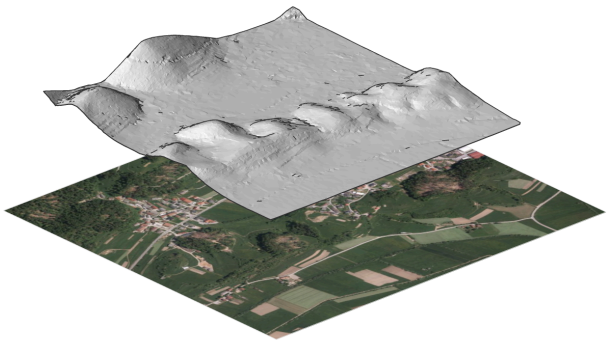
4 Select the Civil 3D surface. A TIN surface is created.

9.7.12 About TIN Assign Image

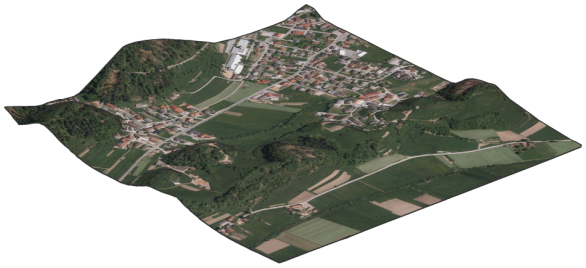
1 Open the drawing file that contains a TIN surface and an image.

2 Launch the TINASSIGNIMAGE command.

3 Select the TIN surface.



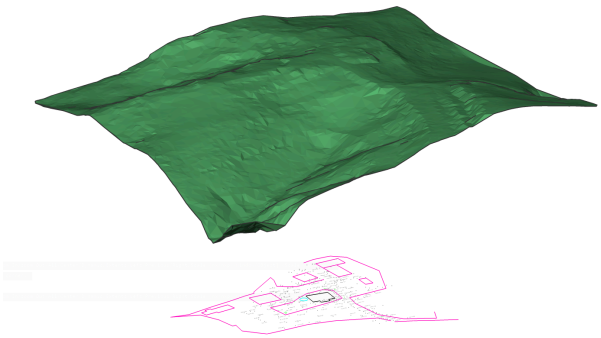
4 Select the raster image.



Uwaga: 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

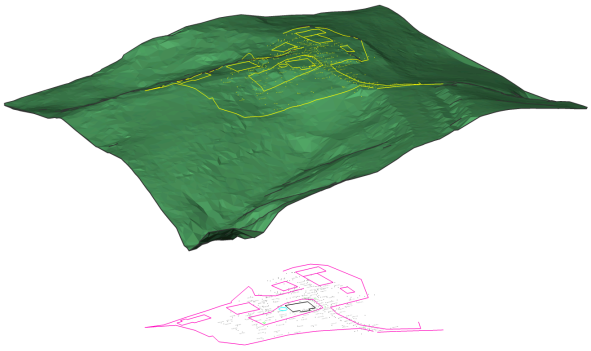
1 Open the drawing file that contains the TIN surface and the entities (points, blocks, text, lines, polylines, circles...).



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

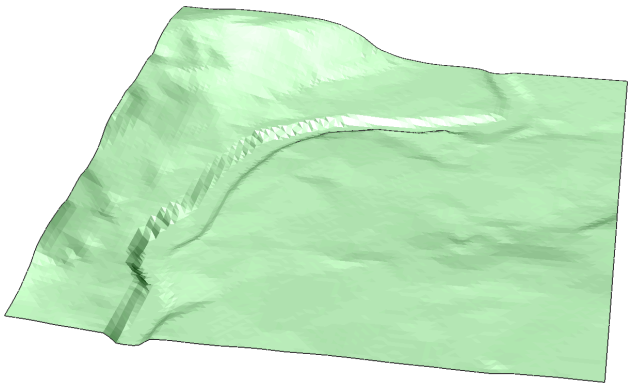
In BricsCAD, topographical surfaces created as a TIN Surface, can be adjusted using different tools.

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.

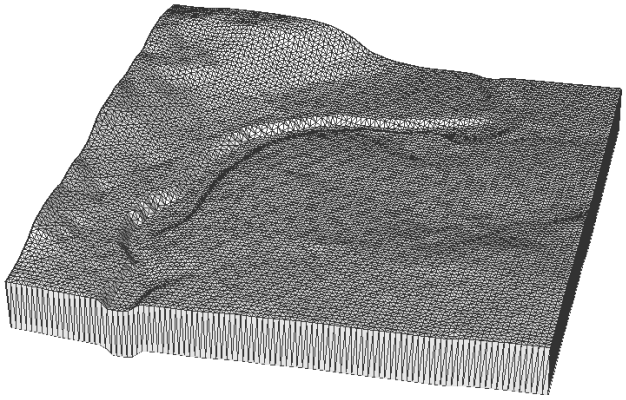
9.9.4 Procedure: Creating a solid from a TIN surface using option Elevation

1 Select a TIN Surface and type TINEXTRACT in the Command line.



2 Select the **Solid** option and press Enter.

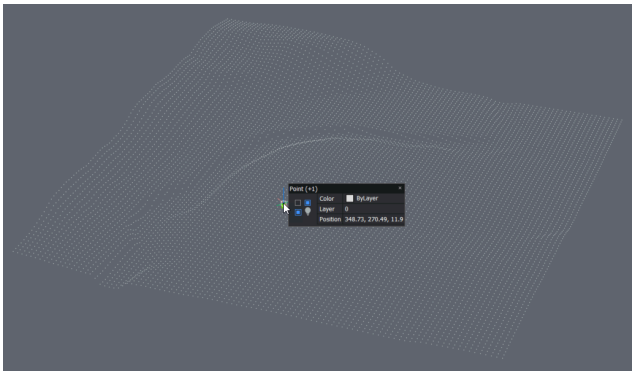
3 Choose the elevation height and press Enter.



9.9.5 Procedure: Creating Points from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

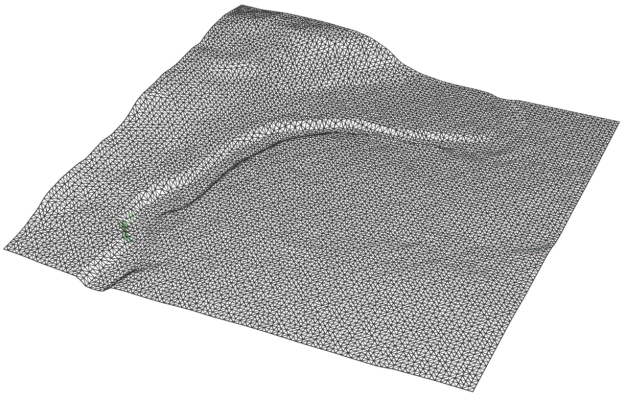
2 Select the **Points** option in the Command line.



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.

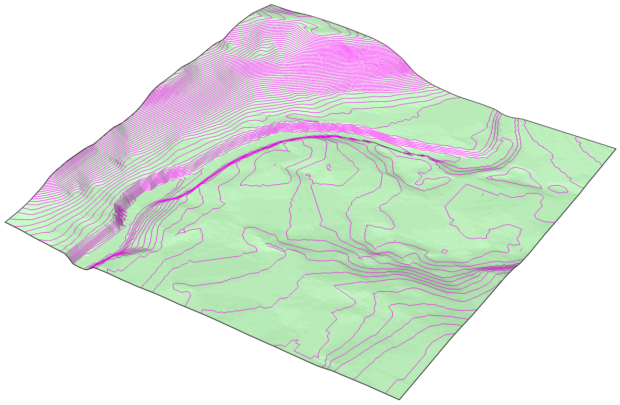


9.9.7 Procedure: Creating a Contours from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Contours** option in the Command line.

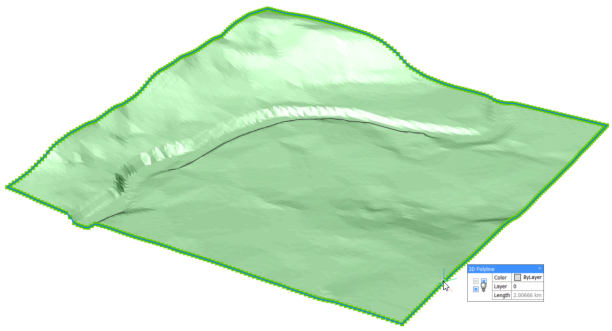
3 Type in **A** for All and press Enter.



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.

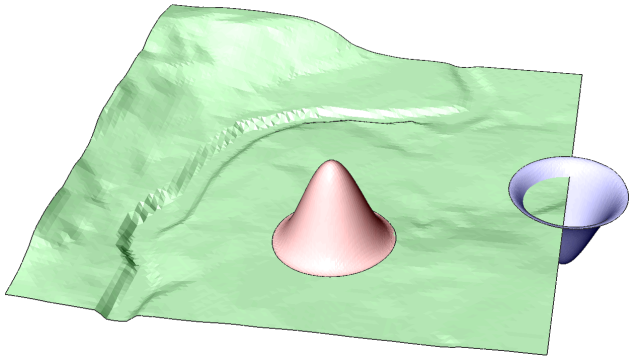


9.9.9 About TIN Merge

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

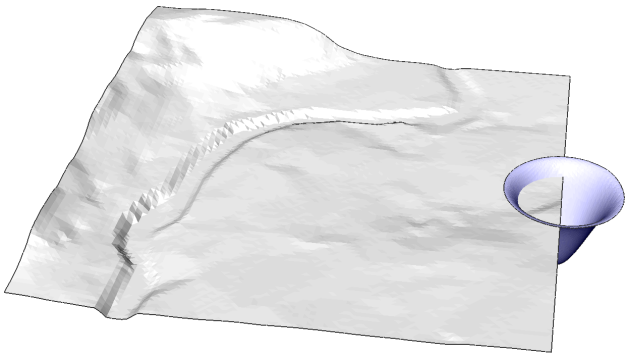
1 Type TINMERGE in the Command line.



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.

Uwaga: 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.

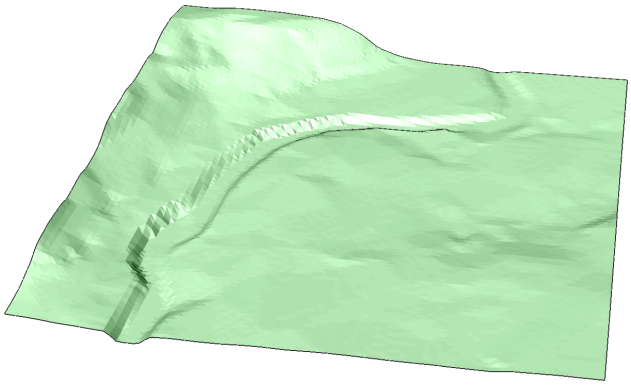
The third surface shows the merge of the red surface as the base surface and the green surface.

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.

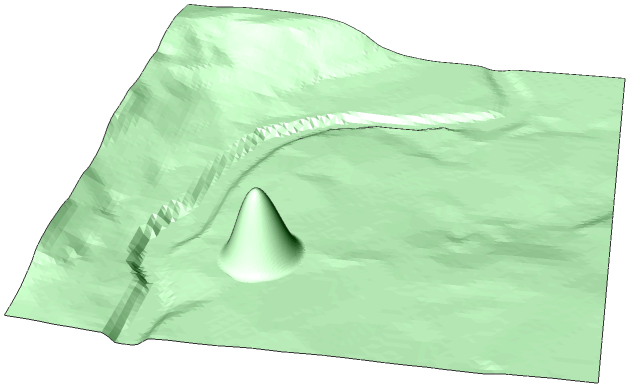
3 Choose a deformation radius.

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.



Uwaga: 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.

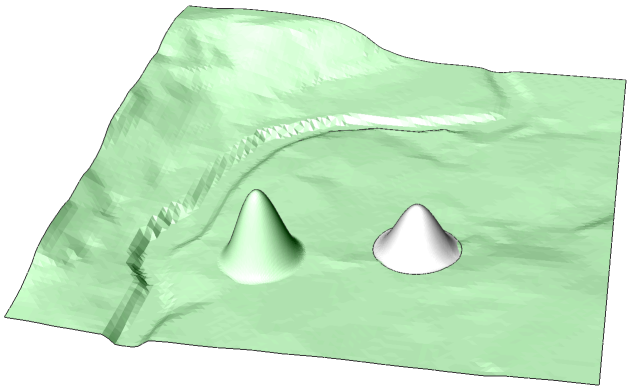
9.9.13 Procedure: Adding deformations on a TIN Surface using specify Contour

1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select the **specify Contour** option.

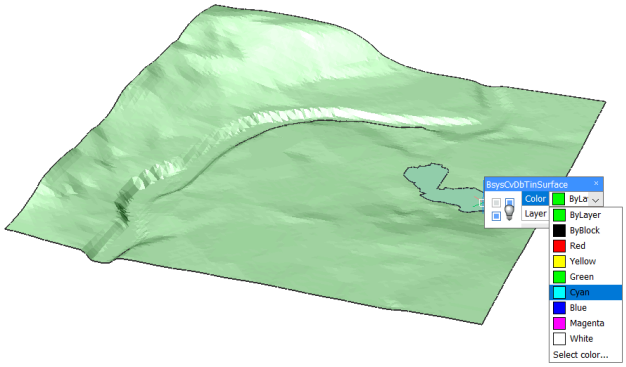
4 Pick a contour on the TIN Surface.



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

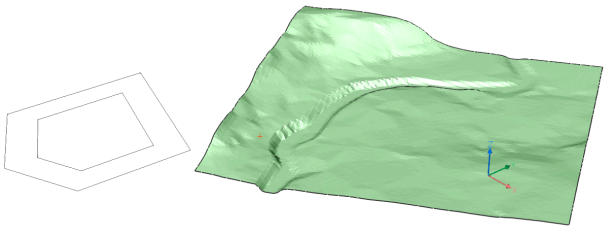
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

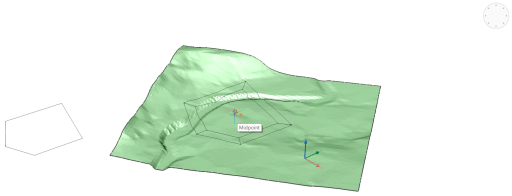
3 Type E and hit ENTER to choose **Select Entity**.

4 Select a closed entity (e.g. Polyline).

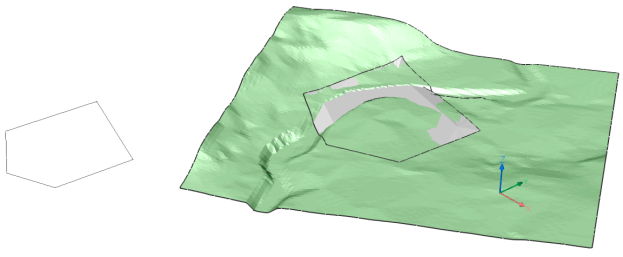
5 Choose an offset distance.



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

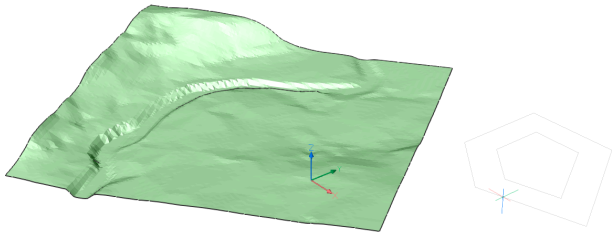
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select **draw Polygon**.

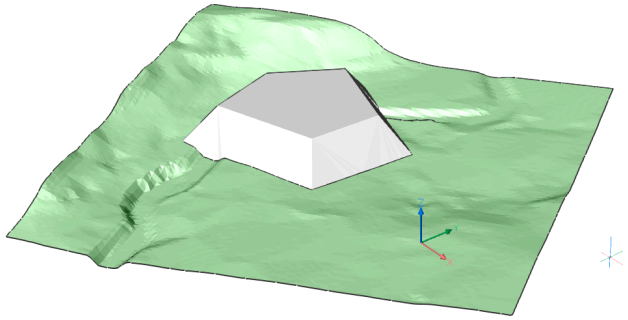
4 Draw a polygon, hit ENTER to close the polygon.

5 Choose an offset distance.



6 Select the area where to apply the deformation.

7 Choose the elevation height.

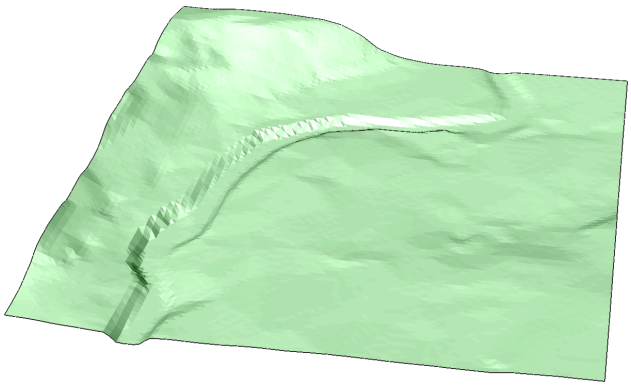


8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for your adjustment.

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.

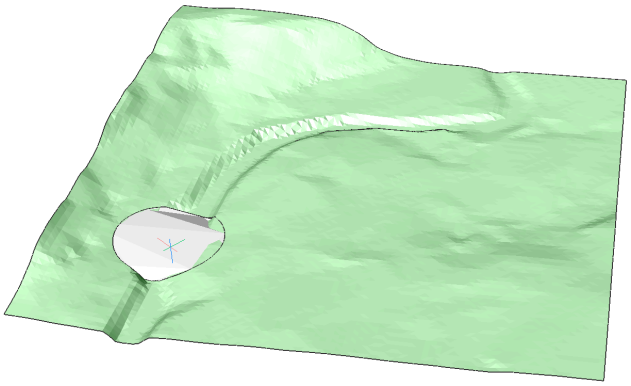


2 Select the **Smoothen** option.

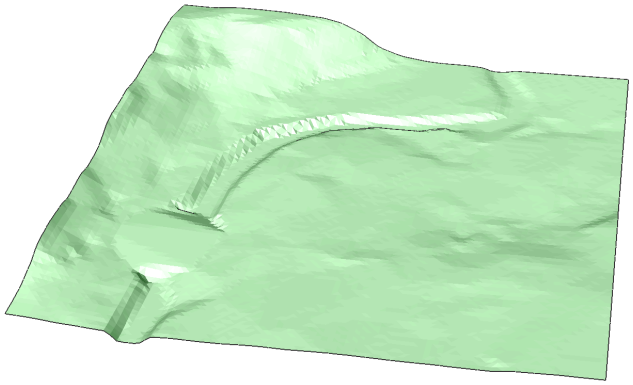
3 Choose a radius.

Uwaga: 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.

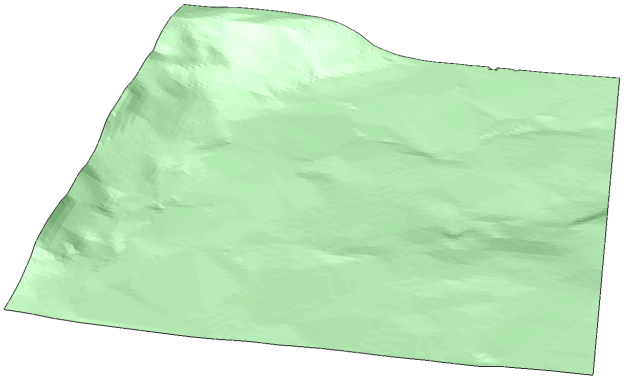
4 A preview of the intervention is shown.



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.

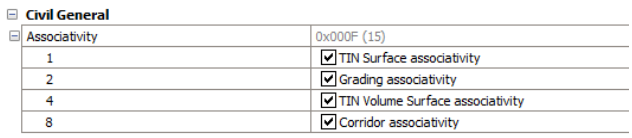
9.10.4 Procedure: Creating a TIN Volume

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**.

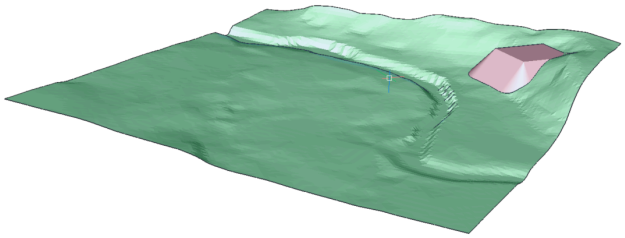
3 Check or uncheck **TIN Volume Surface Associativity**.



9.10.5 Creating a TIN Volume using a comparison TIN Surface

1 Launch the TINVOLUME command.

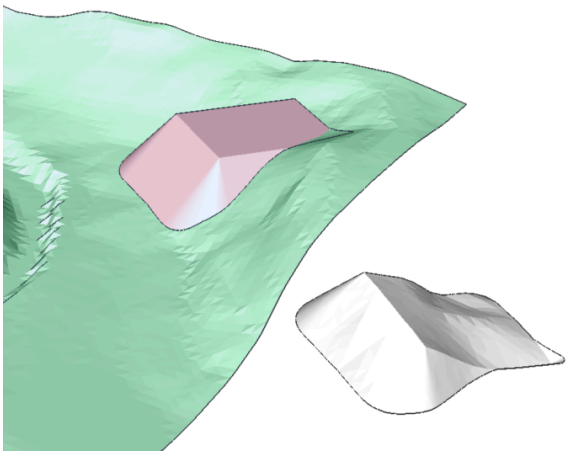
2 Choose a base surface. In this example, we choose the green TIN surface as the base surface.



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

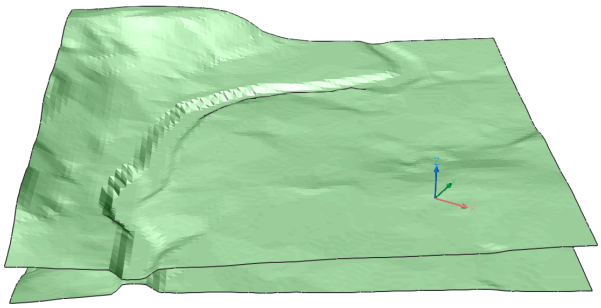
1 Launch the TINVOLUME command.

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.

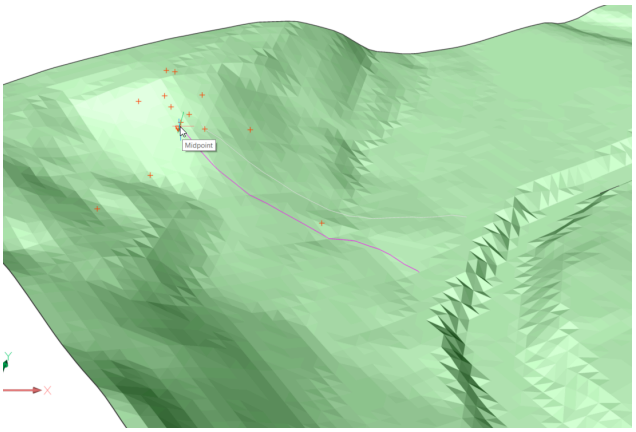
9.10.8 Procedure: Creating a water drop path

1 Launch the TINWATERDROP command.

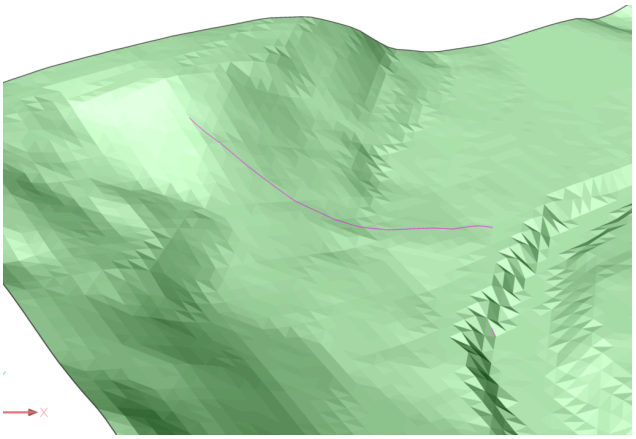
2 Select the TIN surface.

3 Move the cursor over the surface, a line representing the drop path is shown in real-time.

Uwaga: 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.

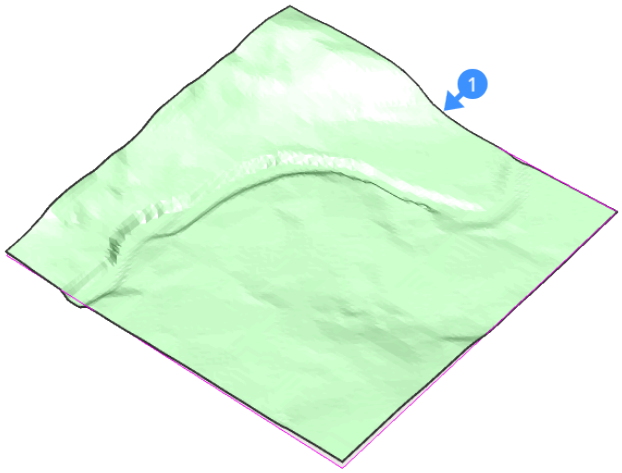
Two projection methods **Slope to surface** and **Offset to slope** are supported:

- **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).

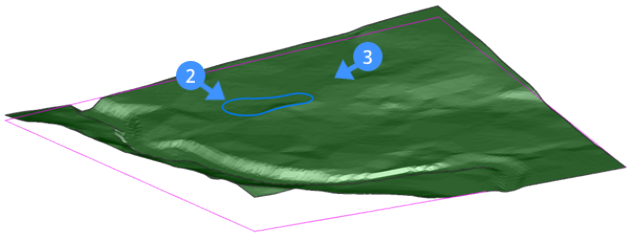


2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your pond.

Uwaga: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

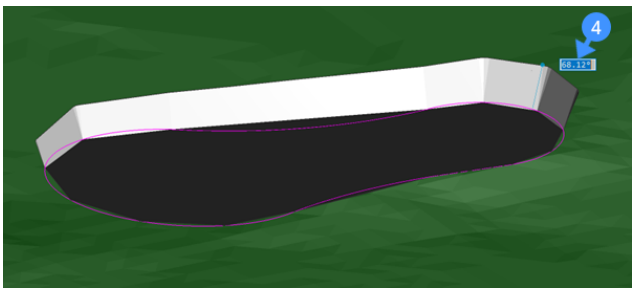
4 Select the target surface.



5 Define the slope of the grading.

Uwaga: 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.

Uwaga: 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.

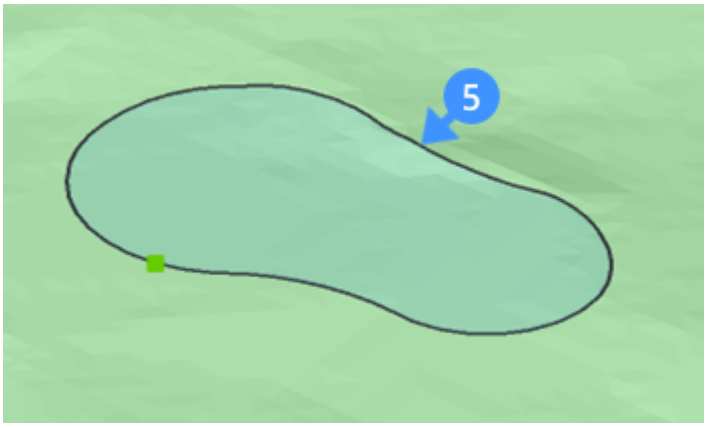
Infill

The internal are of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

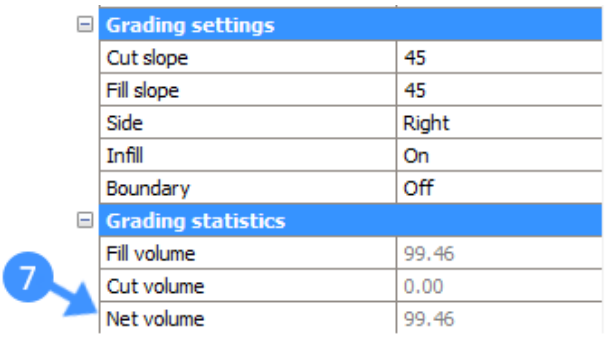
Boundary

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).







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

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).

Uwaga: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

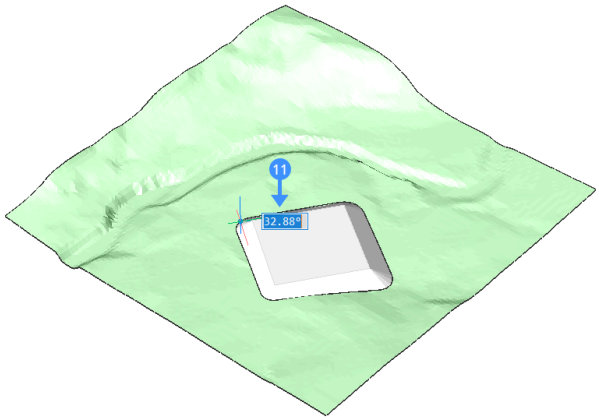
4 Select the target surface (10) to create a graded area.



5 Define the slope of the grading.

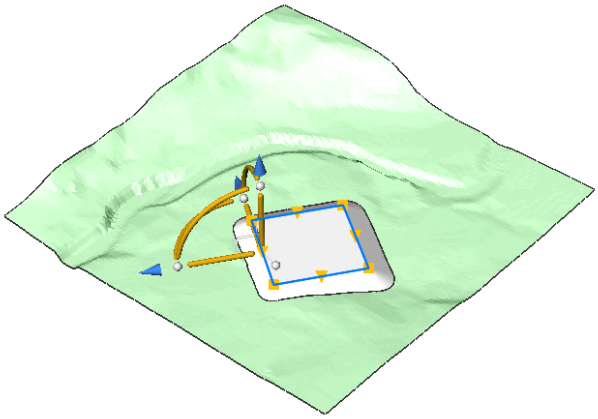
Uwaga: Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Uwaga: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (11).

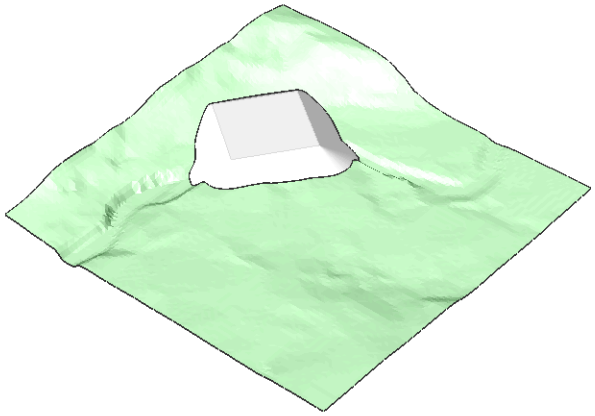


6 Press ENTER to accept the slope of the grading.

Uwaga: 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.

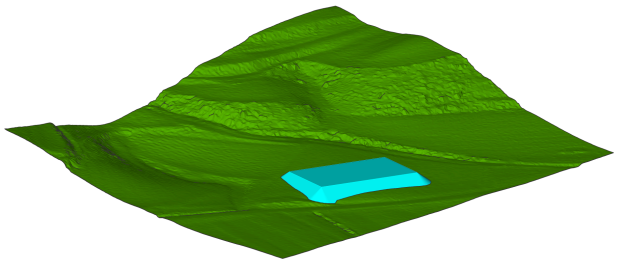


9.11.6 About Balance Grading Volumes

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

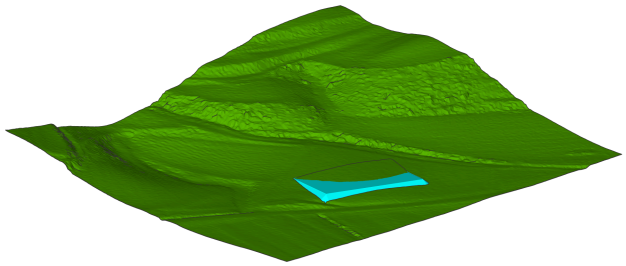
1 Open a drawing that contains a TIN surface and a grading representing a building pad.



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

Uwaga: 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.

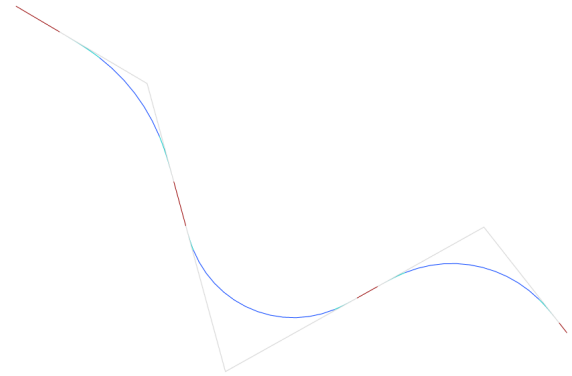
9.12.3 About ALIGNMENT

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

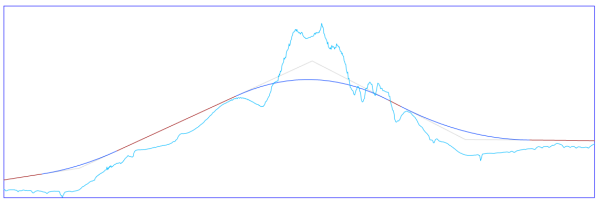
Horizontal alignment is a 2D curve that is needed to precisely position linear objects in a horizontal direction. It becomes defined by the Points of Intersection (PI). Horizontal alignment is the horizontal trace on

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.



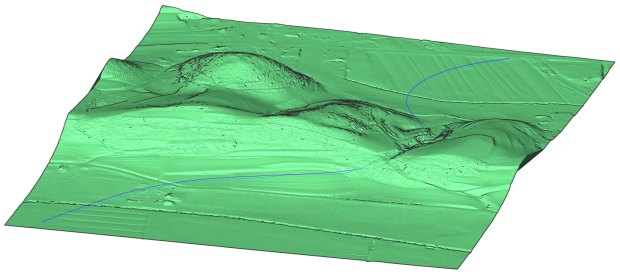
Vertical alignment

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

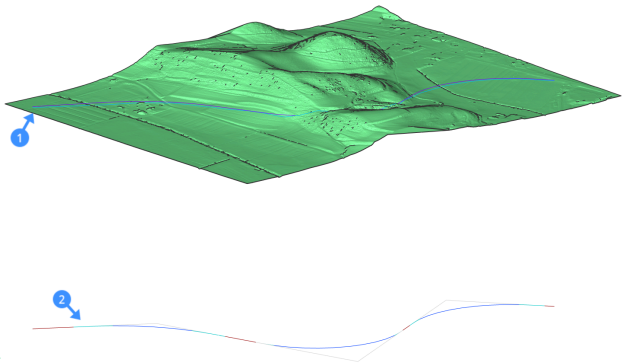
1 Open the drawing file that contains a TIN Surface.

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.



Uwaga: 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

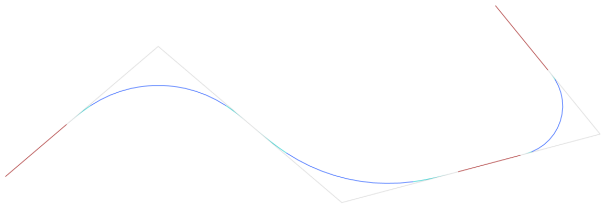
1 Open an empty drawing and launch the ALIGNMENT command in the Command line.

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.

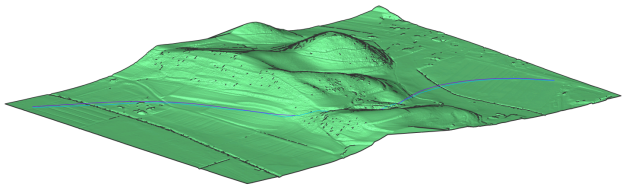


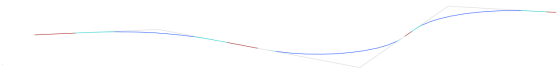
9.12.6 About Vertical Alignment View

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.



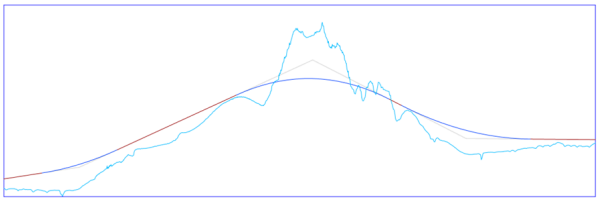


2 Launch the ALIGNMENTVIEW command in the Command line.

3 Select a horizontal or 3d alignment in the drawing.

4 Pick a point as origin for the vertical alignment view.

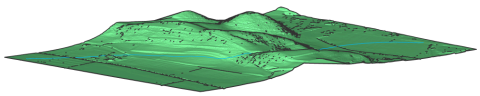
Your vertical alignment view is now generated.

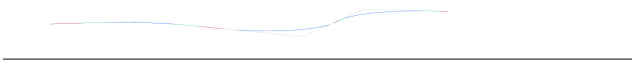


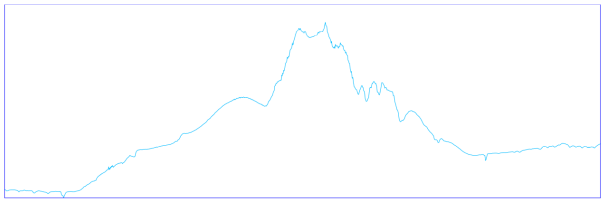
Uwaga: Two vertical alignments are shown. Light blue represents terrain line from the TIN Surface, and another is design vertical alignment. The design vertical alignment has straight gradients and vertical curves and is created automatically based on the accuracy factor. You can change the accuracy 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.



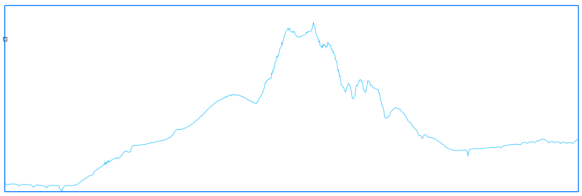




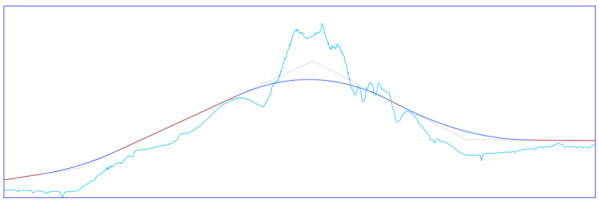
2 Launch the ALIGNMENT command in the Command line.

3 Select the **Vertical** option in the Command line.

4 Select vertical alignment view in the drawing.

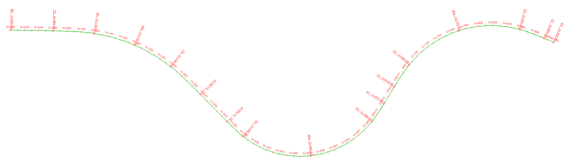


5 Pick the vertical alignment PVI points. Press ENTER when done. If you want to go one step back, use Undo. Automatic 3d alignment option generates design vertical alignment automatically. Your design vertical alignment is now generated.



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.

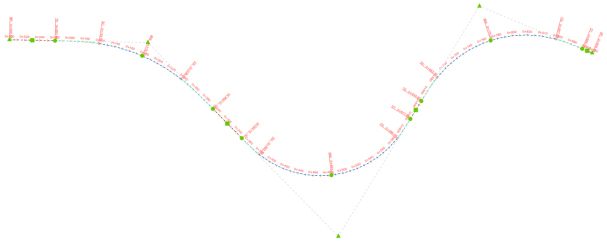


2 Launch the ALIGNMENT command in the Command line.

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

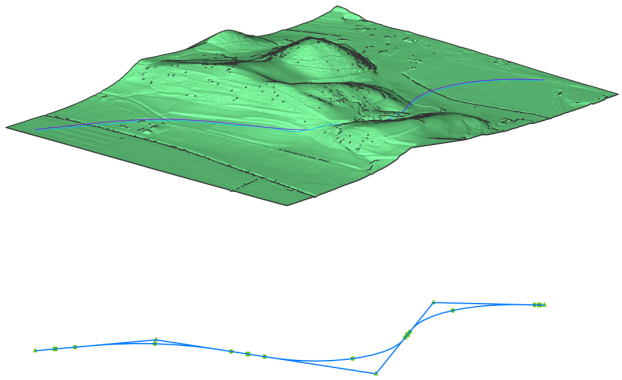
After creating an alignment, you can continue a horizontal or vertical alignment, add or delete PV/PVI points or change the TIN Surface using the command ALIGNMENTEDIT. The 3D alignment will be updated automatically.

9.12.11 Procedure: Continuing a horizontal alignment with ALIGNMENTEDIT

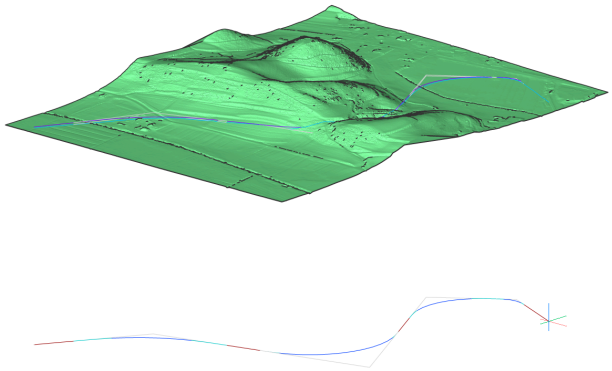
1 Draw a horizontal alignment following the steps from the Procedure: Creating an alignment.

2 Type ALIGNMENTEDIT in the Command line.

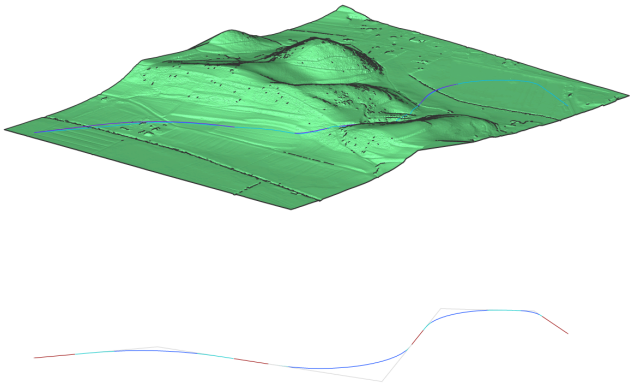
3 Select the horizontal alignment you want to edit.



4 Select points on your surface to continue your existing alignment.

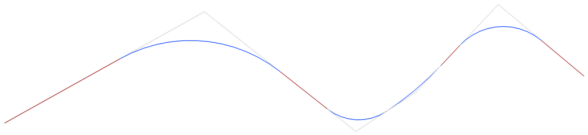


5 Press enter when finished. You have now continued your horizontal alignment. The 3D alignment will be updated automatically.



9.12.12 Procedure: Adding PI to an existing horizontal alignment

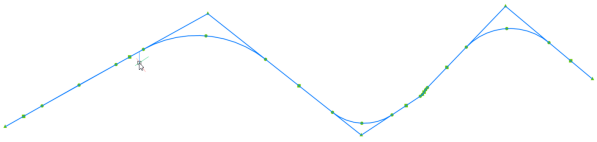
1 Type ALIGNMENTEDIT in the Command line.



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].

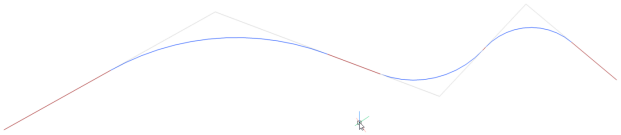
3 Select **Add Pi** in the Command line.

4 Pick point to add new PI.



9.12.13 Procedure: Removing PI to an existing horizontal alignment

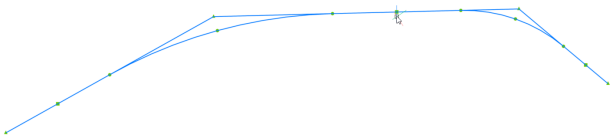
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit.

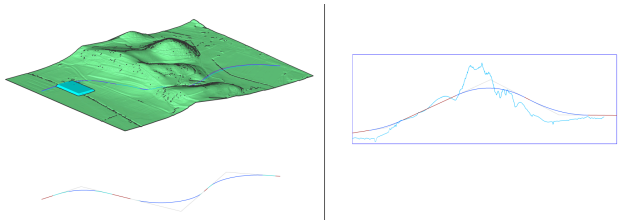
3 Select the **Remove Pi** option in the Command line.

4 Pick a PI point to remove.



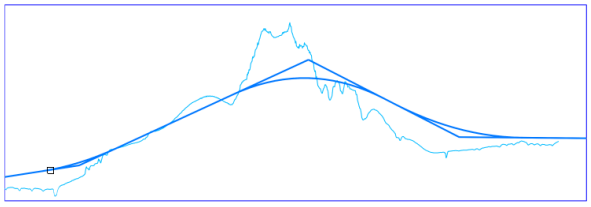
9.12.14 Procedure: Adding additional TIN Surface to Vertical alignment view

1 Open the drawing file that contains a TIN Surface, Alignments, Vertical Alignment View and grading.

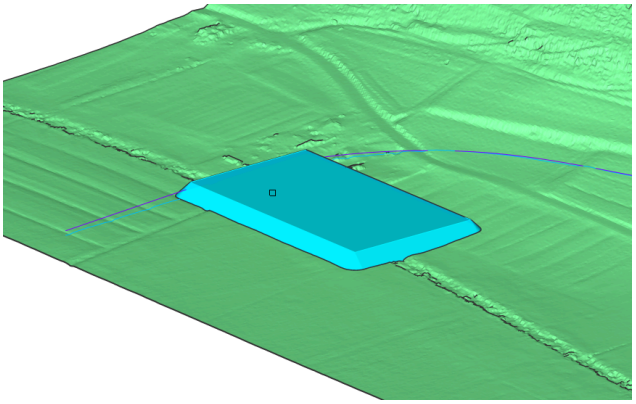


2 Launch the ALIGNMENTEDIT command in the Command line.

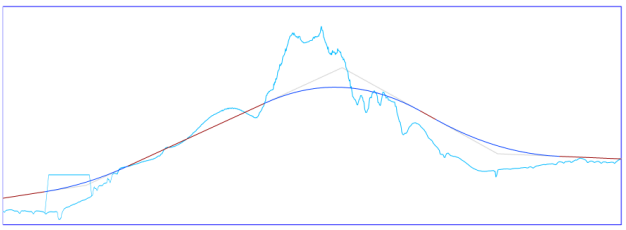
3 Select the **change Tin surface** option in the Command line.



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.

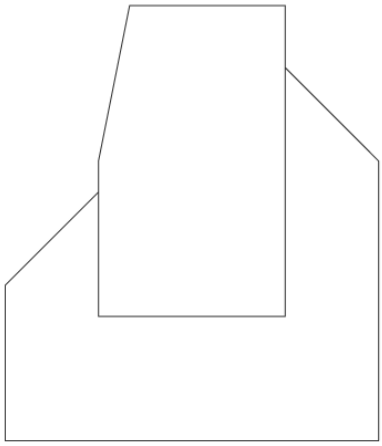
9.13.6 About Corridor Extract

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.

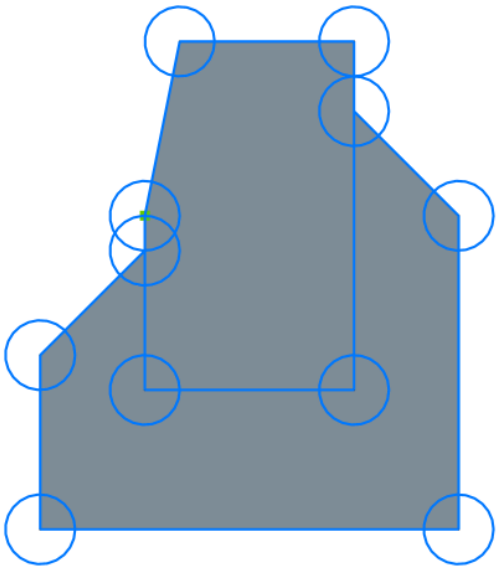
2 Draw closed polylines representing road kerb.



3 Launch the CORRIDORTEMPLATEELEMENT command in the Command line.

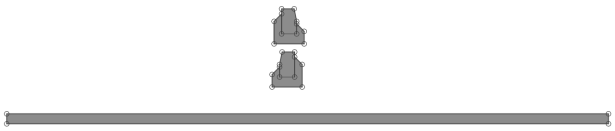
4 Select polylines representing kerb.

5 Pick one of the polyline vertices, the base point position can also be changed afterward.



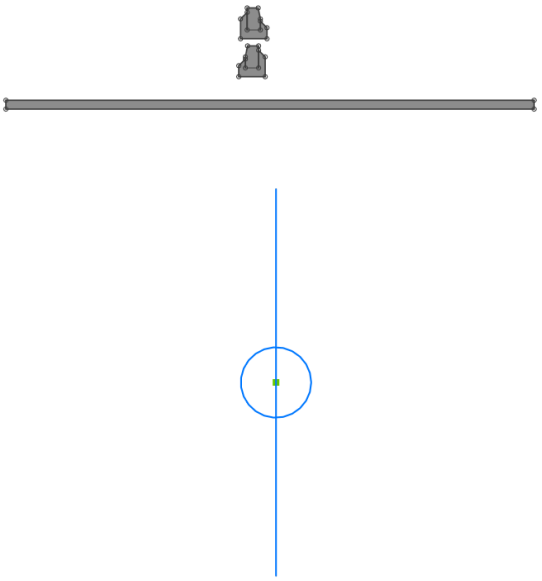
9.13.8 Procedure: Creating a template of road cross-section with asphalt layer and left and right kerb

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.

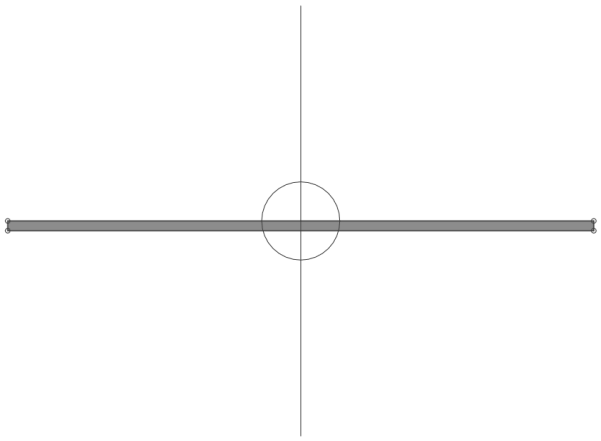


4 Launch the CORRIDORTEMPLATE command in the Command line.

5 Select the **Add template element** option in the Command line.

6 Select the template element representing the asphalt layer.

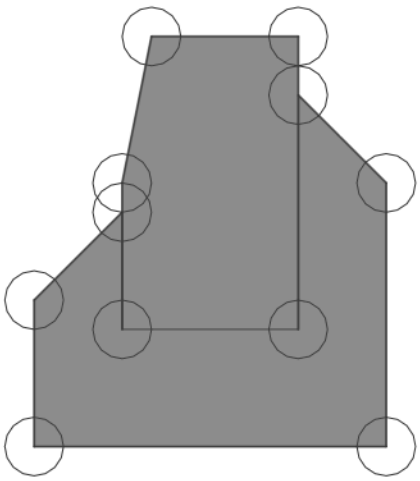
7 Select corridor template or template element point: (select the template)



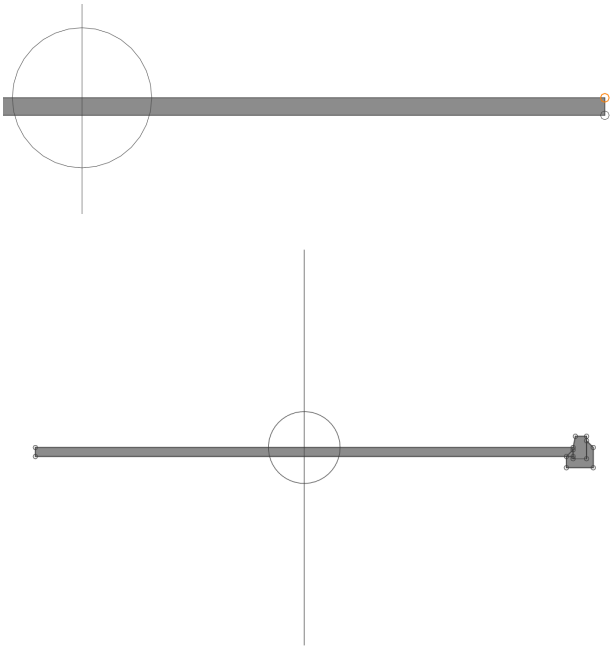
8 Launch the CORRIDORTEMPLATE command in the Command line.

9 Select the **Add template element** option in the Command line.

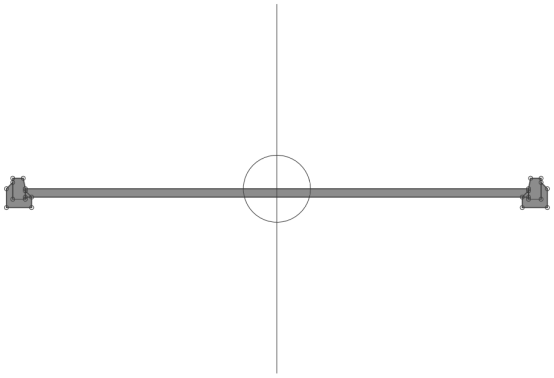
10 Select right top template element point of asphalt layer.



11 Select template element representing right kerb.

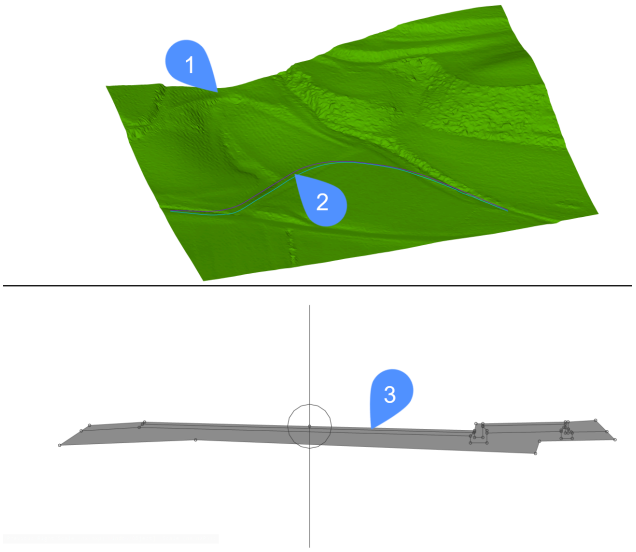


12 To add a kerb to the left side, repeat the steps described in points 9, 10, 11. By selecting the left kerb and left top point of asphalt layer this time.



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).



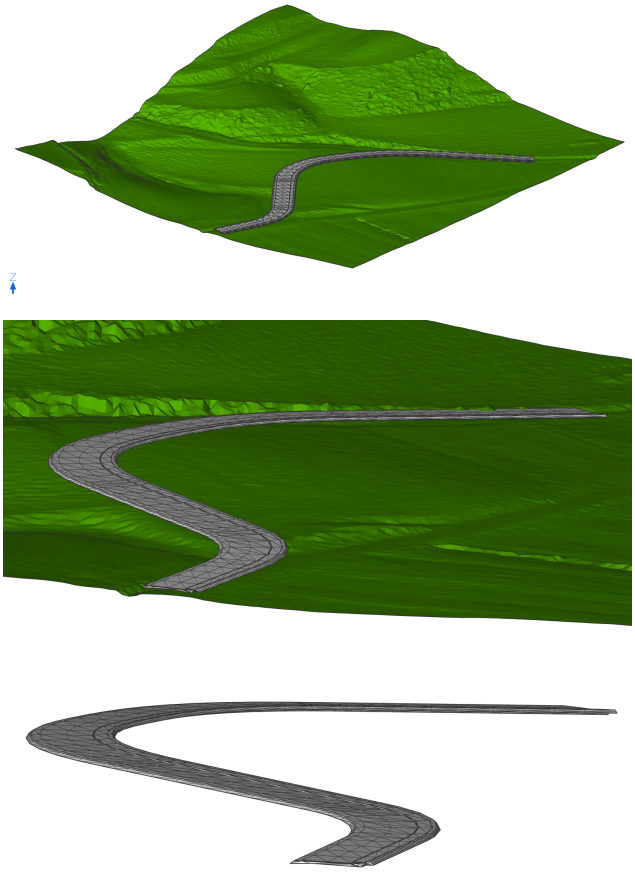
2 Launch the CORRIDOR command in the Command line.

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.



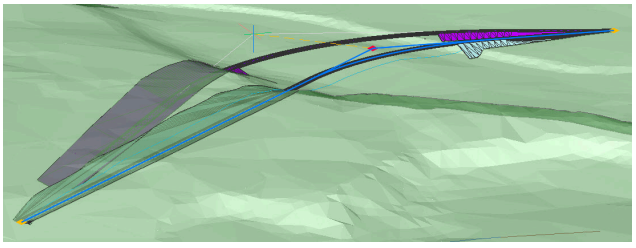
9.13.10 About Corridor Edit

The CORRIDOREDIT command adds or removes regions from an existing Corridor.

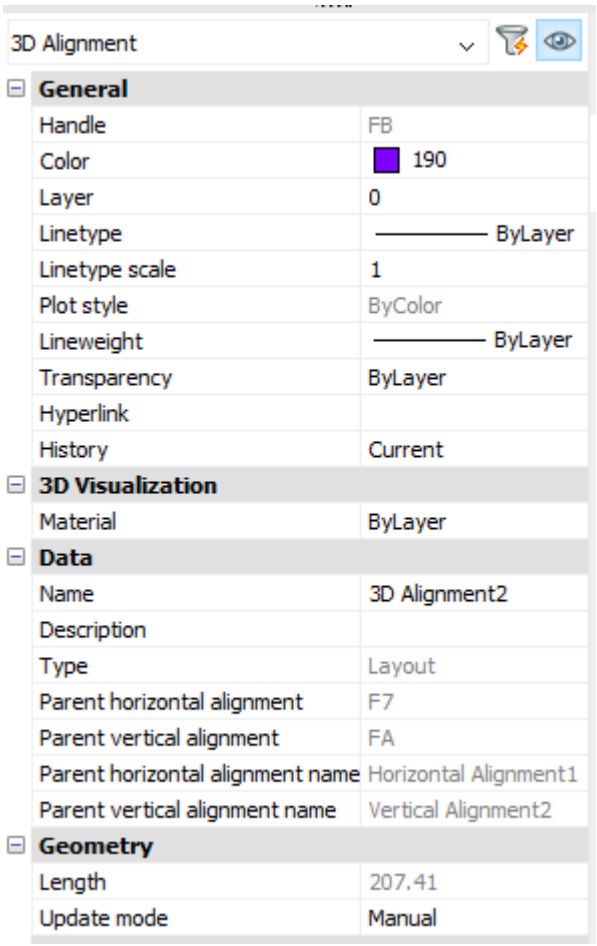
9.13.11 Procedure: Editing a corridor

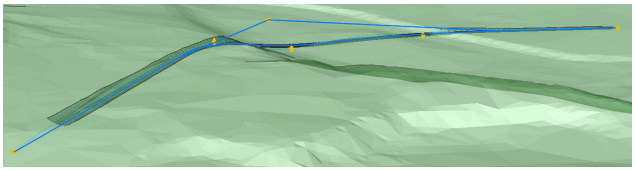
1 Open the drawing file that contains the corridor you would like to edit.

2 Select the 3D alignment that the corridor has as a reference and move the grip points to edit the corridor horizontally.

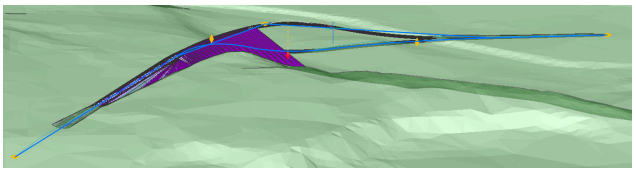


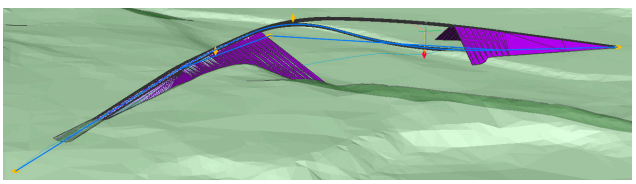
3 To edit the corridor vertically, select the 3D alignment that the corridor has as a reference and open the Properties Panel. Set the **Update mode** to Manual. Extra grip points will appear.



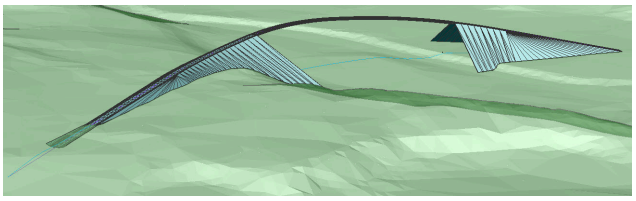


4 Move these grip points to edit the corridor vertically.





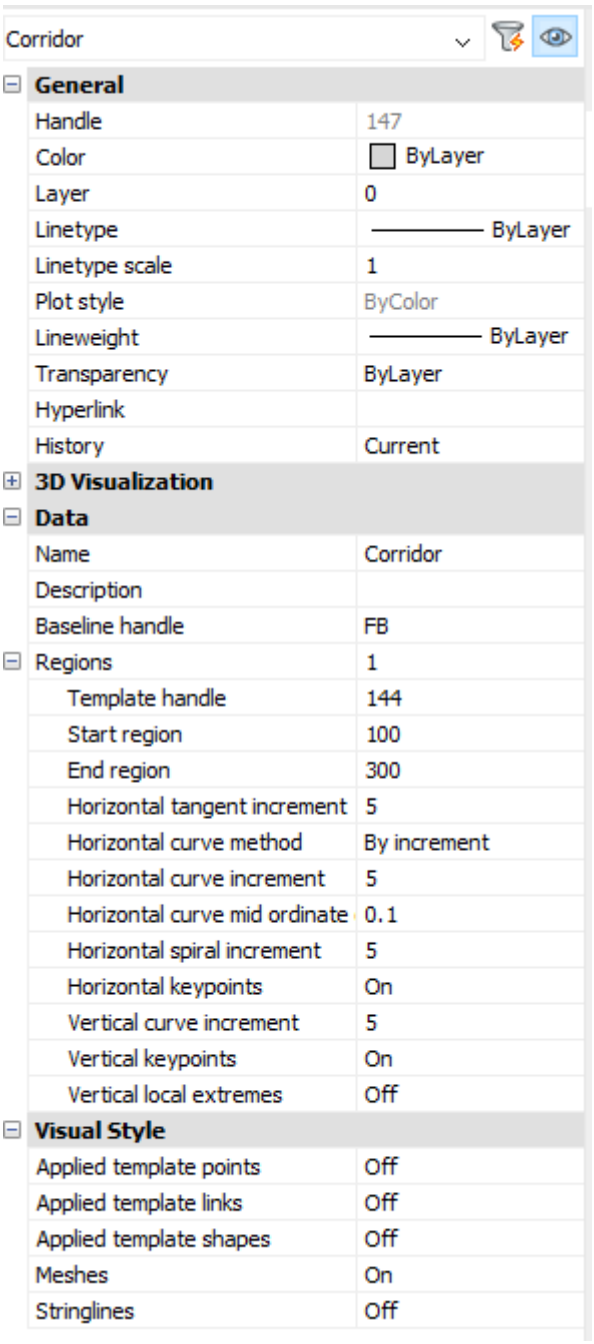
5 Press ENTER and you have finished your edited corridor.



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.

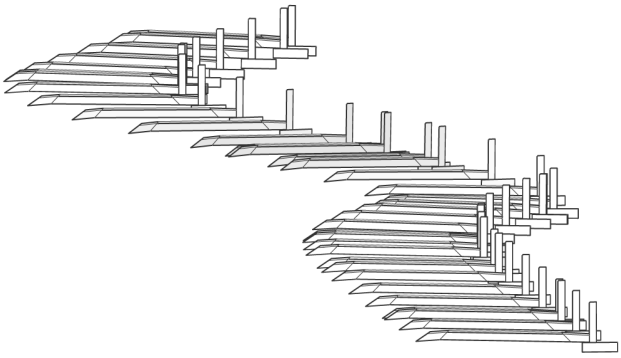


Applied template points

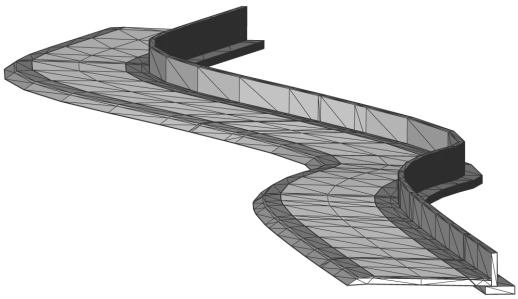
Represents the corridor with template element points.

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. Parametric modeling

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.

Uwaga: 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.

Uwaga: 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.

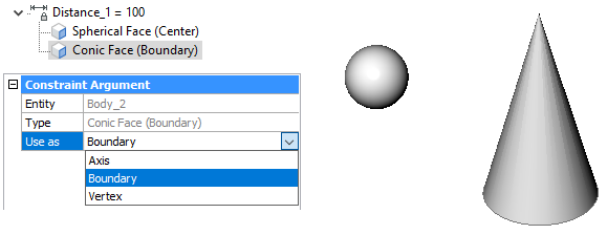
  Any option means that the directions may change upon direct modeling operations or upon a 3D constraint parameter change.

  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.

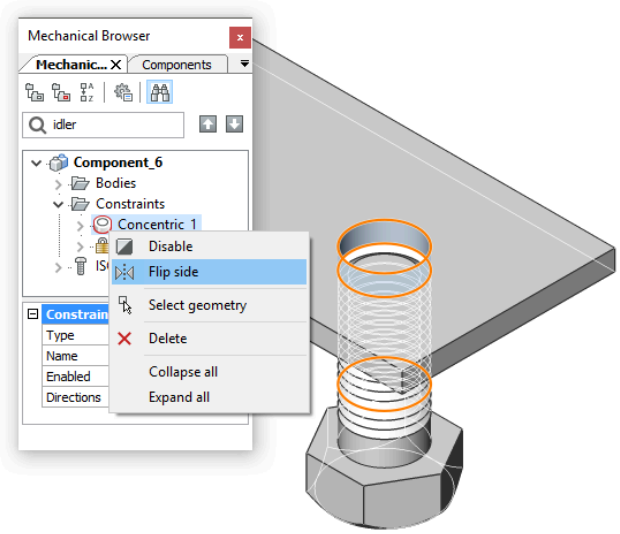
Main Node

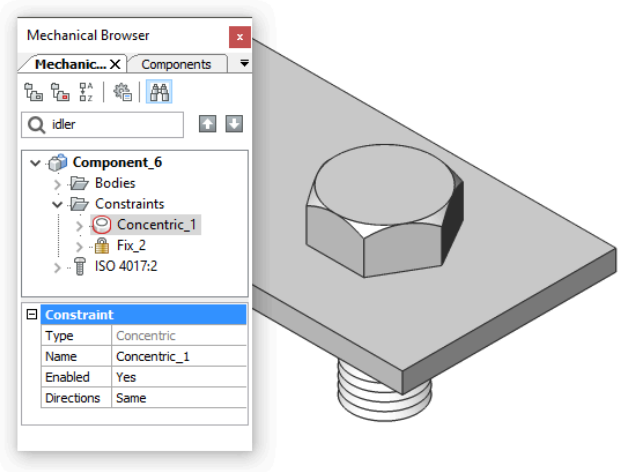
- **Add new parameter**: Creates a new parameter.
- **Collapse All**: Collapse all nodes.
- **Expand All**: Expand all nodes.

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).

Uwaga:

- 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'.

Uwaga: 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. Point cloud

11.1 Working with Point cloud

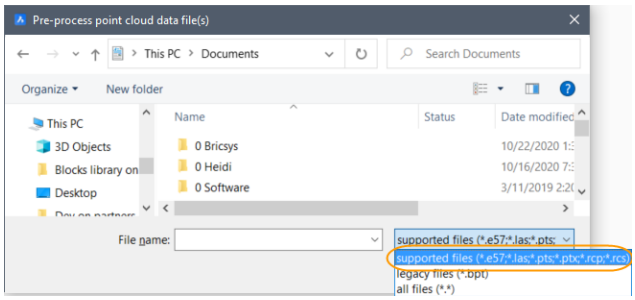
11.1.1 Commands

,,,,,,,, ,, ,,,,,,, ,, ,,

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.

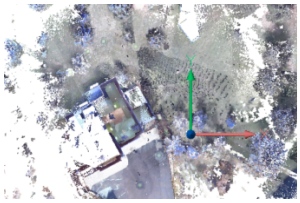


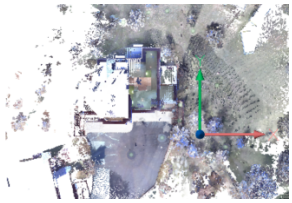
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD v21 than in v20.

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

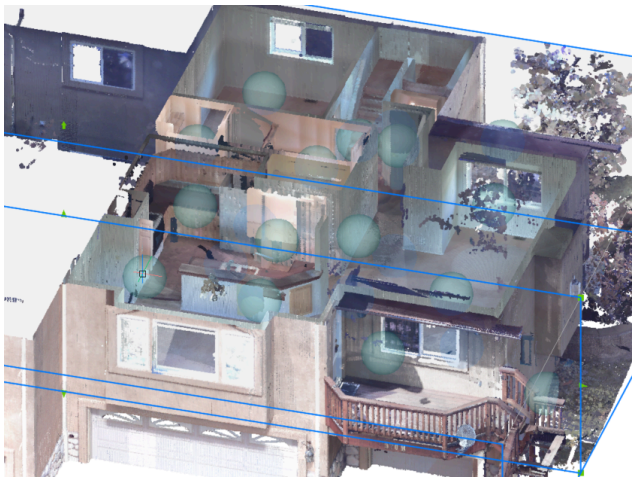
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment, it can analyze the entire point cloud, or you can specify the most relevant area.





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.



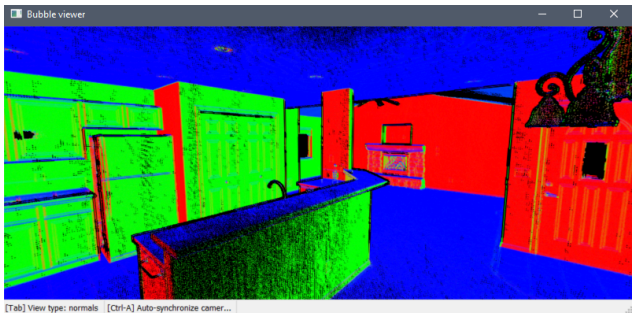
Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location. You can also zoom in and out using the mouse wheel.



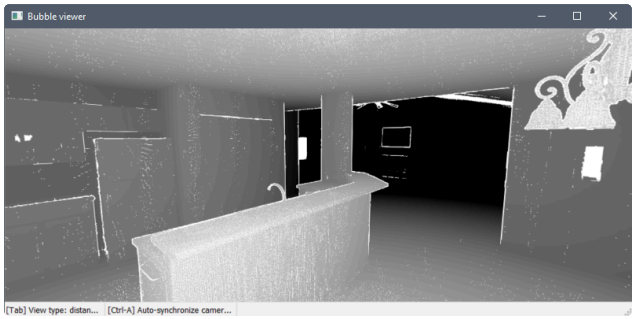
Press the Tab key to cycle between three different visual modes.

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.



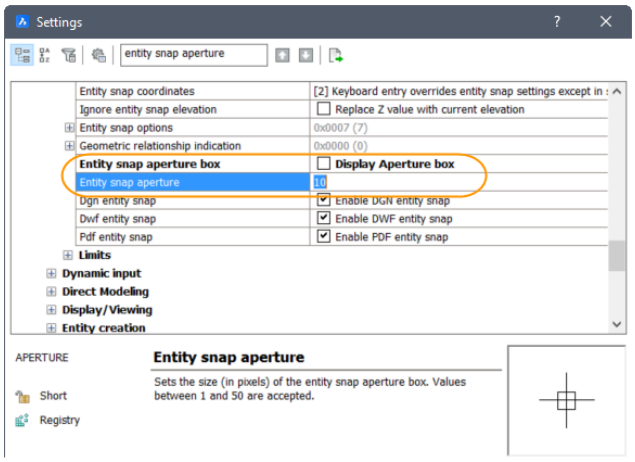
11.1.5 Entity snaps

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.



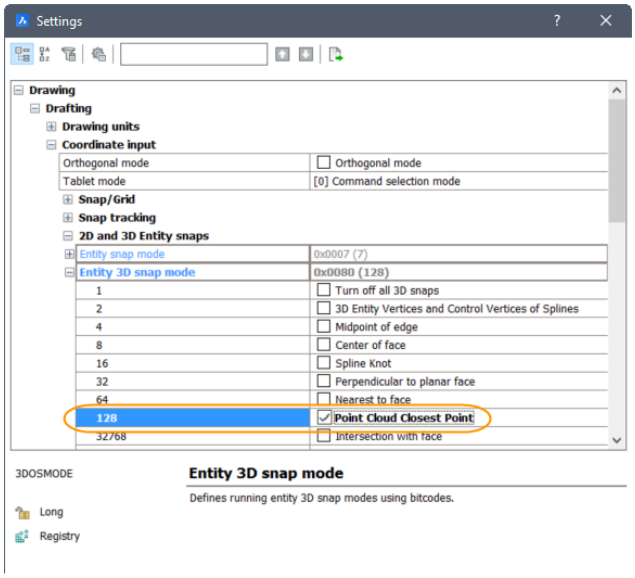


The radius of the imaginary cylinder is defined by the Entity snap aperture box setting.



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

Uwaga: Make sure that only the bit 128 of 3DSMODE is set.



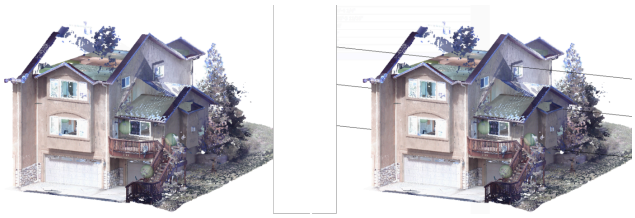
11.1.6 Export

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

11.1.7 Floor Detection

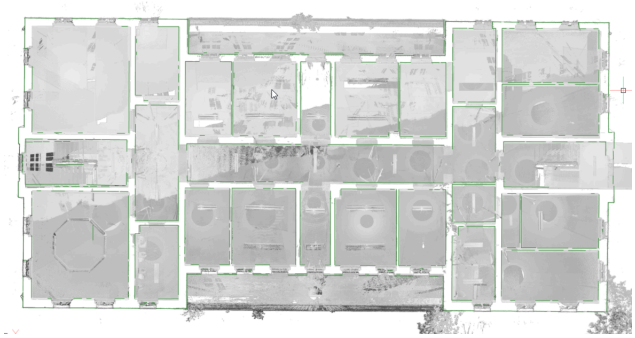
SectionPlanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and SectionPlanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

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.

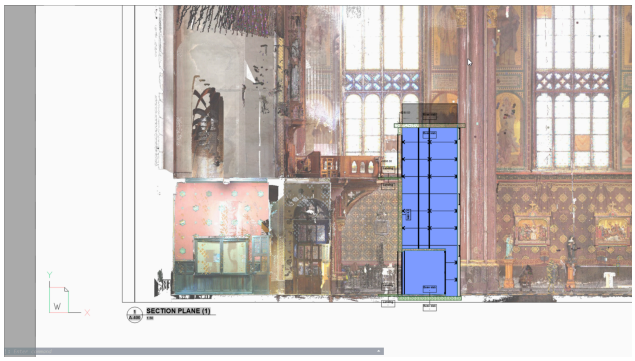


11.1.8 Point cloud projection

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.

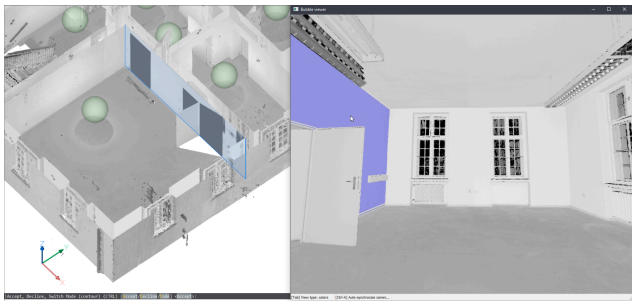


11.1.9 Planar Fit

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

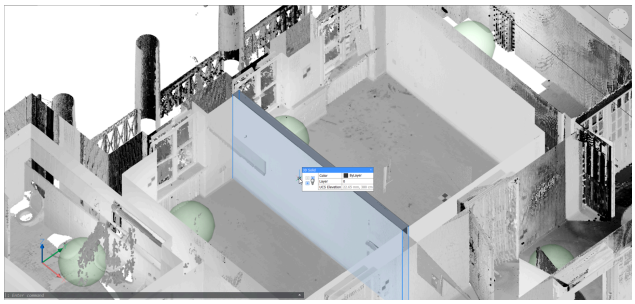
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the 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. Bloki, atrybuty, odnośniki zewnętrzne i komponenty

12.1 Creating blocks

The BLOCK command groups existing entities into a block using a dialog box.

The -BLOCK command creates blocks from existing entities and prompts you in the command bar.

The BEDIT and BCLOSE commands create or edit blocks in a temporary drawing session.

The WBLOCK command writes blocks and other drawing parts to a separate drawing file (short for "write block").

The PASTEBLOCK command pastes entities from the **Clipboard** as a block in the current drawing.

The SECTIONPLANETOBLOCK command saves the selected section plane result to a 2D cross section / elevation block or a 3D cutaway section block.

12.1.1 To create a block

1 Optionally select entities for the block.

2 Launch the BLOCK command.

3 The **Block Definition** dialog box appears:

4 Type a name for the new block in the **Name** field.

5 Do one of the following to specify the **Base Point**:

- Check the **Specify On-screen** option.

  You will be prompted to specify the base point after clicking the **OK** button.

- Click the **Pick point** button ( ).

  The **Block Definition** dialog closes temporarily to let you pick a point in the drawing.

- Type the coordinates of the base point in the X-, Y- and Z-field.

6 The number of preselected entities in step 1 is indicated.

7 To select the more entities for the new block, do one of the following:

- Select the entities before starting the BLOCK command.
- Check the **Specify On-screen** option.

  You will be prompted to select the entities after clicking the **OK** button.

- Click the **Select entities** button ( ).

  The **Block Definition** dialog closes temporarily to let select the entities in the drawing.

8 Set the block properties:

- **Annotative**: creates an annotative block.
- **Match block orientation to layout**: if checked, the orientation of an annotative block is preserved in a rotated viewport.
- **Scale uniformly**: if checked, the X-, Y- and Z-scaling cannot be set differently.
- **Allow exploding**: if not checked, prevents the block from being exploded.
- **Block unit**: sets the insertion unit for the block (see also: Inserting Blocks).

9 Set the behavior of the selected entities after the block is created:
- **Retain**: The selected entities remain in the drawing.
- **Convert to block**: The selected entities are replaced by the newly created block.
- **Delete**: The selected entities are deleted.

10 Click the **OK** button to create the block.

Uwaga:

- If the **Delete** option in step 6 was set, the OOPS command brings back the erased block entities.
- If you type a name of an existing block in step 3, you will be prompted to overwrite the existing block definition. Existing block instances will be replaced with the new block definition.

12.1.2 To save a block to a separate file

1 Optionally select entities for the block.

2 Launch the WBLOCK command.

3 The **Write Block to File** dialog box opens.

4 Click the browse button next to the **Name and path** field.

The **Save Block** dialog box is displayed.

5 Select a folder and type a name in the **File name** field, then click the **Save** button.

The **Save Block** dialog closes.

6 **Insert units**: sets the value of the INSUNITS system variable for the block file.

By default this is the INSUNITS value of the source drawing.

7 The number of preselected entities in step 1 is indicated.

8 To select the more entities for the new block, do one of the following:
- Click the **Block** radio button, then select a block from the drop-down list.
- Click the **Entire Drawing** radio button, to select all the entities in the model space of the current drawing.
- Click the **Entities** radio button, then click the **Select Entities** button.

  The **Write Block to File** dialog temporarily closes to let you select entities in the drawing.

  Right click to stop selecting entities.

9 Click the **Pick point** button () to define the **Base point** of the new drawing

or

Type the coordinates of the base point in the X-, Y- and Z-field.

10 Set the behavior of the selected entities after the block is created:
- **Retain**: The selected entities remain in the drawing.
- **Convert to block**: The selected entities are replaced by the newly created block.
- **Delete**: The selected entities are deleted.

11 Click the **OK** button to create the drawing file.

Uwaga: If the **Delete** option in step 7 was set, the OOPS command brings back the erased block entities.

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

Uwaga: 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 Edycja bloków

Dostęp do Edytora bloku można uzyskać za pomocą polecenia BEDIT lub przez dwukrotne kliknięcie bloku, który chcesz edytować. Używając Edytora Bloku, BricsCAD wyświetla elementy bloku w tymczasowej sesji rysowania. Zawiera wszystkie narzędzia potrzebne do tworzenia i edycji definicji bloku. Aby wyjść z sesji edycji bloku, użyj polecenia BZAMKNIJ i wybierz, czy zapisać, czy odrzucić zmiany. Jeśli wybierzesz polecenie ZAPISZ podczas sesji edycji bloku, BricsCAD zapisze zmiany w definicji bloku i wyjdzie z edytora bloku.

Bloki można również edytować za pomocą polecenia ODNEDYCJA.

12.4 Inserting blocks

The INSERT command lets you insert blocks through a dialog box.

The -INSERT command inserts blocks by prompting in the Command line. BricsCAD searches the block name in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the SRCHPATH system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'.

The INSERTALIGNED command inserts a block entity, with easy alignment on existing entities. The INSERTALIGNED command is similar to the INSERT command, but during the placement the block

dynamically aligns with existing lines, polylines, arcs or circles. The block aligns with the entity to which the cursor snaps. If no entity is snapped, the block is aligned with the X-axis of the WCS.

The MINSERT command inserts a block as a rectangular array; combines the -INSERT and ARRAY commands (short for "multiple insertion").

You can choose to select an existing block definition or to insert an entire drawing as a block. When you insert a drawing, a new block definition is created in the current drawing. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing. When a block is inserted in a drawing, it is treated as a single entity.

Uwaga:

- When inserting a drawing as a block the origin is defined by the INSBASE system variable of the inserted drawing. The variable is set by the BASE command.
- If a block contains attributes, you will be prompted to fill out the text for each attribute in the Command line.
- If the DRAGOPEN system variable is set to zero (OFF), drawings can be inserted as a block by dragging them from the **Windows Explorer** dialog into the current drawing; if DRAGOPEN = 1 (ON), the drawing will be opened instead.
- The INSUNITS and INSUNITSSCALING system variables control the automatic scaling when inserting blocks or attaching Xrefs when the value of INSUNITS is different in the block or Xref and the target drawing.
- If the value of INSUNITS in the source drawing is 0 (Unspecified), the value of the INSUNITSDEFSOURCE system variable is used instead.
- If the value of INSUNITS in the target drawing is 0 (Unspecified), the value of the INSUNITSDEFTARGET system variable is used instead.
- The values of INSUNITSDEFSOURCE and INSUNITSDEFTARGET are saved in the registry and therefore apply to all drawings in which the value of the INSUNITS system variable is not specified (unitless).
- When you type the name of a block at the ‘Block to insert:’ prompt, BricsCAD will look up the block in the block definitions in the current drawing. If the block is not found there, the paths defined by the SRCHPATH system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'.

In the **Insert Block** dialog box:

- The **Unit** field under **Block Unit** is set by the INSUNITS system variable in the source drawing or the **Block Unit** property of a block definition (see Creating Blocks).
- The **Factor** field expresses the relation between the value of the INSUNITS variable in the source drawing and the target drawing. E.g. if INSUNITS is Millimeters in the source and Centimeters the target, the value of the **Factor** field is 0.1.

12.4.1 Using the INSERT command

1 Launch the INSERT command.

1 The **Insert Block** dialog box appears.

2 Do one of the following:

- Select an existing block definition in the **Name** list.

  The name of the selected block is displayed in the **Name** field.

- Click the **Browse** button and select a drawing file.

  The path field shows the location of the selected drawing.

3 (option) Check the **User reference curves for insert** option.

4 Set the **Insertion Point**, **Scale** and **Rotation** modes.

The options are:

- Specify the settings in the **Insert Block** dialog.
- **Specify On-screen**: You will be prompted during the insertion process.

5 (option) Check the **Explode** option to explode the block upon insertion.

6 Click the **OK** button.

You will be prompted in accordance with the selected options.

12.4.2 To insert a block using the Drawing Explorer dialog box

Inserting blocks from the current drawing.

1 Launch the EXPBLOCKS command.

1 The **Drawing Explorer – Blocks** dialog box appears.

2 (option) Set the view mode: **Detail** or **Icon**.

3 (option) Select a block, then right click and choose **Options…** in the context menu.

The **Drawing Explorer Options** dialog box appears.

Set the insertion options, then click the **OK** button.

4 Double click the block icon or the number when in **Detail View** mode.

The **Drawing Explorer** dialog closes to let you insert the block in the drawing.

5 Specify the insertion point.

- Depending on the current insertion options, you are prompted to specify the scale and/or rotation.
- When the **Align** option is selected, you must specify the position of the block with respect to the selected linear entity.

  You are prompted: Enter block mirroring control point:

The **Drawing Explorer** dialog box reopens.

Inserting blocks from another drawing.

1 Open the **Drawings Explorer** dialog box.

2 In the **Drawings** pane, click the **Folders** tab.

3 (option) If no folder are available yet in the **Local Folders** tree, or if you want to add a folder, click the **Add local folder...** button.

The **Choose a folder** dialog box appears.

Browse to the folder you want to add, then click the **Select Folder** button.

The folder and its subfolders are added in the **Local Folders** tree.

4 Select a drawing in the **Local Folders** tree.

5 Expand the drawing, then select **Blocks**.

All blocks in the selected drawing are displayed in the **Blocks** pane.

6 Proceed from step 2 in the previous procedure.

12.4.3 To insert a block aligned with an entity

1 Make sure the appropriate Entity Snaps are active.

2 Launch the INSERTALIGNED command.

3 **You are prompted**: Block to insert (~ to open file dialog) or [? to list blocks in drawing] <current block>:

4 Do one of the following:

- Press Enter to accept the currently selected block.
- Type the name of the block to insert.
- Press the ? key to see a list of available blocks.

You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Multiple blocks]:

5 (option) Choose the **Multiple blocks** option or specify a fixed scale.

6 Specify a point.

7 The block displays the selected entity snap point, aligned with the linear entity.

You are prompted: Enter block mirroring control point:

8 Move the cursor to select the desired position, then click to insert the block.

Uwaga: The block is aligned as follows:

The block is rotated so that its local Z-axis is parallel with the Z-axis of the active UCS, and its local X-axis should be parallel with the tangent to the entity at the position of the snap point. When you click a point on an entity, the block is aligned with the entity. Then, by moving the mouse pointer around the insertion point, you can choose how the block should be mirrored. The origin point of the block lies on the entity.

12.4.4 To insert multiple instances of a block in a rectangular array

1 Launch the MINSERT command.

1 **You are prompted**: Block to insert (~ to open file dialog) or [? to list blocks in drawing] <current block>:

2 Do one of the following:

- Press Enter to accept the currently selected block.
- Type the name of the block to insert.
- Press the ? key to see a list of available blocks.

You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Multiple blocks]:

3 (option) Choose the **Multiple blocks** option or specify a fixed scale.

4 Specify a point.

5 **You are prompted**: Rotation angle for block <0°0'0">:

6 Press Enter to accept the default rotation angle or specify an angle.

7 **You are prompted**: Number of rows in the array <1>:

8 Specify a number.

9 **You are prompted**: Number of columns <1>:

10 Specify a number.

11 **You are prompted**: Vertical distance between rows, or spacing rectangle:

12 Do one of the following:

- Specify a value.
- **You are prompted**: Horizontal distance between columns:

  Specify a value.
- Click two point to specify a rectangle, which sets the horizontal and vertical spacing distances.

The block is inserted as a **Minsert Block** entity.

Uwaga:

- When you choose **Multiple blocks** in step 3, you are prompted to create multiple instances of the block array.
- It is not possible to explode a **MInsert Block** entity.
- You can edit the properties of a **MInsert Block** entity in the **Properties** panel.

12.5 Eksplodujące bloki

Jeśli **Rozbicie** właściwość bloku jest ustawiona, można rozbić wstawiony blok na jego oryginalne elementy składowe. Rozbicie bloku wpływa tylko na to pojedyncze wystąpienie bloku. Oryginalna definicja bloku pozostaje na rysunku i nadal można wstawiać dodatkowe kopie oryginalnego bloku. Jeśli rozbijesz blok zawierający atrybuty, atrybuty zostaną utracone, ale oryginalne definicje atrybutów pozostaną.

Zmienna systemowa EXPLMODE określa, czy możliwe jest rozbicie bloków skalowanych nierównomiernie.

12.5.1 Aby rozbić blok

1 Uruchom polecenie ROZBIJ.

1 **Zostanie wyświetlony monit:** Wybierz podmioty rozbicia [opcje wyboru (?)]:

2 Wybierz jeden lub więcej bloków.

3 Jednostki w zestawie: x

Zostanie wyświetlony monit: Wybierz podmioty rozbicia [opcje wyboru (?)]:

4 Wybierz więcej elementów lub naciśnij klawisz Enter, aby zatrzymać wybieranie.

5 Wybrane bloki zostaną rozbite.

Uwaga: Rozbicie oddziela elementy składowe do następnego najprostszego poziomu złożoności: zagnieżdżone bloki lub polilinie w bloku stają się ponownie blokami lub poliliniami.

12.5.2 Aby ustawić właściwość Explodable bloku

ten **Wybuchowy** właściwość bloku jest ustawiana podczas tworzenia bloku za pomocą polecenia BLOK.

Aby edytować właściwość, wykonaj następujące czynności:

1 Uruchom polecenie EXPBLOCKS.

1 ten **Eksplorator rysunków / bloki** pojawi się okno dialogowe.

2 Wybierz **Widok Szczegółowy** tryb.

3 Kliknij **Eksplodować** zaznacz pole, aby przełączyć **Wybuchowy** własność.

12.6 bloki parametryczne 2D (wkrótce)

Jeśli pochodzisz ze środowiska projektowego AutoCAD®, być może znasz bloki dynamiczne. Możesz także znać wiązania parametryczne 2D. Są to dwie różne funkcje, które w pewnym stopniu się pokrywają.

Zamiast oferować funkcjonalność bloków dynamicznych, BricsCAD skupia się całkowicie na parametryzacji. Umożliwia tworzenie definicji bloków 2D i 3D przy użyciu tego samego zestawu narzędzi i przepływu pracy. Jeśli nigdy nie pracowałeś z ograniczeniami parametrycznymi, mogą brzmieć przerażająco. Ale są dość intuicyjne i logiczne. I niesamowicie potężny!

12.6.1 Ogólna procedura tworzenia bloków parametrycznych 2D

Utwórz geometrię.

Utwórz wiązania geometryczne.

Utwórz wiązania wymiarowe i parametry.

Edytuj wiązania wymiarowe i parametry w **Parametry i ograniczenia** panel lub **Przeglądarka mechaniczna** płyta.

Opcjonalnie utwórz stany widoczności, aby pokazać/ukryć geometrię, używając polecenia STANY WIDOCZNOŚCI.

Zapisz rysunek jako blok biblioteki, używając polecenia UTWÓRZBIBLIOTEK.

Uwaga: Polecenie PARAMETRIZE2D automatycznie tworzy wiązania geometryczne i wymiarowe.

12.6.2 Wstawianie bloku parametrycznego

Bloki parametryczne można wstawiać do rysunku za pomocą poleceń INSERT lub BMINSERT lub z menu **Biblioteka** płyta.

12.6.3 Ustawianie wartości parametrów

Wybierz blok parametryczny, a następnie zmień wartości wyeksponowanych parametrów w **Parametry** w **Nieruchomości** płyta.

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!

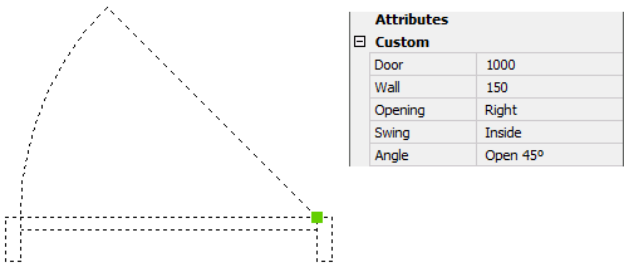
BricsCAD partly supports dynamic blocks, you can:

- 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

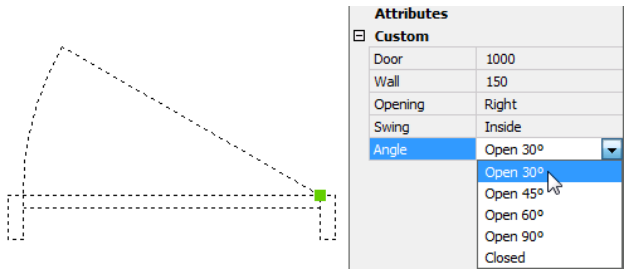
1 Select the dynamic block in the drawing.

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:

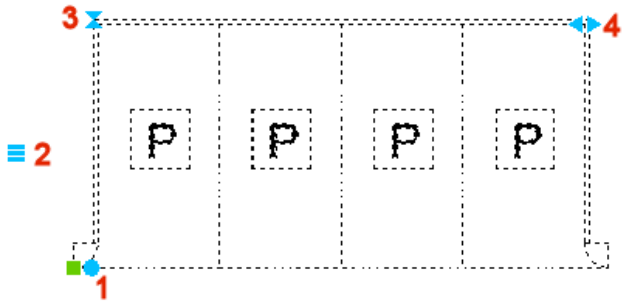
- Point
- XY

- 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 Defining attributes

The ATTDEF command defines attributes through a dialog box.

The -ATTDEF command defines attributes at the command prompt.

12.8.1 General procedure to define attributes

- Specify the characteristics of the attribute, including its name, prompt, and default value.
- Specify the insert coordinates.
- Define the text properties.
- (option) Set the attribute flags.

12.8.2 To define an attribute

1 Do one of the following:

- Click the **Attributes** > **Define Attributes...** tool button ( ) on the **Tools** toolbar.
- Choose **Define Attributes...** in the **Tools** menu.
- Launch the ATTEDEF command.

The **Define Attribute** dialog appears.

2 Define the Attribute:

- Type a name in the **Tag** field.

  The Tag identifies each occurrence of an attribute in the drawing. The name you type in the **Tag** field can be any combination of characters except spaces. Lowercase letters are automatically converted to uppercase.

- Type a prompt in the **Prompt** field.

  The attribute prompt appears when you insert a block containing the attribute. If a prompt is omitted, the Tag is used as the prompt instead.

- Do one of the following:

• Type a default value in the **Default** text field.

  For variable attributes, the default value is replaced by the actual value when you insert a block containing the attribute.

• Click the **Insert Field** button () to assign a field value to the attribute (see 'Using sheet set and sheet properties in a title block').

• If the **Multiple lines** flag is checked, the **Default** text field is dimmed.

• Click the button next to the **Default text** field.

  The **Define Attribute** dialog temporarily closes to let you create an Mtext entity.

- Do one of the following:

• Specify the X, Y, and Z coordinates for the attribute insertion point.

• Click the Pick Point button () to specify the insertion point in the drawing.

  You are prompted: Select insertion point.

  The **Define Attribute** dialog closes temporarily.

- (option) Set the **Invisible** attribute flag.

  Attributes with the **Invisible** flag set are neither displayed nor printed.

  If the ATTMODE (Attribute Display Mode) variable is set to **2**, all attributes are displayed, including **Hidden** attributes.

- (option) Set the **Constant** attribute flag.

  Attributes with the **Constant** flag set cannot be edited.

- (option) Set the **Verify** attribute flag.

  Attributes with the **Verify** flag set must be explicitly validated when a block containing such attributes is inserted.

- (option) Set the **Preset** attribute flag.

  When inserting a block, you are not prompted to define attributes of which the **Preset** flag is set. You can still edit the attributes afterwards.

- (option) Set the **Lock Position** attribute flag to lock the position of the attribute within the block reference.

  Unlocked attributes can be moved relative to the rest of the block using grip editing, and

multiline attributes can be resized.

- Define the **Text** properties: **Text Style**, **Justification**, **Annotative**, **Text height** and **Rotation**.

3 Click the **OK** button to create the attribute in the drawing.

The **Define Attribute** dialog closes.

12.8.3 Using sheet set properties in a title block

1 Create the sheet set.

2 Define the custom sheet set and sheet properties.

3 Open a sheet (layout) from the sheet set.

4 Create the title block source entities in the paper space layout:

- Create the borders, line work and fixed texts.
- Create attributes with fields that refer to sheet set, subset and sheet properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

Custom sheet set properties:

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

Sheet properties and sheet custom properties:

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 Create a block from the title block source entities.

6 Copy the block to a drawing in one of your block folders.

7 Delete the title block source entities.

The title block can be inserted in the other sheets of the sheet set and in any other sheet set where the same custom sheet set and sheet properties exist. All fields that refer to the default or custom sheet set, subset or sheet properties are filled out automatically. If no value is available, a field displays hyphens (----).

12.9 Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.

Attributes can be visible or hidden. Hidden attributes are neither displayed nor plotted, but the information is still stored in the drawing and written to a file when you extract it.

ATTDEF command - defines attributes through a dialog box.

- ATTDEF command - defines attributes through the Command line.

ATTDISP command - toggles the display of attribute text.

ATTEDIT command - edits attribute values and properties.

-ATTEXT command - extracts attributes from drawings to data files through the command bar.

ATTREDEF command - redefines a block and updates associated attributes.

ATTSYNC command - synchronizes attribute definitions in all block references of a specified block definition.

BATTMAN command - (short for "block attribute manager") edits all aspects of attributes in a block definition, and then optionally applies the changes to all blocks of the same name in the drawing.

EATTEDIT command - edits attribute values and most properties.

12.9.1 General procedure to work with attributes

1 Define the attributes.

2 Create a block.

You can attach attributes to a block. Include the attributes when the program prompts you to select the entities to be included in the block definition. If a block contains attributes, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

1 Insert the block in a drawing.

2 (option) Edit the attributes in a block.

3 Extract attribute information.

12.10 Edycja definicji atrybutów

Właściwości atrybutów można edytować w **Nieruchomości** płyta.

Polecenie ATTEDIT edytuje wartości i właściwości atrybutów.

Polecenie BATTMAN (skrót od „menedżer atrybutów bloku”) edytuje wszystkie aspekty atrybutów w definicji bloku, a następnie opcjonalnie stosuje zmiany do wszystkich bloków o tej samej nazwie na rysunku.

Uwaga: Atrybuty w definicji bloku można edytować w **Nieruchomości** panelu podczas używania poleceń BEDYCJA do edycji bloku.

12.11 Edycja atrybutów bloku

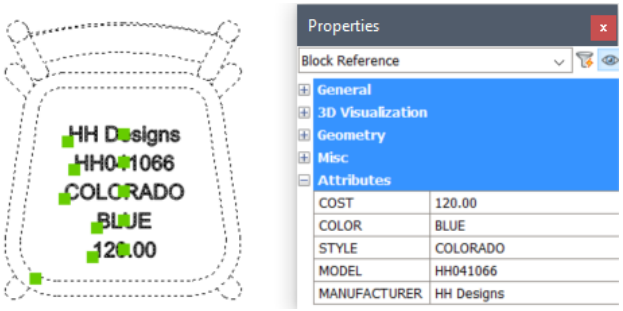
Atrybuty bloku można edytować w **Nieruchomości** płyta.

1 Wybierz blok zawierający atrybuty.

1 Aktualna wartość atrybutów jest wyświetlana w **Nieruchomości** płyta.

2 Wybierz atrybut, który chcesz edytować.

3 Pole ustawień wybranego atrybutu jest aktywne.



4 Wpisz nową wartość, a następnie naciśnij Enter lub wybierz inny atrybut.

12.12 Inserting blocks with attributes

When you place a block which contains attributes, you will be prompted to define the attributes in the Command line or through a dialog box, depending on the value of the ATTDIA system variable:

- ON: attributes are displayed in a dialog box.
- OFF: you are prompted in the Command line to define the attributes.

12.12.1 To place a block with attributes

1 Do one of the following:

- Click the **Insert Block** tool ( ) on the **Insert** toolbar or the **Insert / Blocks** ribbon panel.
- Choose **Insert Block** in the **Insert** menu.
- Launch the INSERT command.

The **Insert Block** dialog box appears.

2 Select the block, then click the **OK** button.

You are prompted: Insertion point for block or [Scale/X scale/Y scale/Z scale/Rotate/Multiple blocks]:

3 Specify a point.

4 All block entities are placed in the drawing, except the attributes.

5 Depending on the value of the ATTDIA system variable:
- **You are prompted**: <Attribute Prompt 1> <Default Text 1>:
- The **Edit Attributes** dialog box appears.

6 Do one of the following:
- In the Command line, for each attribute:
  - Press Enter or right click to accept the default text.
  - Type a new text in the Command line, then press Enter and define the next attribute.
- In the **Edit Attributes** dialog box:
  - Select an attribute in the list.
  - The current value is displayed in the **Value** field.
  - Type a new value in the **Value** field.
  - Press **OK** when all attributes are properly defined.

12.13 Extracting attribute information

The ATTEXT command copies data from attributes to a text file via a dialog box.

The -ATTEXT command copies data from attributes to a text file via the Command line.

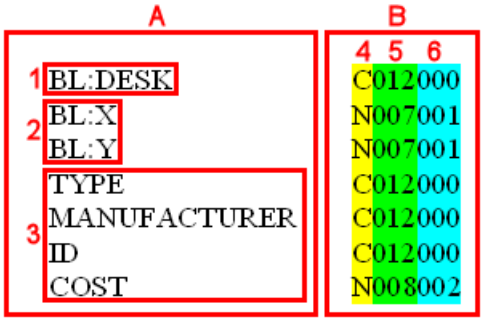
The text file can be imported in a database or a spreadsheet.

You can save the file in any of the following formats:

- **Comma Delimited Format** (CDF): Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- **Space Delimited Format** (SDF): Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- **Drawing Exchange Format** (DXF): Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BricsCAD recognizes 15 different fields, which contain elements such as the block name, the X, Y, and Z coordinates of its insertion point, the layer on which it is inserted, etc. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with **BL:**. The next nonblank character must be either a C (indicating a character string field) or an N (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:



A. Field name

B. Field format

1. Block name (must be preceded by **BL:**)
2. Coordinate fields (must be preceded by **BL:**)
3. Attribute tags
4. Indicates the field data type: **C** for character fields, **N** for numeric fields
5. Field width (number of characters)
6. Number of decimal places for numeric fields, **000** for character fields

12.13.1 To extract attribute information

1 Do one of the following:
   - Click the **Extract Attributes** tool button on the **Tools** toolbar.
   - Choose **Extract Attributes** in the **Tools** menu.
   - Launch the ATTEXT command.

   The **Extract Attributes** dialog box appears.

2 Click the **Select** button.

   The **Extract Attributes** dialog box closes temporarily.

3 Select the blocks, then press Enter or right click.

4 The **Extract Attributes** dialog box reopens.

5 Select the text file format: **DXF** (drawing exchange format), **CDF** (comma delimited format) **or SDF** (space delimited format).

6 Select the **Template file**.

7 Specify the **Output file** name.

8 Click the **Extract** button.

You are prompted: <number> records extracted.

12.14 Managing Xrefs

The EXPXREFS command opens the **External References** section of the **Drawing Explorer** dialog box.

The XREF command opens the **Attachments** panel.

The **Drawing Explorer - External References** dialog box allows you to:

- Attach a drawing as an external reference ( ).
- Detach an external reference ( ).
- Reload an external reference ( ).
- Unload an external reference ( ).
- Bind an external reference ( ).
- Insert an external reference ( ).
- Modify the attachment type: **Attach** or **Overlay**.
- Modify the **Saved Path** of an Xref.
- Modify the **Found Path** of an overlay.

12.14.1 To detach external references

1 Select the Xref(s) you want to detach.

1 Press and hold the Ctrl key to select multiple Xrefs or to unselect a selected Xref.

2 Click the **Detach Xref** tool button ( ) on the **Details** toolbar.

The selected Xref(s) are detached. You are not prompted to confirm the detachment.

12.14.2 To unload external references

Do one of the following:

- Click the **Loaded** column of a loaded Xref.
- Select the Xref(s), then either click the **Unload Xref** tool button ( ) on the **Details** toolbar or click the **Loaded** column of one of the selected Xrefs.

12.14.3 To reload external references

Do one of the following:

- Click the **Loaded** column of an unloaded Xref.
- Select the Xref(s), then either click the **Reload Xref** tool button ( ) on the **Details** toolbar or click the **Loaded** column of one of the selected Xrefs.

12.15 Attaching an external reference

The XATTACH command attaches a drawing as an external reference (Xref). The attached Xref appears as an insert in the current drawing. Like blocks, attached Xrefs can be nested. The attached drawing represents the most recently saved version.

12.15.1 To attach an external reference

1 Do one of the following:

- Click the **Attach Xref...** tool button ( ) on the **Insert** toolbar or the **Insert / References** ribbon panel.
- Choose **Attach Xref...** in the **Insert** menu.
- Click the **Attach Xref** tool button ( ) on the **Details** toolbar on the **Drawing Explorer / Xrefs** dialog box.
- Click the **Attach Dwg...** tool button ( ) on the **Attachments** panel.
- Launch the XATTCH command.

The **Select Reference File** dialog box appears.

2 Select the drawing you want to attach, then press the **Open** button.

The **Attach External Reference** dialog box appears.

3 The full path to the selected drawing is displayed in the **Path** field.

(option) Select the **Path Type**: **Full**, **Relative** or **No path**.

4 Choose the **External Reference Type**:

- **Attachment**: If the current drawing is attached as an Xref to another drawing, an attachment becomes a nested Xref.
- **Overlay**: An overlay is ignored when the drawing to which it is attached is then attached as an Xref to another drawing.

5 Choose whether to specify the **Insertion Point** on screen or in the **X**, **Y**, **Z** fields.

6 Choose whether to specify the **Scale** on screen or in the **X**, **Y**, **Z** fields.

Check the **Uniform Scale** option to specify the scale in the **X** field.

7 Choose whether to specify the **Rotation** angle on screen or in the **Angle** field.

8 Click the **OK** button to attach the Xref.

Depending on the options in steps 5, 6 and 7 you are prompted to specify the insertion point, scale and/or rotation angle.

Uwaga:

- The XDWGFADECTL (Xref database fade control) system variable sets the fading level for external references.
- The origin point of an external reference is controlled by the INSBASE (insertion base point) system variable. Use the BASE command to define the INSBASE system variable in the external reference drawing.

- The **Unit** field under **Block Unit** is set by the INSUNIT (Insertion Units) system variable in the external reference. The **Factor** field expresses the relation between the value of the INSUNIT system variable in the Xref and the parent drawing.

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 Editing blocks and external references

The REFEDIT command (short for "reference editor") edits block references and externally-referenced drawings (Xrefs) through a dialog box. What REFEDIT does is referred to as **in-place reference editing**. The command either works on a portion of the reference or on the entire drawing or all entities in the block.

The -REFEDIT command edits block references and externally-referenced drawings through the Command line.

The REFSET command adds and removes entities from the reference being edited.

The REFCLOSE command closes the reference editor.

The BEDIT command edits the entities that make up blocks in a temporary drawing session called the **Block Editor**, which uses the local coordinate system of the block.

The -BEDIT command prompts for a block name at the Command line, and then opens the **Block Editor** environment.

The XFADECTL system variable sets the fading level for the entities which are not in the working in the current refedit session. Values between 0 (no fading) and 90 are accepted.

12.17.1 To edit Xrefs

1 Do one of the following:

- Double click the Xref.
- Click the **Refedit** tool button () on the **Refedit** toolbar.
- Launch the REFEDIT command.

You are prompted: Select reference:

Click the Xref or block.

The **Reference Edit** dialog box appears.

The dialog box helps you to visually identify the reference to edit and controls how the reference is selected.

- **Path**: Location of the selected reference; if the reference is a block, no path is displayed.
- **Reference Name**: The name of the currently selected reference and any references, that are nested within the selected reference.
- **Preview**: Displays the reference as it was last saved in the drawing. Please note that the preview image is not updated when changes are saved back to the reference.

2 (option) If multiple nested references are displayed in the **Reference name** tree, select a reference to modify.

Click the expand/collapse button to show/hide nested references.

You can edit only one reference a time.

3 Choose one of the following to **Select nested entities ...** :

- **Automatically**: All the entities and nested blocks in the selected reference are included in the reference editing session.
- **Prompt**: After you close the **Reference Edit** dialog, you are prompted to select the entities and nested blocks in the reference that you want to edit. Only these entities will be temporarily extracted and be made available for modification within the context of the current drawing.

This temporary working set of entities can be edited and then saved back to update the reference being edited. To distinguish 'current drawing' from 'editing set', BricsCAD displays all of the entities belonging to the current drawing as faded. The fading level is controlled through the XFADECTL

settings variable, which has a default value of **50**.

4 Check the **Settings**:

- **Enable unique layer and symbol names**:
  - If selected, named objects in Xrefs are altered, adding a **$#$** prefix.
  - If cleared, the names of layers and other named objects remain the same as in the reference drawing.
- **Display attribute definitions for editing**: Controls whether attribute definitions in blocks are extracted and displayed during reference editing.
- **Lock entities not in working set**: Entities in the reference being edited that are not included in the temporary working set cannot be manipulated, nor can entities in the host drawing or other Xrefs. This behavior is similar to objects on locked layers.

5 Click the **OK** button to start editing the working set.

The **Reference Edit** dialog closes.

6 (option) Click the **Add to Refedit** tool button ( ) to add entities to the working set.

The selected entities will be removed from the host drawing and added to the reference being edited when the working set is saved back.

7 (option) Click the **Remove from Refedit** tool button ( ) to remove entities from the working set.

The selected entities will be removed from the reference being edited when the working set is saved back. These entities are added to the host drawing.

8 Do one of the following:

- Click the **Refclose and Save** button ( ) to save the changes back to the Xref source drawing or the block definition in the current drawing.
- Click the **Refclose and discard** button ( ) to stop the refedit session without saving. The source drawing or block definition is left unchanged.

12.18 Working with groups

The GROUP command creates and modifies named groups of entities, in a dialog box.

The -GROUP command creates and modifies named groups of entities, at the command line.

Working with groups is a technique to manipulate a set of several objects as though it were a single object. A group can also be considered as a 'Named Selection'. You can pick a single object and the entire set is selected as though it were a block, or you can edit individual objects without affecting the rest of the group and without breaking up the group. You can use nested groups if necessary and a single entity can be a member of different groups.

Uwaga: Make sure bitcode 1 of the PICKSTYLE system variable is set (value = 1 or 3) to enable the selection of groups.

12.18.1 To create a group

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears:

2 Do one of the following:

- Type a name in the **Name** field.
- Check the **Unnamed** option to create an unnamed group.

3 (option) Type a description in the **Description** field.

4 (option) Check / uncheck the **Selectable** option.

5 Click the **Select entities and create group** button.

The **Entity Grouping** dialog box temporarily closes to let you select entities.

6 Select the entities you want to include in the new group.

7 Right click to conclude the selection of entities.

8 The **Entity Grouping** dialog box reopens.

The newly created group is added.

9 Click the **OK** button to close the **Entity Grouping** dialog box.

12.18.2 To modify a group

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Select the group you want to modify.

3 (option) Double click in the **Name** field to edit the group name.

4 (option) Double click in the **Description** field to edit the group description.

5 (option) Click in the **Selectable** column to toggle the **Selectable** option on/off.

6 (option) Click the **Add entities to group** button to add new entities to the group.

The **Entity Grouping** dialog box temporarily closes to let you select the new group entities.

7 (option) Click the **Remove entities from group** button to remove entities from the group.

The **Entity Grouping** dialog box temporarily closes to let you select the entities.

8 Click the **OK** button to close the **Entity Grouping** dialog box.

12.18.3 To ungroup entities

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Select the group or groups you want to delete.

3 Click the **Un-group selected groups** button.

The selected groups are removed.

Uwaga: When you ungroup entities, the group is deleted, but the entities remain in the drawing.

12.18.4 To change the order of entities

1 Do one of the following:

- Click the **Group** tool button ( ) on the **Tools** toolbar or the **Home / Groups** ribbon panel.
- Choose **Group** in the **Tools** menu.
- Launch the GROUP command.

The **Entity Grouping** dialog box appears.

2 Click the **Re-Order entities** button.

The **Order Group** dialog box appears:

3 (option) Click the **Highlight** button to see the current position of each entity.

The **Object Grouping** dialog box opens.

4 (option) Click the **Reverse Order** button to reverse the order of all entities in the group.

5 (option) To change the order of a single entity do the following:

- In the **Remove from position** field, type the current position of the entity you want to move.
- In the **Enter new position number for the object** field, type the new position.

6 (option) Type the number of objects you want to reorder in the **Number of objects** field.

7 Click the **Re-Order** button.

A message box confirms that the group has been re-ordered.

12.19 Working with underlays

PDF files can be attached as an underlay to a drawing file. Much like raster image files and external references (Xrefs), a PDF underlay is not part of the drawing, but is linked to it. The path to the underlay is saved in the parent drawing. You can edit the path to make sure the correct PDF is found. By default the folder of the parent drawing is searched first. If the PDF underlay is not found there, the saved path folder is searched. If the PDF underlay is not found there either, BricsCAD reports a 'Missing or invalid reference' in the drawing.

The PDFADJUST command allows to adjust the fade, contrast and monochrome settings of a PDF underlay in the Command line.

The PDFATTACH and -PDFATTACH commands insert a PDF file as an underlay into the current drawing.

The PDFLAYERS command allows you to control the display of layers in a PDF underlay.

The PDFCLIP command crops the display of a PDF. You can choose between a polygonal or a rectangular boundary. Each instance of the same PDF underlay file can have a different boundary.

The PDFOSNAP system variable allows to snap to the geometry in the PDF underlay if the system variable is **On**.

The PDFNOTIFY system variable controls whether warnings for missing or modified PDF underlays are issued.

The XNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and PDF attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is **On**.

If the value of the FRAME system variable = 3, the PDFFRAME system variable controls the visibility of PDF frames. The options are:

- 0: Hide PDF frames.
- 1: Display and plot PDF frames.
- 2: Display but do not plot PDF frames.

12.19.1 Controlling the PDF underlay caching

A multi-resolution persistent image cache is used to display attached PDF underlays, enabling (very) fast zoom and pan operations. The highest cached resolution is 5000 x 5000 pixels. Still, when zooming in very close, the display of the PDF underlay will become pixelated. So a hybrid modus can be used which switches to the real-time generation of a crisp PDF underlay display when zooming in very closely. The initial generation of the image cache may take a few seconds, but from then on processing gets (very) fast, and remains just as swift in subsequent sessions.

The PDFCACHE system variable controls how PDF caching is applied.

- 0 = don't use PDF cache, always use real-time generation of the PDF underlay image. (this mode was the default in previous BricsCAD versions).
- 1 = use PDF cache, switch to real-time generation when zooming in very close.
- 2 = always use PDF cache (default).

12.19.2 Attaching a PDF underlay

1 Do one of the following:

- Click the **Attach PDF** tool button ( ) on the **Insert** toolbar or the **Insert / PDFs** ribbon panel.
- Click the **New** tool button ( ) on the **Details** toolbar of the **Drawing Explorer - PDF Underlays** dialog box.
- Type **PDFattach** in the Command line, then press Enter.

The **Select PDF Underlay File** dialog opens.

2 Select the file, then double click the file or click the **Open** button on the **Select PDF Underlay File** dialog.

The **Attach PDF Underlay** dialog box appears.

3 In case of a multi-page PDF, click the down arrow in the **Name** field, then select the page of the PDF you want to load.

4 Click the **Path** option button, then select either:
- **Full path**: The full path will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.
- **Relative path**: The relative path, with respect to the folder of the parent drawing, will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.
- **No path**: Only the folder of the parent drawing is searched when the drawing is loaded. If the overlay file is not found there, "Missing or invalid reference" is displayed at the insertion point of the PDF overlay.

5 Choose whether you want to specify the **Insertion Point**, **Scale** and the **Rotation** angle on-screen or not.

6 Click the **OK** button.

The **Attach PDF Underlay** dialog closes.

7 Depending on the insertion options chosen in the previous step, **you are prompted** to specify the **Insertion Point**, **Scale** and/or **Rotation** angle.

12.19.3 Inserting a PDF underlay

If you want to insert a second instance of a PDF that is already attached or another page of such underlay do the following:

1 Launch the EXPPDFS command.

1 The **Drawing Explorer - PDF Underlays** dialog box appears.

2 Select the PDF underlay.

3 Do one of the following:
- Right click and choose **Insert** in the context menu.
- Choose **insert** in the **Edit** menu on the **Drawing Explorer - PDF Underlays** dialog.

The **Drawing Explorer - PDF Underlays** dialog closes.

4 Continue with steps 3 through 7 of the previous procedure.

12.19.4 Controlling the display of a PDF underlay

1 Click the frame of the PDF underlay.

1 The frame of the selected PDF highlights.

The properties of the underlay are displayed in the **Properties** panel.

2 Under **Misc**, click **Show underlay** in the **Properties** bar, then select **Yes** or **No**.

12.19.5 Setting the layer display in a PDF underlay

1 Launch the PDFLAYERS command.

1 **You are prompted**: Select PDF underlay.

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

The **Underlay Layers** dialog box appears.

4 The icon in the **On** column indicates the current state of the layers; **On** ( ) or **Off** ( ).

5 Click the icon in the **On** column to toggle the display of a layer.

6 (option) Click the **Search for layer** field, then type a layer name to search for a layer.

Use wildcard characters (? or *) to limit the number of layers in the list.

7 (option) Click the **Reference name** button to select another PDF underlay.

8 Click the **OK** button to stop.

12.19.6 Clipping a PDF underlay

1 Launch the PDFCLIP command.

1 **You are prompted**: Select PDF underlay:

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:

4 Press Enter to accept the **New** default command option.

You are prompted: Enter PDF clipping type [Polygonal/Rectangular] <Rectangular>:

5 Do one of the following:

- Press Enter to accept the **Rectangular** default command option to define a rectangular clipping boundary.
- Type P, then press Enter to define a polygonal clipping boundary.

6 Define the clipping boundary.

7 The defining points must lie inside the frame of the PDF underlay. If you click outside the PDF underlay the point is placed on the PDF underlay frame.

12.19.7 Toggling the display of the clipped part of a PDF underlay

1 Click the frame of the PDF underlay.

1 The frame of the selected PDF highlights.

The properties of the underlay are displayed in the **Properties** panel.

2 Under **Misc**, click **Show clipped** in the **Properties** bar, then select **Yes** or **No**.

12.19.8 Deleting a clipping boundary

1 Launch the PDFCLIP command.

1 **You are prompted**: Select PDF underlay.

2 Click the frame of the PDF underlay.

3 The frame of the selected PDF highlights.

You are prompted: Enter PDF clipping option [ON/OFF/Delete/New] <New>:

4 Type D in the Command line, then press Enter.

5 The clipping boundary is deleted and the complete PDF underlay appears.

12.19.9 Adjusting the display settings of a PDF underlay

The PDFADJUST command allows to adjust the **Fade**, **Contrast** and **Monochrome** settings of a PDF underlay in the Command line.

Select the PDF underlay, then adjust the display settings under **Underlay Adjust** in the **Properties** panel.

12.20 Components and component inserts

A component is a named group of entities, that can be inserted in a drawing with the BmInsert command. Components can be parametric: e.g. windows and doors, in BricsCAD BIM and standard mechanical components in BricsCAD Mechanical.

Any .dwg file can be used as a component. The same component can be inserted once or several times into one or more other components. When the component is modified, all inserts of the component will reflect these changes automatically. Components cannot self-reference.

A component insert is a named entity. The default name of a component insert is composed of the name of the component and the serial number of the insert. Internally a component insert is represented either as a reference to an anonymous block or as an external reference.

Properties of inserted components, including their parameters, can be edited in the **Mechanical Browser** panel and **Properties** panel.

12.20.1 External and local components

Components can be either local or external. A local component is stored in the .dwg file with the assembly as an anonymous block. An external component is defined in a separate .dwg file and represented by an Xref or as a local copy stored as an anonymous block. When the corresponding .dwg file is modified all inserts of external components can be updated using the BMUPDATE command or the **Mechanical Browser** context menu. If you use external components in your assembly, always remember to transfer all related files when you want to share your model.

Use the ETRANSMIT command to create a package of a drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables and font map files.

You can always convert a local component to an external one and vice versa.

To edit a local component, it first needs to be converted to an external component using the BMEXTERNALIZE command. Alternatively, you can use the BMOPENCOPY command to open a copy of a component, edit and save it. Then use the BMREPLACE command to replace the original with the copy.

You can choose the default insertion type for your component: **Local** or **External**. To change the type, select the root node in the **Mechanical Browser** and select **Local** or **External** from **Insert as properties**.

Depending on the value of BMAUTOUPDATE, inserts of external components are updated automatically on opening the assembly document or manually with the BMUPDATE command. You can change this value in the **Settings** dialog box. The options are:

- Update only when the BMUPDATE command is used.
- Update automatically upon opening the file.

12.20.2 Converting components

The BMEXTERNALIZE command converts local components to external components.

The BMLOCALIZE command converts external components to local components.

Do one of the following:

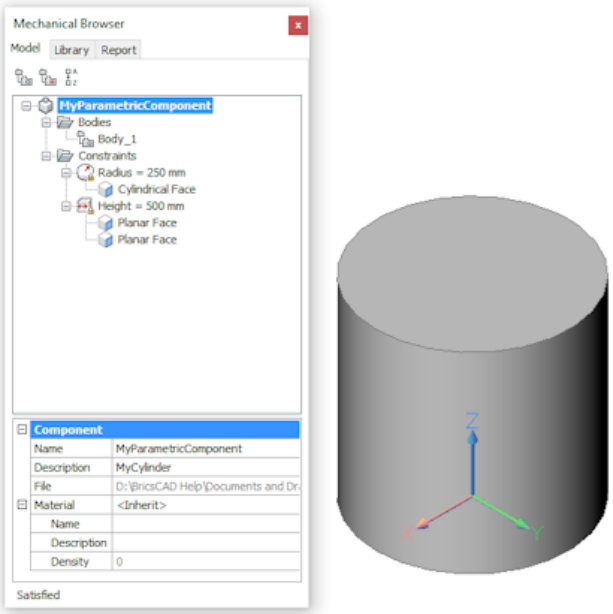
- Click **Switch Component to External** () or **Switch Component to Local** on the **Assembly** ribbon tab. The tools are also available from the **Assembly** toolbar and the **Assembly** menu.
- Right-click on the component insert in the **Mechanical Browser** and select **Switch to External** or **Switch to Local** in the context menu.
- Hover your mouse cursor over a component insert in a drawing window and select **Switch to External** () or **Switch to Local** in the **Assembly Modeling** command group in the Quad.

When you convert an external component to a local one, the .dwg file with the component definition is not deleted, but future changes will not affect your assembly.

You can always distinguish between inserts of local components () and inserts of external components () by their icons in the **Mechanical Browser**.

12.20.3 Parametric Components

You can make parametric changes of inserted components at the assembly level. A component is considered parametric if it contains at least one 2D or 3D dimensional constraint or an associative array with an expression.



Uwaga:

- BricsCAD automatically maintains associativity between the definition of a parametric component and its inserts. So, you can open the parametric component for editing, modify its geometry and save it. To see the result, execute the BmUpdate command to update the mechanical structure of your assembly.
- If you need to go back to the default value of a component's parameter, just edit the **Expression** field and delete its content.

You can control if a particular component parameter will be available for editing in a drawing that contains the inserts of these components (at the assembly level). To do that, select a parameter in the **Mechanical Browser** and change the value of the **Exposed** field in the properties grid of the parameter.

The options are:

- **Off**: Hides the parameters at the assembly.
- **On**: Makes the parameter visible at the assembly level.
- **Auto**: Hides all parameters with values controlled by expressions and exposes all other parameters.

You can associate units with a parameter. If you assign a value, directly or via an expression, to a component parameter at the assembly level, it automatically converts from drawing units of the assembly drawing to the units of the parameter. Values assigned to unitless parameters are not converted.

By default, all parameters are unitless except parameters of dimensional constraints and parameters, which are used as expressions for dimensional constraints. BricsCAD automatically assigns drawing units to such parameters.

12.20.4 To edit the parameters of a component insert

When a parametric component is inserted in a model, you can edit the parameters either in the **Mechanical Browser** panel or in the **Properties** panel.

The component insert immediately updates according to the new value of a parameter. Please note that this does not modify the definition of the parametric component, which is a separate drawing file. As a result, multiple inserts of the same parametric component can exist in the same model, with different values for each of their parameters:

To edit a parameter in the Mechanical Browser

1 Expand the **Parameters** node of the component.

2 Select the parameter.

3 In the **Parameter** section, type a new value in the **Expression** field.

To edit a parameter in the Properties bar

1 Select the component in the drawing.

2 Expand the **Parameters** settings group.

3 Type a new value for one of the available parameters.

Uwaga:

- In the **Properties** panel, the current value is displayed. If this value is controlled through an expression, the expression will be overwritten by the newly entered value.
- If multiple parametric components are selected, the shared parameters can be edited simultaneously. The settings fields of parameters that currently have different values reads: *varies*.

12.20.5 To replace components

The BMREPLACE command replaces component inserts. Only top-level inserts can be replaced.

Uwaga: Parameter values that are shared between the replacement component and the replaced component are copied automatically.

12.20.6 Components, blocks or external references?

Components are based on blocks and external references (Xrefs) but the behavior of components is different. An external component is linked to a file, much like an Xref, but an external component can be represented by either an Xref (BMINSERT acts like XATTACH) or by a local anonymous block (BMINSERT acts like INSERT).

Components can be either parts or assemblies:

- Part: composed of entities only; no inserts of subcomponents.
- Assembly: composed of entities and subcomponent inserts. Even when the subcomponents are removed, the assembly status is kept.

When an external component is a part and its parameters are not modified, the component is inserted as an Xref. This is listed in the **External References** section of the **Drawing Explorer** dialog. Xref layers have a 'file_name' prefix.

When an external component is an assembly, it is inserted as a local anonymous block. All of its layers are merged with the layers of the main document and no prefix is added to the layer name.

12.20.7 Converting blocks and external references

The BMMECH command converts the current drawing into a mechanical component. If the drawing already is a mechanical component BMMECH does nothing.

Block references are converted to inserts of local components and external references are converted to inserts of external components.

The BMUNMECH command converts the current mechanical component into a plain drawing. Mechanical components are converted back to blocks and external references. The command applies to drawings that are a mechanical component only.

12.20.8 Component-based features

A feature is a physical constituent of a part with engineering significance. Examples of features are holes, ribs, slots, pockets, forms. If applied to a 3D solid, a feature removes and/or adds geometry. To create your own features, use the BricsCAD component extension technology.

An extended component has BC_SUBTRACT and/or BC_UNITE layers. When you use the BMINSERT command to insert such a component on the face of a 3D solid, the target solid is modified as follows:

- All 3D solids on the BC_SUBTRACT layer of the component are subtracted from the target 3D solid.
- All 3D solids on the BC_UNITE layer of the component are united with the target 3D solid.

All faces created after Boolean operations form a component-based feature. They are updated when you move the component insert or when you modify any parameters. When the insert is deleted, the target solid is healed. Adjust the target 3D solids with the BMLINK command.

Some examples of parametric component-based features are included with BricsCAD. You can use them as samples to create your own features.

12.20.9 Assign physical materials to components

You can assign a physical material to a mechanical component. If a mechanical component does not have an assigned material, it will inherit the material from its parental component. If the parental component does not have an assigned material, the material of the nearest component, will be used. As a result, the same material can be assigned to all components that have a common parent.

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Browse** button (...).

3 In the **Physical Materials** dialog box, select a material from either the **In Project** or the **In Library** list.

Materials that are chosen from the **In Library** are copied to the **In Project** database automatically.

4 Click the **OK** button to assign the selected material to the component.

To remove the material from the component:

1 Select the root component in the **Mechanical Browser**.

2 Select the **Material** property and press the **Clear** button (X).

Inserting a component with a physical material:

When you insert a mechanical component in a model, all its materials are copied to the material library of the model. If a material with the same name already exists in the target model, this material will be used instead. As a result, the properties of the material in the target model are used. This replaces the properties of the material from the inserted component.

Uwaga: Materials that are copied to the document material library will not be erased in **Undo** operations.

13. Zarządzanie obrazami

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.

Uwaga: 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. Adnotacja

14.1 Dimensions

14.1.1 Dimensioning tools overview

Dimensioning tools can be found in:

- The **Dimensions** toolbar.
- The **Dimensions** menu.
- The **Annotate / Dimensions** ribbon panel.
- The **Draw** tab of the **No Selection** Quad.

Icon	Tool name	Command
	Linear	DIMLINEAR
	Aligned	DIMALIGNED
	Arc	DIMARC
	Radius	DIMRADIUS
	Diameter	DIMDIAMETER
		DIMDISASSOCIATE
		DIMREASSOCIATE
	Angular	DIMANGULAR
	Baseline	DIMBASELINE

Icon	Tool name	Command
	Continue	DIMCONTINUE
	Ordinate	DIMORDINATE
	Rotated	DIMLINEAR
	Make Oblique	DIMEDIT + O
	Multileader	MLEADER
	Edit Multileader	MLEADEREDIT
	Multileader add leaders	AIMLEADEREDITADD
	Multileader remove leaders	AIMLEADEREDITREMOVE
	Tolerance	TOLERANCE
	Center Lines	DIMCENTER
	Edit Dimension Text	DIMEDIT + E
	Rotate Dimension Text	DIMEDIT + R
	Reposition Dimension Text	DIMTEDIT

Icon	Tool name	Command
	Restore Text Position	DIMEDIT
	Apply Style	-DIMSTYLE + A
	Save Style	-DIMSTYLE + S
	Restore Style	-DIMSTYLE
	Dimension Variable Status	-DIMSTYLE + ST
	Leader	DIMLEADER
	Quick Leader	QLEADER

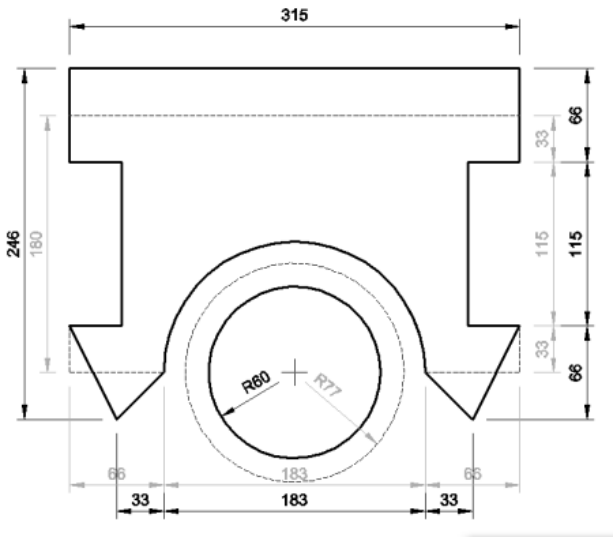
14.1.2 Wymiary asocjacyjne

Wymiary zespolone są aktualizowane automatycznie, gdy wymiarowany element jest przesuwany lub modyfikowany.

BricsCAD tworzy wymiary skojarzone, jeśli wartość zmiennej systemowej DIMASSOC wynosi 2 (domyślnie).

Skojarzona właściwość wymiarów dotyczy również wymiarów, które są umieszczane w obszarze papieru i są skojarzone z elementami obszaru modelu.

W razie potrzeby możesz użyć polecenia REGENWYM, aby zaktualizować wszystkie wymiary zespolone.



Do tworzenia wymiarów zespolonych można użyć następujących trybów lokalizacji: **punkt końcowy** , **punkt środkowy** , **środek** , **prostopadły** , **kwadrant** , **wprowadzenie** , **punkt** , **skrzyżowanie** oraz **pozorne przecięcie** .

Polecenie ODŁĄCZWYM Usuwa zespolenie z wybranych elementów wymiaru.

Polecenie ZESPOŁWYMIAR ponownie kojarzy lub kojarzy wymiary z elementami lub punktami na elementach.

Zmienna systemowa GENERATEASSOCVIEWS umożliwia tworzenie wymiarów zespolonych dla wygenerowanych widoków za pomocą poleceń PODSTAWA WIDOKU i PRZEKRÓJWIDOK oraz wygenerowanych rysunków za pomocą polecenia BIMSECTIONUPDATE.

14.1.3 Ustawienia wymiarów

Ustawienia wymiarowania są zapisywane w stylu wymiarowania. Elementy wymiaru są tworzone przy użyciu ustawień w bieżącym stylu wymiarowania.

Polecenie WYMSTYL tworzy i modyfikuje style wymiarowania poprzez **Eksplorator rysunków** .

Polecenie -WYMSTYL tworzy i modyfikuje style wymiarowania w wierszu polecenia.

Styl wymiarowania składa się z sześciu grup ustawień

- **Linie i Strzałki**
- **Tekst**
- **DOPasuj**
- **Jednostki główne**
- **Alternatywne Jednostki**
- **Tolerancje**

14.1.3.1 Aby wyświetlić aktualny stan zmiennych wymiaru

1 Wykonaj jedną z następujących czynności:

- Kliknij **Stan zmiennej wymiaru** przycisk narzędzia () na **Wymiary** pasek narzędzi.
- Wybierać **Stan zmiennej wymiaru** w **Wymiary** menu.

Na pasku poleceń zostanie wyświetlona lista wszystkich zmiennych wymiaru.

2 Naciśnij **klawisz funkcyjny F2** , aby otworzyć okno **Historia komunikatów BricsCAD**.

3 Naciśnij Enter, aby kontynuować aukcję.

4 Naciśnij **klawisz funkcyjny F2** , aby zamknąć okno **Historia komunikatów 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.

Uwaga: 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.

Uwaga:

- 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.

Uwaga: 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.

Uwaga:

- 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.

Uwaga:

- 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.

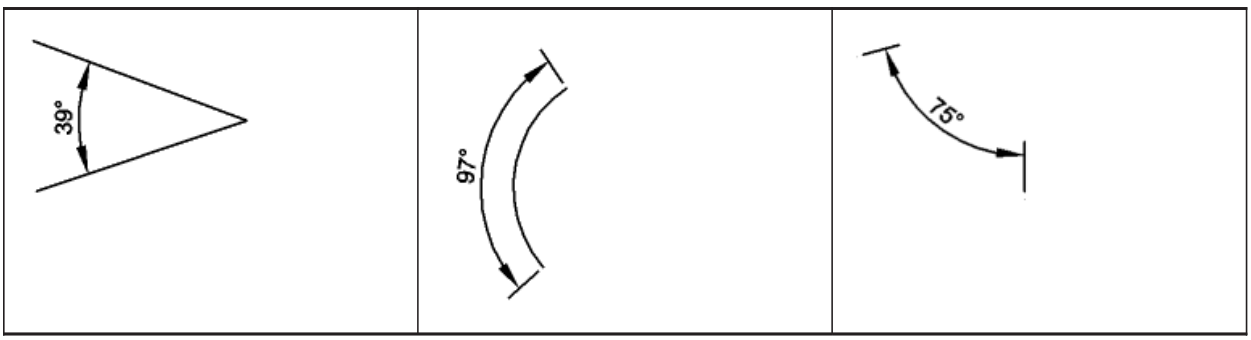
14.1.7 Tworzenie wymiarów kątowych

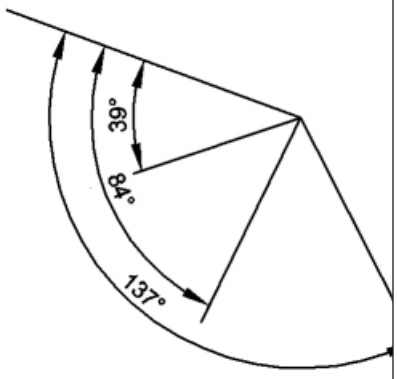
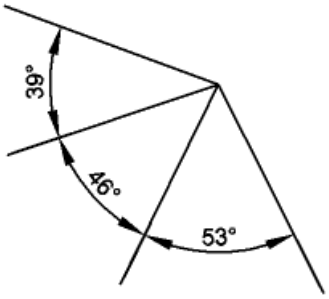
Polecenie WYMIAROWY tworzy wymiar kątowy.

Wymiary kątowe opisują kąt mierzony między dwiema liniami, dwoma liniowymi segmentami polilinii lub kąt łuku kołowego. Kąt można również zwymiarować, wybierając wierzchołek kąta i dwa punkty końcowe.

Zaczynając od istniejącego wymiaru kątowego, można dodać wymiar bazowy lub wymiar ciągły.

- Kątowy wymiar bazowy wstawia dodatkowy wymiar ze wspólnego początku pierwszej linii pomocniczej.
- Ciągły wymiar kątowy kontynuuje wymiar kątowy z drugiej linii pomocniczej poprzedniego wymiaru kątowego.



wymiar kątowy mierzony między dwiema liniami	wymiar kątowy łuku	wymiar kątowy zdefiniowany przez wierzchołek i dwa punkty końcowe
		
kątowe wymiary bazowe	wymiary ciągłe kątowe	

14.1.7.1 Aby zwymiarować kąt między dwiema liniami

1 Uruchom polecenie WYMIAROWY.

1 **Pasek poleceń wyświetli:** Wybierz linię, łuk lub okrąg [opcje wyboru (?)] <Naciśnij klawisz ENTER, aby określić kąt>:

2 Wybierz linię lub liniowy segment polilinii.

3 **Pasek poleceń wyświetli:** Inny wiersz dla wymiaru kątowego [opcje wyboru (?)]:

4 Wybierz drugą linię lub liniowy segment polilinii.

5 Wymiar kątowy jest wyświetlany dynamicznie.

Pasek poleceń wyświetli : Położenie łuku wymiarowego [Kąt/Tekst]:

6 Określ punkt, aby umieścić wymiar.

7 Przesuń kursor, aby zwymiarować kąt wewnętrzny między liniami lub jego uzupełnienie.

14.1.7.2 Aby zwymiarować kąt zawarty w łuku

1 Uruchom polecenie WYMIAROWY.

1 **Pasek poleceń wyświetli:** Wybierz linię, łuk lub okrąg [opcje wyboru (?)] <Naciśnij klawisz ENTER, aby określić kąt>:

2 Wybierz łuk.

3 Wymiar kątowy jest wyświetlany dynamicznie.

Pasek poleceń wyświetli : Położenie łuku wymiarowego [Kąt/Tekst]:

4 Określ punkt, aby umieścić wymiar.

14.1.7.3 Aby utworzyć wymiar kątowy zdefiniowany przez wierzchołek i dwa punkty końcowe

1 Uruchom polecenie WYMIAROWY.

1 **Pasek poleceń wyświetli:** Wybierz linię, łuk lub okrąg [opcje wyboru (?)] <Naciśnij klawisz ENTER, aby określić kąt>:

2 Naciśnij Enter.

3 **Pasek poleceń wyświetli**: Wierzchołek kąta:

4 Określ punkt.

5 **Pasek poleceń wyświetli** : Pierwsza strona kąta:

6 Określ punkt.

7 **Zostanie wyświetlony monit** : Druga strona kąta:

8 Określ punkt.

9 Wymiar kątowy jest wyświetlany dynamicznie.

Pasek poleceń wyświetli : Położenie łuku wymiarowego [Kąt/Tekst]:

10 Określ punkt, aby umieścić wymiar.

11 Przesuń kursor, aby zwymiarować kąt wewnętrzny lub zewnętrzny.

14.1.8 Tworzenie wymiarów średnicowych i promieniowych

Polecenia DIMCENTER i DIMDIAMETER tworzą wymiary średnicowe i promieniowe w celu opisania promieni i średnic okręgów i łuków kołowych.

14.1.8.1 Aby utworzyć wymiar średnicowy

1 Uruchom polecenie WYMŚREDNICA.

1 **Pasek poleceń wyświetli** : Wybierz łuk lub okrąg do zwymiarowania [opcje wyboru (?)]:

2 Wybierz łuk lub okrąg.

3 Wymiar średnicowy jest wyświetlany dynamicznie.

4 Określ punkt, aby umieścić element wymiaru.

14.1.8.2 Aby utworzyć wymiar promieniowy

1 Uruchom polecenie WYMPROMIEŃ.

1 **Pasek poleceń wyświetli** : Wybierz łuk lub okrąg do zwymiarowania [opcje wyboru (?)]:

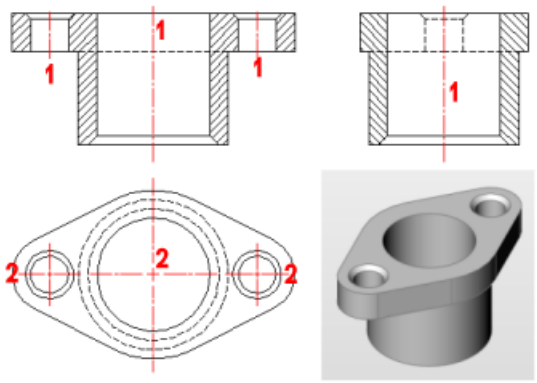
2 Wybierz łuk lub okrąg.

3 Wymiar promieniowy jest wyświetlany dynamicznie.

4 Określ punkt, aby umieścić element wymiaru.

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.

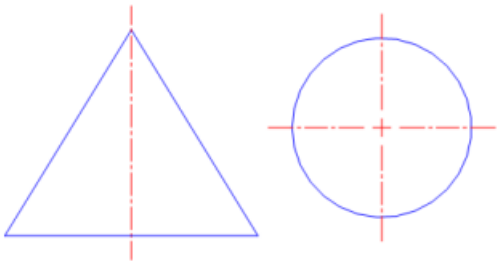
The CENTERDISASSOCIATE command breaks the association of centerline and a center mark entities.

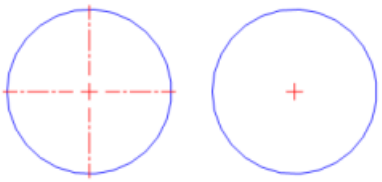
The CENTERRESET command resets centerline and center mark entities.

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
CENTERMARK EXE	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 Tworzenie wymiarów współrzędnościowych

Wymiar współrzędnościowy opisuje prostopadłą odległość od początku lub punktu bazowego (początek bieżącego układu współrzędnych (GUW lub LUW). Wymiary współrzędnościowe składają się ze współrzędnej x lub y oraz linii odniesienia. Wymiar współrzędności x mierzy odległości wzdłuż osi x, podczas gdy wymiar współrzędności y mierzy odległości wzdłuż osi y.

Podczas wybierania punktów współrzędnych program automatycznie wykrywa, czy punkt jest oparty na współrzędnej x czy y, w zależności od kierunku przeciągania drugiego punktu. Możesz również określić, czy rzędna reprezentuje rzędną x czy y. Tekst wymiarowy współrzędności jest zawsze wyrównywany do linii odniesienia rzędnych, niezależnie od orientacji tekstu określonej przez bieżący styl wymiarowania.

14.1.10.1 Aby utworzyć wymiar współrzędnościowy

1 Ustaw początek układu współrzędnych w punkcie odniesienia dla wymiarów współrzędnościowych.

2 Uruchom polecenie WYMORDYN.

3 **Pasek poleceń wyświetli** : Wybierz punkt dla wymiaru współrzędnościowego:

4 Określ punkt.

5 Wymiar współrzędnościowy jest wyświetlany dynamicznie.

Zostanie wyświetlony monit: punkt końcowy Leader [Xdatum / Ydatum / Tekst / Kąt]:

6 Kliknij, aby umieścić wymiar współrzędnościowy.

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.

Uwaga:

- 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.

Uwaga:

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.

Uwaga: To restore non-rotated extension lines, execute the above procedure and type 0 (zero) at the 'Enter obliquing angle' prompt.

14.1.12 Tworzenie liderów i adnotacji

Linia odniesienia składa się z grotu strzałki, linii, polilinii lub splajnu oraz opisu.

Ogólna procedura umieszczania adnotacji to:

- Określ położenie grotu strzałki.
- Utwórz segmenty linii odniesienia.
- Wpisz tekst adnotacji.

14.1.12.1 Aby utworzyć linię odniesienia i adnotację

1 Wykonaj jedną z następujących czynności:

- Kliknij **Lider** przycisk narzędzia ( ) na **Wymiary** pasek narzędzi lub panel wstążki.
- Wybierać **Lider** w **Wymiary** menu.
- Uruchom polecenie LEADER lub DIMLEADER.

Zostanie wyświetlony monit : Początek lidera

2 Określ położenie grotu strzałki.

3 **Zostaniesz poproszony** : Następny punkt:

4 Określ punkt końcowy pierwszego segmentu linii odniesienia.

5 **Pasek poleceń wyświetli** : Aby wskazać [Format/Cofnij/Adnotacja] <Adnotacja>:

6 Dodaj więcej segmentów do linii odniesienia, a następnie naciśnij klawisz Enter lub kliknij prawym przyciskiem myszy, aby zatrzymać dodawanie segmentów.

7 **Zostanie wyświetlony monit:** pierwszy wiersz tekstu adnotacji / <Opcje>:

8 Wpisz pierwszy wiersz tekstu adnotacji, a następnie naciśnij klawisz Enter.

9 **Pasek poleceń wyświetli** : Następny wiersz tekstu adnotacji:

10 Wykonaj jedną z następujących czynności:

- Naciśnij klawisz Enter lub kliknij prawym przyciskiem myszy, aby utworzyć linię odniesienia i adnotację.
- Kontynuuj dodawanie linii do tekstu adnotacji, a następnie naciśnij klawisz Enter lub kliknij prawym przyciskiem myszy, aby utworzyć linię odniesienia i adnotację.

14.1.12.2 Aby utworzyć tylko lidera

1 Wykonaj jedną z następujących czynności:

- Kliknij **Lider** przycisk narzędzia ( ) na **Wymiary** pasek narzędzi lub panel wstążki.

- Wybierać **Lider** w **Wymiary** menu.
- Uruchom polecenie LEADER lub DIMLEADER.

Zostanie wyświetlony monit : Początek lidera

2 Określ położenie grotu strzałki.

3 **Zostaniesz poproszony** : Następny punkt:

4 Określ punkt końcowy pierwszego segmentu linii odniesienia.

5 **Pasek poleceń wyświetli** : Aby wskazać [Format/Cofnij/Adnotacja] <Adnotacja>:

6 Dodaj więcej segmentów do linii odniesienia, a następnie naciśnij klawisz Enter lub kliknij prawym przyciskiem myszy, aby zatrzymać dodawanie segmentów.

7 **Zostanie wyświetlony monit:** pierwszy wiersz tekstu adnotacji / <Opcje>:

8 Naciśnij Enter.

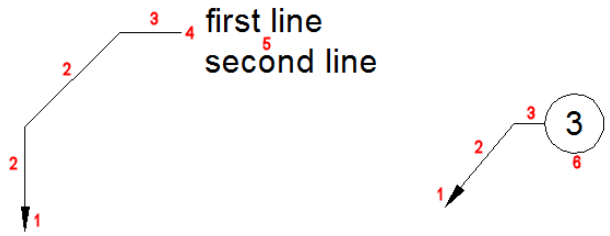
9 **Pasek** poleceń wyświetli: Opcje tekstu wymiaru [Blok/Kopiuj/Brak/Tolerancja/Wtekst] <Wtekst>:

10 Wybierz **Nic** opcja.

14.1.13 Creating multileaders

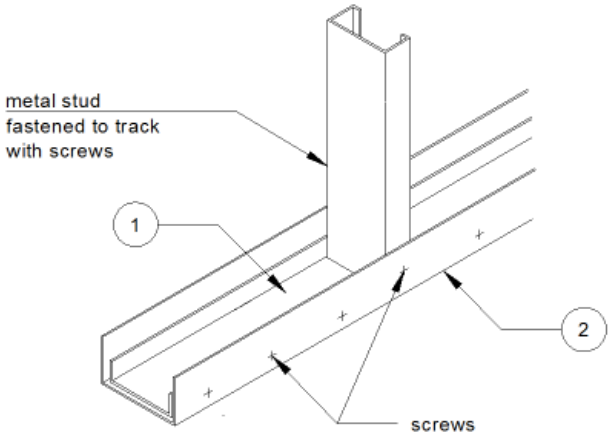
A multileader consists of:

- 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 Adding geometric tolerances

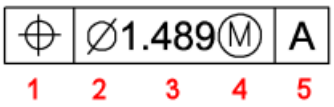
Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The geometric tolerance is, in essence, the tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability.

BricsCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface.

The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size.

The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.



1 Geometric characteristic symbol

2 Diameter symbol

3 Tolerance value

4 Material condition symbol

5 Datum reference

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

14.1.14.1 Creating a geometrical tolerance frame

1 Do one of the following:

- Click the **Tolerance** tool button ( ) in the **Dimensions** toolbar or ribbon panel.
- Choose **Tolerance** in the **Dimensions** menu.

- Type **tolerance** in the Command line, then press Enter.

The Geometric Tolerance dialog opens.

2 Compose the geometrical tolerance frame in the **Geometric Tolerance** dialog box.

3 Click the **OK** button.

The **Geometric Tolerance** dialog box closes.

4 Specify a point to insert the geometric tolerance frame in the drawing.

14.1.14.2 Editing a geometrical tolerance frame

1 Double click the geometrical tolerance frame.

1 The **Geometric Tolerance** dialog box opens.

2 Edit the settings.

3 Click the **OK** button.

The **Geometric Tolerance** dialog box closes.

The content of the geometrical tolerance frame is updated.

14.2 Texts

14.2.1 Zmienne systemowe tekstu

Zmienne systemowe można ustawić za pomocą **Ustawienia** Okno dialogowe.

Nazwa zmiennej	Tytuł zmiennej	Opis
FONTALT	Czcionka alternatywna	Definiuje czcionkę która będzie użyta gdy używana nie zostanie odnaleziona.
UKRYJTEKST	Ukryj tekst przy ukryciu	Określa czy polecenie UKRYJ dotyczy obiektów tekstowych.
MIRRTEKST	Lustro Tekstu	Określa czy tekst jest odwracany przez polecenie LUSTRO.
WTEKST	Edytor tekstu wielowierszowego	Ustawia pierwszorzędny i drugorzędny edytor tekstu dla obiektów tekstu wielowierszowego.
WTEKST NAPRAWIONO	Pole tekstu wielowierszowego	Kontroluje czy BricsCAD przybliża, obraca i/lub przesuwa widok, aby dopasować do tekstu wielowierszowego podczas edycji.
TRYBQTEKST	Tryb szybkiego tekstu	Kontroluje sposób wyświetlania obiektów typu tekst. Jeśli ON, wyświetla tekst jako ramkę.

Nazwa zmiennej	Tytuł zmiennej	Opis
TEKST	Kąt tekstu	Zachowuje kąt obrotu ostatnio utworzonego obiektu tekstowego.
WYPEŁNIJ TEKST	Wypełnienie tekstu	Określa, czy czcionki True Type są wyświetlane jako wypełnione, czy jako obrysowane. 
TEKSTQLTY	Jakość tekstu	Definiuje gładkość czcionek True Type do drukowania i renderowania.
ROZMIAR CZCIONKI	Rozmiar tekstu	Ustawia domyślną wysokość dla nowych elementów tekstowych. TEXTSIZE nie działa, jeśli bieżący styl tekstu ma stałą wysokość.
STYLTEKSTU	Styl tekstu	Zachowuje nazwę aktualnego stylu tekstu.
TSPACEFAC	Współczynnik odstępu tekstu	Określa odległość między wierszami tekstu wielowierszowego mierzoną jako współczynnik wysokości tekstu. Akceptowane są wartości od 0.25 do 4.
TSPACETYP	Typ spacji tekstu	Określa typ odstępów między wierszami używany dla tekstu wielowierszowego. Przynajmniej: dostosowuje odstępy między wierszami na podstawie najwyższych znaków w wierszu. Dokładnie: użyto określonego odstępu między wierszami, niezależnie od wielkości poszczególnych znaków.
TSTACKALIGN	Podkreślenie tekstu	Określa położenie tekstu: dół, środek, góra.
TSTSTACKSIZE	Rozmiar tekstu	Określa procent a, w jakim wysokość ułamka tekstu piętrowego jest względem bieżącej wysokości zaznaczonego tekstu. Akceptowane są wartości od 25 do 125.

14.2.2 Working with text styles

When you add text to a drawing, it uses the current text style, which determines the font, size, angle, orientation and other properties of the new text entity.

Every drawing has a default text style, named **Standard**, which initially uses the **Arial** font. You cannot delete or rename the **Standard** style, but you can modify it.

If you change the **font** or **orientation** of an existing style, all existing text entities created using that style are automatically updated to reflect the change. Changing any other property has no effect on existing text.

You can create and use an unlimited number of additional text styles.

Annotative text styles create annotative text entities. In an annotative text style, the **Height** defines the paper space height of the text; the model space height depends on the current annotation scale as defined by the CANNOSCALE and CANNOSCALEVALUE system variables.

14.2.2.1 To open the Text Style Explorer dialog box

Do one of the following:

- In the **Tools** > **Drawing Explorer** menu, choose **Text Styles ...**
- Click the **Style** field in the **Status** bar.
- Launch the STYLE command.

14.2.2.2 To create a text style

1 Open the **Text Style Explorer** dialog box.

2 Do one of the following:

- Click the **New** tool button () in the **Details** section of the **Drawing Explorer** dialog.
- Select an existing style, then right click and choose **New** in the context menu.

A **NewStyle** text style is created, inheriting the properties of the currently selected style.

3 Type a name in the **Text Style Name** field, replacing the **NewStyle** default name.

4 Define the text style properties.

5 Close the **Text Style Explorer** dialog.

14.2.2.3 To edit a text style

1 Open the **Text Style Explorer** dialog box.

2 Select the text style.

3 Define the text style properties.

4 Close the **Text Style Explorer** dialog.

14.2.2.4 To set a text style current

Do one of the following:

- Right click the **Style** field in the **Status** bar, then click the text style.
- Open the **Text Style Explorer** dialog box.

  The current text style is indicated in the **Current** column.

  Do one of the following:

  - Click the **Current** field of the text style.
  - Right click the text style, then choose **Set Current** in the context menu.

14.2.2.5 To delete a text style

1 Open the **Text Style Explorer** dialog box.

2 Select the text style.

3 Do one of the following:

- Click the **Delete** tool button ( ).
- Select the text style, then choose **Delete** in the context menu.

If the text style is not used in the drawing or in a dimension style, the text style is deleted, else an alert box appears.

4 Do one of the following:

- Click the **Delete** button to delete the text style and all entities using it.
- Click the **Change...** button to delete the text style and assign a different text style to the entities, which currently use the selected style.
- Click the **Cancel** button to cancel the delete procedure.

Uwaga: It is not possible to delete the current text style.

14.2.3 Creating text

The TEXT command places single lines of text in the drawing; optionally evaluates LISP expressions.

The MTEXT command places formatted, paragraph text in a bounding box that limits the extent of the text.

14.2.3.1 To create single line texts

1 Do one of the following:

- Click the **Text** tool button ( ) on the **Draw** toolbar or the **Annotate / Text** ribbon panel.
- Choose **Text** in the **Draw** menu.
- Launch the TEXT command.

2 **You are prompted**: Text start point or [use defined Style/Align on line/Fit between points/Center horizontally/Middle (hor/vert)/Right justify/Justification options...]:

3 Specify a point.

4 **You are prompted**: Height of text <current height>:

5 Do one of the following:

- Press Enter or right click to accept the current height.
- Type a value.
- Click a point to define the height graphically.

You are prompted: Rotation angle of text <current angle>:

6 Do one of the following:

- Press Enter or right click to accept the current rotation angle.
- Type a value.
- Click a point to define the rotation angle graphically.

7 Type the text, then press Enter.

8 A new empty text is created.

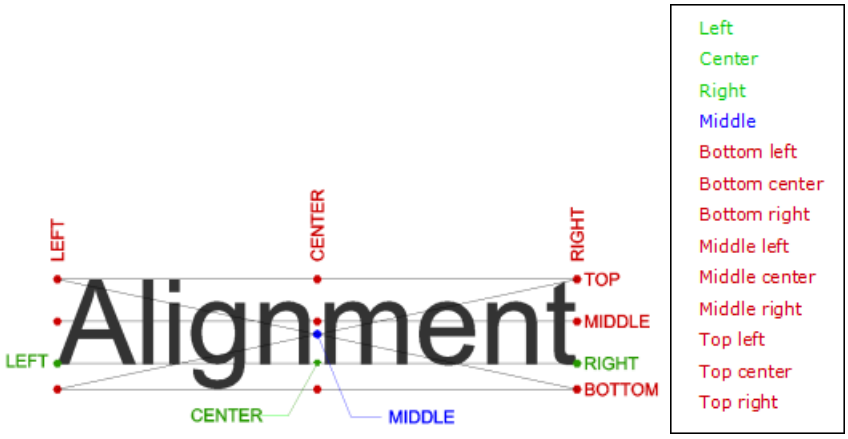
9 Do one of the following:

- Type the text.
- Press Enter to stop.

10 (option) Keep adding texts then press Enter twice to stop.

Uwaga: When you have already created text in the current BricsCAD session, the most recent text entity highlights when you repeat the TEXT command. Press Enter or right click to create a new text below the most recently created text, using the same text height and rotation angle.

Single line text justification options:



14.2.3.2 To create multiline text

1 Do one of the following:

- Click the **Multiline Text** tool button (A) on the **Draw** toolbar or the **Annotate / Text** ribbon panel.
- Choose **Multiline Text** in the **Draw** menu.
- Launch the MTEXT command.

You are prompted: First corner for block of text:

2 Specify the insertion point of the multiline text entity.

3 **You are prompted**: Justification/Rotation/Style/Height/Direction/Width/<Opposite corner for block of text>:

4 Do one of the following:

- Specify the opposite corner of the text block to define the maximum width of the multiline text.
- If **Dynamic Dimensions** are active, the height and width of the text field are displayed dynamically. Type the height and width in the dynamic dimension fields. Press the Tab key to switch; press Enter to confirm.
- Type @ in the Command line, then press Enter for non-wrapped (zero width) text (= unlimited line width).

The **Text formatting** toolbar opens right above the text block.

5 Type your text.

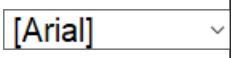
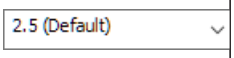
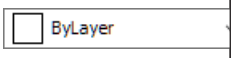
6 The maximum line width is defined by the width of the text block as defined in step 3.

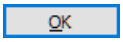
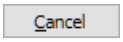
Press Enter to start a new paragraph.

7 To conclude the command and close the **Text formatting** toolbar do one of the following:

- Click the **OK** button
- Press Ctrl+Enter.
- Click outside the MText entity.

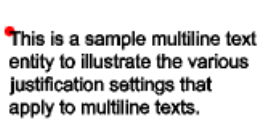
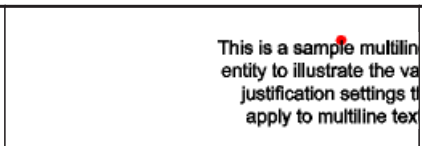
Text formatting toolbar tools and settings

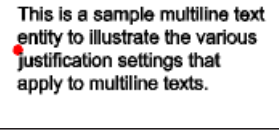
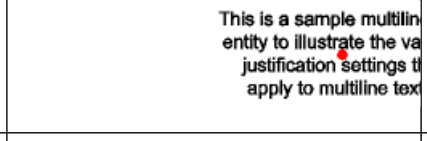
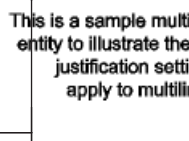
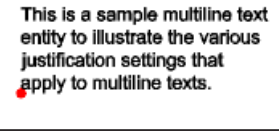
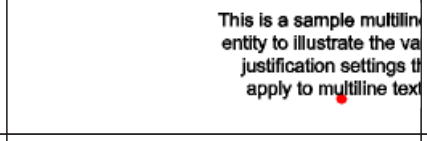
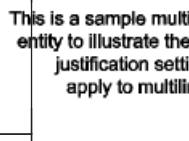
Button or setting	Name	Description
	Text Style	Sets the text style
	Text Font	Sets the text font
	Text Height	Sets the text height
	Bold	Creates bold text
	Italic	Creates italic text
	Underscore	Creates underscored text
	Overscore	Creates overscored text
	Uppercase	Converts selected text to uppercase
	Lowercase	Converts selected text to lowercase
	Annotative	Toggles the Annotative property
	Line Spacing	Sets the line spacing
	Columns	Arranges the text in columns
	Color	Sets the color

Button or setting	Name	Description
	Oblique Angle	Sets the oblique angle
	Width Factor	Sets the width factor
	Character Spacing	Sets the character spacing
	Undo	Undoes recent actions
	Redo	Redoes previously undone actions
	Stacked Fraction	Creates stacked fractions
	Special Character	Inserts special characters
	Field	Inserts a variable field
	Justify	Sets the text justification
	Background Mask	Adds a colored background
	Ruler	Toggles the ruler display
	OK	Closes the **Text formatting** toolbar and applies the changes
	Cancel	Closes the **Text formatting** toolbar and leaves the MText entity unchanged

Multiline text justification

The red dot indicates the insertion point of the multiline text entity.

		
Top left	Top center	Top right

		
Midde left	Middle center	Middle right
		
Bottom left	Bottom center	Bottom right

14.2.3.3 Including special text characters and symbols

You can use control codes and unicode strings to include special characters in text entities.

To include control codes, as you type text, type two percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character.

Special text characters

Control Code	Unicode String	Result
%%d	\U+00B0	Draws a degree symbol (°)
%%p	\U+00B1	Draws a plus-or-minus symbol (±)
%%c	\U+2205	Draws a diameter symbol (Ø)
%%k	\U+0336	Starts to strikethrough text. Repeat the code to stop.
%%u	\U+005F	Starts to underscore text. Repeat the code to stop.
%%o	\U+203E	Starts to overscore text. Repeat the code to stop.

Symbols and Unicode strings

Name	Symbol (*)	Unicode String
Almost equal	≈	\U+2248
Angle	∠	\U+2220

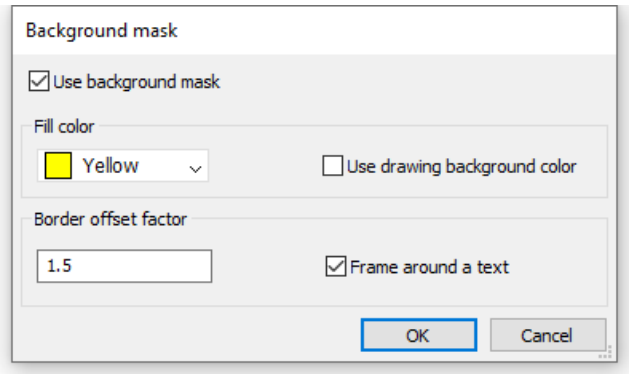
Name	Symbol (*)	Unicode String
Center line	℄	\U+2104
Delta	Δ	\U+0394
Electrical phase	ϕ	\U+0278
Identity	≡	\U+2261
Not equal	≠	\U+2260
Omega	Ω	\U+03A9
Squared	²	\U+00B2
Cubed	³	\U+00B3

(*) If supported by the text font.

14.2.3.4 To add a background mask

1 When creating or editing a multiline text entity, click the **Background Mask** tool button ( ) on the **Text Formatting** toolbar.

The **Background Mask** dialog box appears:



2 Check the **Use background mask** option.

3 Do one of the following:

- Choose a color in the **Fill color** drop-down list.
- Choose **Select color...** in the **Fill color** drop-down list and choose a color in the **Select color** dialog box.
- Check the **Use drawing background color** option.

4 Define the **Border offset factor**, which sets the margin around the text for the background. The value is based on the text height and must be in the range 1-5. A factor of 1.0 fits the multiline text entity exactly.

5 (option) Tick the **Frame around a text** option to create a text frame.

6 Click **OK** to create the background mask.

14.2.4 Editing text

You can edit and modify a text entity as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

You can edit text properties in the **Properties** panel.

The TEXTED system variable controls the text edit behavior Text entities:

- 0 = in-place editor
- 1 = pop up dialog
- 2 = in-place editor with repeated input

The MTEXTED system variable allows to use an external text editor for editing Mtext entities.

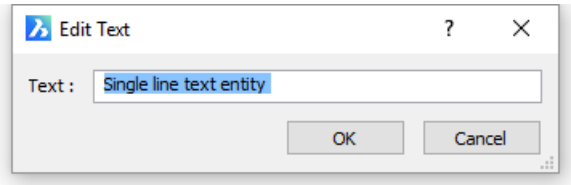
The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.

14.2.4.1 To edit a Text entity

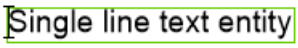
1 Double click the text entity.

2 Depending on the value of the TEXTED system variable:

3 The **Edit Text** dialog box appears



or the text is edited 'in place'.



4 Edit the text, then press Enter.

5 Do one of the following:

- Click another text entity to edit.
- Press Enter to stop editing text entities.

14.2.4.2 To edit an Mtext entity

1 Double click the Mtext entity.

2 The **Text formatting** toolbar displays.

3 Edit the text, then click the **OK** button or click outside the text block.

4 Do one of the following:
   - Click another Mtext entity to edit
   - Press Enter to stop editing Mtext entities.

14.2.4.3 To edit a selection of Mtext entities

1 Select the Mtext entities.

2 In the **Properties** panel select **Contents**.

3 Press the **Browse** button (...) on the right hand side of the **Contents** field.

4 The **Text formatting** toolbar opens for the first Mtext entity in the selection.

5 Edit the Mtext.

6 Press the **OK** button or click outside the text block.

The next Mtext entity in the selection can be edited now.

7 Repeat steps 5 and 6 until the last Mtext is edited.

14.2.4.4 To an external text editor for Mtexts

1 Launch the SETTINGS command to open the **Settings** dialog box.

2 Under **Drawing** > **Drafting** > **Entity** > **Creation** > **Texts** > **Multiline Text**, select the MTEXTED system variable.

3 Type the file name and full path of the external text editor in the Command line, then press Enter.

4 e.g. C:\Program Files\Windows NT\Accessories\Wordpad.EXE

Uwaga: To restore the built-in text editor, delete the content of the MTEXED system variable.

14.2.5 Rozbijający tekst

Polecenie TXTEXP rozbija elementy Tekst i Wtekst na polilinie 2D.

14.2.5.1 Aby rozbić tekst

1 Wybierz tekst(y)

2 Wykonaj jedną z następujących czynności:
   - Wybierać **Eksplodować** w **Edytować** zakładka **Kwadrat** .
   - Uruchom polecenie TXTEXP.

Uwagi:

- Pełne wypełnienie niektórych czcionek TTF jest usuwane podczas eksplozji takiego tekstu, pozostawiając tylko kontur tekstu.
- Użyj polecenia POŁĄCZ, aby utworzyć zamknięte polilinie z rozbitych konturów tekstu.
- **Rozbijając** narzędzie w **Edytorze** zakładki **Kwadratu** uruchamia polecenie TXTEXP. Polecenie ROZBIJ nie może rozbić elementów tekstowych.

14.2.6 Znajdowanie i zastępowanie tekstu

Polecenie ZNAJDŹ pozwala określić tekst, który chcesz znaleźć. Możesz znaleźć lub zamienić tekst. Opcjonalnie możesz powiększyć różne wystąpienia elementów tekstowych, które zawierają wyszukiwany tekst.

14.2.6.1 Aby ustawić opcje Znajdź i zamień

1 Uruchom polecenie ZNAJDŹ.

1 **Znajdź i zamień** pojawi się okno dialogowe.

2 Kliknij przycisk **Opcje**.

3 Zaznacz opcje na **Znajdź i zamień opcje** w oknie dialogowym.

4 Kliknij przycisk **OK** .

14.2.6.2 Aby znaleźć tekst

1 Uruchom polecenie ZNAJDŹ.

1 **Znajdź i zamień** pojawi się okno dialogowe.

2 (opcjonalnie) Ustaw opcje **Znajdź i zamień**.

3 (opcjonalnie) Kliknij przycisk **Wybierz elementy** ( ), aby utworzyć zbiór wskazań.

Jeśli nie wybrano żadnych elementów, przeszukiwany jest cały rysunek.

- **Okno dialogowe Znajdź i zamień zostaje** tymczasowo zamknięte, aby umożliwić wybór elementów .
- Kliknij prawym przyciskiem myszy, aby zakończyć wybór obiektów.

Pole **Szukaj w** odczytuje **bieżący wybór**, a nie **cały rysunek**.

4 Wpisz wyszukiwany tekst w polu **Znajdź ciąg tekstowy** .

5 Kliknij **Odnaleźć** przycisk.

6 Zawartość pierwszej jednostki tekstowej zawierającej ciąg wyszukiwania jest wyświetlana w okienku **Wyniki wyszukiwania.**

7 (opcjonalnie) Kliknij przycisk **Zoom To** .

Rysunek jest powiększany na elemencie tekstowym.

8 Wykonaj jedną z następujących czynności:

9 Powtórz kroki 5 i 6, aby znaleźć więcej wystąpień tego samego ciągu wyszukiwania.

10 Powtórz kroki od 2 do 6, aby znaleźć inny ciąg tekstowy.

11 Naciśnij klawisz Esc lub kliknij przycisk **Zamknij** , aby zatrzymać.

14.2.6.3 Aby zamienić tekst

1 Uruchom polecenie ZNAJDŹ.

1 **Znajdź i zamień** pojawi się okno dialogowe.

2 (opcjonalnie) Ustaw opcje **Znajdź i zamień**.

3 (opcjonalnie) Kliknij przycisk **Wybierz elementy** ( ), aby utworzyć zbiór wskazań.

Jeśli nie wybrano żadnych elementów, przeszukiwany jest cały rysunek.

- **Okno dialogowe Znajdź i zamień zostaje** tymczasowo zamknięte, aby umożliwić wybór elementów .
- Kliknij prawym przyciskiem myszy, aby zakończyć wybór obiektów.

Pole **Szukaj w** odczytuje **bieżący wybór**, a nie **cały rysunek**.

4 Wpisz wyszukiwany tekst w polu **Znajdź ciąg tekstowy** .

5 Wpisz tekst zastępczy w **Zamienić** pole, a następnie naciśnij klawisz Enter.

Wymienione są wszystkie encje tekstowe zawierające wyszukiwany ciąg.

6 Wykonaj jedną z następujących czynności:

- Kliknij element na liście, aby podświetlić element tekstu na rysunku, a następnie kliknij **Zastępować** przycisk.
- Kliknij przycisk **Zamień**.

14.2.7 Checking spelling

The spelling checker checks the spelling of text in the current drawing. You can check the spelling of one or more selected text entities or check the entire drawing.

During the spelling check, the SPELL command matches the words in the drawing or the current selection set to the words in the current main dictionary and the current custom dictionary. If a word is not found in either dictionaries you can select the correct spelling in the suggestions field or you can add the word to the current custom dictionary. Custom dictionaries can be used for discipline-specific words.

14.2.7.1 Dictionary settings

Which dictionaries are used by the SPELL command is defined by the DCTMAIN (main dictionary) and DCTCUST (custom dictionary) system variables.

Uwaga: We only deliver the English dictionary because of the license type on the dictionary files: the English version can be distributed freely, while all other languages are not free. However, you can download other languages and use them for free (see To add a main dictionary below).

14.2.7.2 To open the Spell Checking dialog box

Do one of the following:

- Choose **Check Spelling** in the **Tools** menu.
- Click the **Check Spelling** tool in the **Annotate / Text** ribbon panel.
- Launch the SPELL command.

14.2.7.3 To check the spelling

1 Open the **Spell Checking** dialog box.

2 (option) Click the **Select** button ( ) to compose a selection set.

If no entities are selected, the entire drawing is searched.

- The **Spell Checking** dialog temporarily closes to let you select entities.
- Right click to conclude the selection of entities.

The **Where to check** field reads **Current Selection** instead of **Entire Drawing**.

3 (option) Change the current dictionaries (see below).

4 Click the **Start** button.

The first possibly misspelled word is displayed in the **Current word** section.

5 The content of the text entity in which the **Current word** is found, is shown in the **Context** field.

6 Do one of the following:

- Click the **Ignore** button to leave this instance of the **Current word** unchanged.
- Click the **Ignore All** button to leave all instances of the **Current word** unchanged.
- Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change** button to replace this instance of the **Current word** by the word in the **Suggestions** field.
- Accept the word in the **Suggestions** field or select a word in the **Suggestions** list, then click the **Change All** button to replace all instances of the **Current word** by the word in the **Suggestions** field.
- Click the **Add** button to add the **Current word** to the current custom dictionary.
- Click the **Lookup** button to list words similar to the word in the **Suggestions** field.

7 Repeat the previous step until the **Spelling check is complete** message appears.

14.2.7.4 To change dictionaries

1 Open the **Spell Checking** dialog box.

2 Click the **Change Dictionaries...** button.

The **Change Dictionaries** dialog box appears.

3 Click the **Main dictionary** list button to change the main dictionary.

4 Select a new main dictionary.

5 (option) Click the **Browse...** button to select a new **Custom dictionary**.

6 (option) Type a word in the **Custom dictionary words** field, then click the **Add** button to add a word to the current custom dictionary.

7 Click the **Apply & Close** button.

Uwaga:

- Custom dictionaries are best saved in the **Support** folder of the **Roamable** root folder.
- If you want to share the custom dictionary with MS Word, point to the MS Word custom dictionary in step 5 in the above procedure. By default, CUSTOM.DIC is the custom dictionary in MS Word, which can be found under *C:\Users\<user name>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

14.2.7.5 To add a new main dictionary

1 Open the **Spell Checking** dialog box.

2 Click the **Change Dictionaries...** button.

3 Click the **Download...** button.

Your browser opens: https://extensions.openoffice.org/dictionaries

4 Scroll down to the language of your choice, then click the link to open the page of the selected language.

5 Click the **Download extension** button to start downloading the ***.oxt** file.

6 Change the **oxt** extension to **zip**.

7 Extract the ***.aff** and ***.dic** files from the ZIP file to the folder **C:\Program Files\Bricsys\BricsCAD Vx \Support**.

The dictionary is now available in the **Main Dictionary** list on the **Change Dictionaries** dialog box.

14.2.7.6 To add a new custom dictionary

1 Open a text editor, such as **Notepad**.

2 (option) Type the words you want to use in your custom dictionary.

3 Each word must be on a separate line.

4 Save the file in the **Support** folder of the **Roamable** root folder using the **.cus** extension.

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.

Uwaga: 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 Working with tables

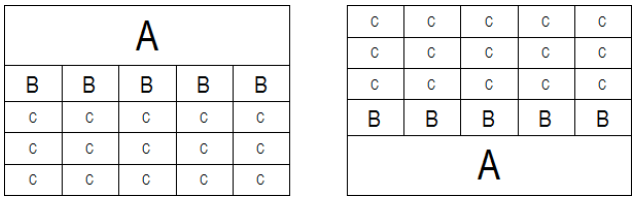
A table is an entity that is composed of rows and columns, much like a Microsoft Excel spreadsheet. A table entity can be created as an empty table or you can import data from a CSV (Comma Separated Values) file. The appearance of the table text and gridlines is controlled by a table style. Table styles are saved in the DWG file.

A default table consists of:

- a **Title** cell (A)
- a row of **Header** cells (B)
- one or more rows of **Data** cells (C).

A table can have one of two directions:

- **Down** (title row at the top)
- **Up** (title row at the bottom).



14.2.9.1 To create an empty table

1 Do one of the following:

- Click the **Table** tool button ( ) on the **Draw** toolbar or the **Home / Annotations** ribbon panel.
- Choose **Insert Table...** in the **Draw** menu.
- Launch the TABLE command.

The **Insert Table** dialog box appears.

2 Choose a Table Style from the **Table Style** list.

3 (option) Set the cell style for the first, second and other rows.

4 By default the cell style for the first row is **Title**, the cell style for the second row is **Header** and the cell style for the other rows is **Data**.

5 Under **Table Options**, choose **Start from Empty Table**.

6 Under **Insertion Behavior**, do one of the following:

- Choose **Specify Insertion Point**, then specify the number of columns and the column width and number of data rows and the row height.
- Choose **Specify Window**, then do one of the following:

• Specify the number of columns and data rows.

• Specify the column width and the row height.

• Specify the column width and the number of data rows.

• Specify the number of columns and the row height.

7 Click the **OK** button.

Depending on your choice in step 5, **you are prompted** to specify an insertion point or a window.

The table is created and the **Text Formatting** toolbar appears (see Creating Texts).

8 Do one of the following:

- Start adding text in the table.
- Click outside the table to close the **Text Formatting** toolbar.

14.2.9.2 To create a table from data

When importing data from a CSV (Comma Separated Values) file, please make sure the list separator character on your system is the same as the one used in the CSV file (see Defining the list separator character).

1 Do one of the following:

- Click the **Table** tool button ( ) on the **Draw** toolbar or the **Home / Annotations** ribbon panel.

- Choose **Insert Table...** in the **Draw** menu.
- Launch the TABLE command.

The **Insert Table** dialog box appears.

2 Under **Table Options**, choose **From Data**.

3 Click the **Browse** button (...).

4 The **Open** dialog box appears.

Choose a CSV file, then click the **Open** button.

5 Click the **OK** button on the **Insert Table** dialog.

6 Click a point in the drawing to place the table.

7 The table is created and the **Text Formatting** toolbar appears (see Creating Texts).

8 Do one of the following:
- Start editing the text in the table.
- Click outside the table to close the **Text Formatting** toolbar.

9 (option) Select the table and drag the grips to adjust the dimensions of the table.

14.2.9.3 To create a table from an Excel spreadsheet

1 Open the spreadsheet in **Microsoft Office Excel**.

2 Select the cells you want to create a table from.

3 Do one of the following to copy the selection to the clipboard:
- Press **Ctrl + C**.
- Right click and choose **Copy** in the context menu.

4 In BricsCAD, choose **Paste special...** in the **Edit** menu or launch the PASTESPEC command.

5 The **Paste Special** dialog box appears.

6 Choose **XML Spreadsheet** in the **As** option list.

7 Click in the drawing to insert the table.

14.2.9.4 Editing a table using grips

When a table is selected, multiple grips are shown:

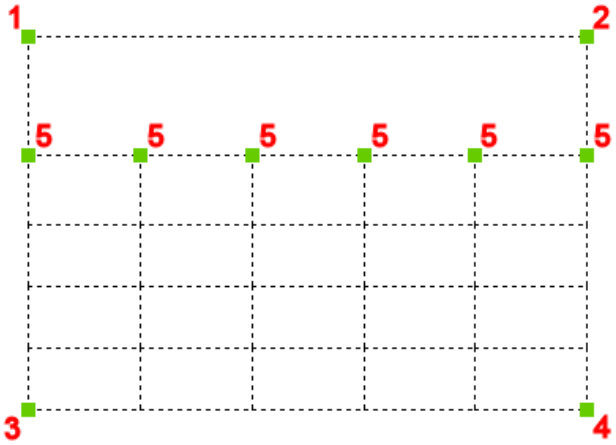
1 Moves the table.

2 Modifies the width uniformly.

3 Modifies the height uniformly.

4 Modifies the width and height uniformly.

5 Modifies the width of a column.



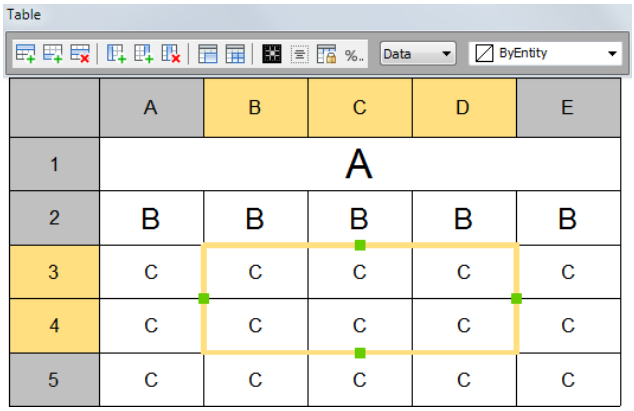
14.2.9.5 Editing a table using the Table toolbar

1 To open the **Table** toolbar, do one of the following:
   - Click the cell you want to edit.
   - Press and hold the left mouse button then define a rectangle to select multiple cells.

2 Column letters and row numbers are displayed.

3 The column(s) and row(s) of the selection are highlighted.

   A rectangle with four handles is drawn around the selection.



4 (option) Drag the grips of the selection to modify the dimensions.

5 (option) Click a tool in the **Table** toolbar.

6 (option) Right click to open a context menu:
   - **Delete Cell Content**: deletes the text in the selected cells.
   - **Clear Cell Overrides**: resets all cell properties of the selected cells to the **Cell Style** settings as defined in the **Table Style**.
   - **Clear Table Overrides**: resets all cell properties of all cells to the **Cell Style** settings as defined in the **Table Style**.

7 (option) Repeat steps 3 and 4 to continue editing the table.

8 (option) Double click a cell to edit the cell text.

9 (option) Do one of the following:
- Click another cell.
- Press and hold the left mouse button, then drag the cursor to select multiple cells.

10 Click outside the table to close the **Table** toolbar.

Table tools

Icon	Tool name	Description
	Insert row above	Inserts an empty row above the selection.
	Insert row below	Inserts an empty row below the selection.
	Delete row	Deletes the row(s) of the selection.
	Insert column left	Inserts an empty column at the left of the selection.
	Insert column right	Inserts an empty column at the right of the selection.
	Delete column	Deletes the column(s) of the selection.
	Merge	Merges the selected cells.
	Unmerge	Splits previously merged cells.
	Border style	Sets the border style of the selection (not implemented yet).
	Text alignment	Sets the text alignment of the selection.
	Lock / Unlock	Locks / Unlocks the selection.
	Set cell format	Sets the cell format of the selection (not implemented yet).
	Set cell style	Sets the style of the selected cells: **Title**, **Header** or **Data**.
	Set cell background	Sets the background color of the selection.

14.2.9.6 Editing a table in the Properties panel

1 Click one of the grid lines.

1 The properties of the table are displayed in the **Properties** panel.

2 (option) Apply a different **Table Style**.

3 (option) Change the table direction.

4 (option) Modify the width and/or height of the table.

5 (option) Modify the vertical and/or horizontal cell margins.

6 Press the Esc key to stop editing.

14.2.9.7 Editing a cell in the Properties bar

1 Click the cell.

1 The properties of the cell are displayed in the **Properties** panel.

2 (option) Modify the width and/or height of the cell.

3 The width and height of the column and/or row of the cell are changed.

4 (option) Modify the text rotation.

5 The options are: 0°, 90°, 180° and 270°.

6 (option) Edit the cell content.

7 (option) Choose a different text alignment option.

8 Press the Esc key to stop editing.

14.2.9.8 Evaluating formulas

Currently formulas are supported partially. There are more complex formulas which are not supported yet, but in most cases formulas in Excel sheets will remain functional when the Excel sheet is imported as a table. Editing and creating formulas through the field dialog is not supported yet, but in-place editing/creating is possible. E.g.: if you type =SUM(A2:A4) in a cell, it will be interpreted as a formula, and the sum of the values in the cells A2 through A4 is calculated.

14.2.9.9 To edit the table content

1 Do one of the following:

- Double click the cell you want to edit.
- Launch the TABLEDIT command.
- **You are prompted**: Pick table cell to edit.

  Click the cell you want to edit.

The **Text Formatting** toolbar is displayed above the selected cell (see Creating Texts).

2 Edit the cell.

3 (option) Press Alt + Enter to create a second line in the same cell.

4 To edit another cell:

- Press the Tab key to go to the next cell in the same row.
- Press Shift + Tab to go to the previous cell in the same row.
- Press Enter to go the next cell in the same column.
- Press Shift + Enter to go to the previous cell in the same column.
- Press one of the arrow keys.

5 Click outside the table to stop editing.

14.2.9.10 To export a table

When exporting data to a CSV (Comma Separated Values) file, the list separator character off your system is used in the CSV file (see Setting the list separator character).

1 Launch the TABLEEXPORT command.

1 **You are prompted**: Select table to export:

2 Click the table you want to export the data of.

3 The **Export Data** dialog opens.

4 Select a folder to save the data in.

5 Type a name in the **File name** field.

6 Click the **Save** button.

The data in the selected table are saved as a CSV file.

14.2.9.11 Setting the list separator character

1 Open the Windows **Settings** dialog box.

2 Open the **Regions** settings category.

3 Click **Additional date, time and regional settings**.

4 Click **Change date, time or number formats**.

5 Click **Additional settings**.

6 Set the **List separator** character.

7 Click **OK** to confirm.

14.3 Annotative Entities

14.3.1 Annotative entities

Annotations are all entities in your drawing which are not a model or geometry, such as dimensions, texts, symbols, etc. A common problem with annotations is that they are often scale dependent. Some annotations must be scaled to the plot scale, or are meant to be shown at certain scales only.

Annotative styles and entities change the way in which they are displayed (size, location, and orientation) depending on the current scale factor and the viewport.

The following entities can have the annotative property:

- text and multiline text
- hatches
- blocks and attributes
- dimensions and tolerances
- leaders and multileaders

The following styles can have the annotative property:

- text styles (see the STYLE command)
- dimension styles (see the DIMSTYLE command)
- multileader styles (see the MLEADERSTYLE command)

Uwaga:

- Entities that are created using an annotative style, get the current annotation scale assigned automatically.
- When an annotative style is applied to a non-annotative entity, the annotative property is not assigned automatically.

14.3.1.1 Commands

ANNORESET: Resets all scale representations of the selected annotative entities to the location of their current scale representation, thus synchronizing all scale representations.

ANNOUPDATE: Updates the selected annotative entities to match their current style definition.

OBJECTSCALE: Adds or removes supported scales for annotative entities through a dialog box.

-OBJECTSCALE: Adds or removes supported scales for annotative entities at the command line.

SCALELISTEDIT: Edits the scale list of a drawing, used in **Plot Scale** section of the **Print** and **Page Setup** dialogs and the **Standard scale** property of a paper space viewport. Defines the available annotation scales for annotative entities.

-SCALELISTEDIT: Edits the scale list of a drawing at the Command line.

14.3.1.2 System variables

ANNOALLVISIBLE: Toggles the display of annotative entities that do not support the current annotation scale.

ANNOTATIVEDWG: Specifies whether or not the drawing behaves as an annotative block when inserted into another drawing.

CANNOSCALE: Sets the name of the current annotation scale for the current space. Only scales that exist in the current scale list are accepted. The CANNOSCALE setting is saved with model space and with each layout viewport in the various layouts. When you create an annotative entity the current annotative scale is applied automatically.

CANNOSCALEVALUE: Displays the value of the current annotation scale. E.g. the value of the 1:2 scale is 0.5.

HPANNOTATIVE: Controls whether new hatch entities are annotative.

MSLTSCALE: If ON, linetypes in model space are scaled to the current annotation scale. A regen is needed when you toggle the value of MSLTSCALE. This variable is the equivalent of the PSLTSCALE (Paperspace linetype scale) variable in paper space.

MSOLESCALE: Controls the size of an OLE object containing text when pasted in model space. The MSOLESCALE system variable only affects the initial scaling. Existing OLE objects are not affected if the variable is modified.

SAVEFIDELITY: Controls whether the current display of the drawing is preserved when opened in a program that does not support annotative entities.

SELECTIONANNODISPLAY: Controls whether all scale representations are displayed when an annotative entity is selected.

14.3.1.3 Setting the current annotation scale

To set the current annotation scale, do one of the following:

- Click the **Annotation Scale** field under **Misc** in the **Properties** panel, then press the down-arrow button and select a scale in the list.
- Right click the **Annotation Scale** field in the **Status** bar, then select a scale in the list. If the **Annotation Scale** field is not available, click the down arrow at the right hand side of the **Status** bar and choose **Annotation Scale** in the list.

Uwaga: Only scales that exist in the **Scale** list of the drawing are available. Use the SCALELISTEDIT command to edit the **Scale** list.

14.3.1.4 To make an annotation entity annotative

Annotation entities which are created using a non-annotative style can be made annotative using the following procedure:

1 Select the annotation entity.

2 In the **Properties** panel, click the **Annotative** property.

3 Select **Yes**.

The current annotation scale is applied to the selected entity.

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.

Uwaga: 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 Teksty opisowe

Elementy tekstowe są tworzone przy użyciu bieżącego stylu tekstu:

- Jeśli styl tekstu jest opisowy, element tekstowy automatycznie pobiera właściwość opisową i przypisywana jest bieżąca skala opisu.
- Jeśli styl tekstu nie jest opisowy, musisz ręcznie zmienić tekst w opisowy.

Teksty opisowe mają dwie właściwości wysokości:

- **Wysokość tekstu modelu** : wysokość wyświetlania na ekranie w obszarze modelu, w odniesieniu do bieżącej skali adnotacji.
- **Wysokość tekstu na papierze** : wysokość na papierze, jeśli została wydrukowana w obsługiwanej skali adnotacji.

14.3.3.1 Aby nadać encji tekstowej opisowy

1 Wybierz element tekstowy.

1 Właściwości wybranej encji tekstowej są wyświetlane w **Nieruchomości** płyta.

2 w **Nieruchomości** panelu, wybierz **Opisowy** własność.

3 Wybierać **tak** z listy opcji.

Przypisana jest bieżąca skala opisu.

Tekst nie jest skalowany: aktualna wysokość tekstu staje się **Wysokość tekstu modelu** .

14.3.3.2 Aby usunąć właściwość opisową

1 Wybierz element tekstowy.

1 Właściwości wybranej encji tekstowej są wyświetlane w **Nieruchomości** płyta.

2 w **Nieruchomości** panelu, wybierz **Opisowy** własność.

3 Wybierać **Nie** z listy opcji.

Jeśli element tekstowy obsługuje wiele skal adnotacji, tekst jest przeskalowywany z powrotem do jego początkowej skali.

14.3.4 Opisowe kreskowania i wypełnienia gradientowe

Zmienna systemowa HPANNOTATIVE określa, czy nowe elementy kreskowania lub gradientu są opisowe.

Opcja HPANOTATIVE jest domyślnie WYŁĄCZONA. Zmienna systemowa nie jest zapisywana i jest resetowana do wartości domyślnej po zamknięciu BricsCAD.

14.3.4.1 Aby ustawić zmienną systemową HPANNOTATIVE

Wykonaj jedną z następujących czynności:

- w **Ustawienia** okno dialogowe, pod **Redakcja** > **Tworzenie jednostki** > **Włazy**.
- Sprawdź **Opisowe** opcje na **Kreskowanie i gradient** Okno dialogowe.
- Wybierz **Opisowy** w wierszu polecenia polecenia -KRESKUJ.

14.3.4.2 Aby uczynić kreskowanie lub element gradientu opisowym

1 Wybierz element kreskowania lub wypełnienia gradientowego.

1 Właściwości wybranej jednostki są wyświetlane w **Właściwości** płyta.

2 Wybierz **Opisowy** własność.

3 Wybierać **tak** z listy opcji.

Bieżąca skala opisu jest stosowana do wybranego kreskowania lub elementu gradientu, a kreskowanie jest odpowiednio skalowane.

14.3.4.3 Aby usunąć właściwość opisową

1 Wybierz kreskowanie lub wypełnienie gradientowe.

1 Właściwości wybranej jednostki są wyświetlane w **Właściwości** płyta.

2 Wybierz **Opisowy** własność.

3 Wybierać **Nie** z listy opcji.

14.3.5 Bloki opisowe

Właściwość opisowa bloku jest ustawiana podczas tworzenia definicji bloku za pomocą polecenia BLOK lub później, w **Eksplorator rysunków / bloki** dialog.

Bloki opisowe mogą mieć opcjonalną właściwość, aby zawsze odpowiadać orientacji układu, niezależnie od obrotu zawartości rzutni obszaru papieru.

14.3.5.1 Aby utworzyć opisową definicję bloku

1 Uruchom polecenie EXPBLOCKS.

1 ten **Eksplorator rysunków / bloki** pojawi się okno dialogowe.

2 W razie potrzeby wybierz **Widok Szczegółowy** .

Definicje bloków opisowych mają **Opisowe** Ikony ( ) w **Opisowych** kolumnach.

3 Kliknij w **Opisowe** pole definicji bloku, aby włączyć/wyłączyć właściwość opisową.

14.3.5.2 Aby ustawić Orientację dopasowania na właściwość Układ

1 Uruchom polecenie EXPBLOCKS.

1 Wyświetlone zostanie okno dialogowe **Solids**

2 W razie potrzeby wybierz **Widok Szczegółowy** .

3 Zaznacz **Dopasuj orientację do układu** w polu wyboru definicji bloku, aby ustawić właściwość.

14.3.6 Wymiary opisowe

Elementy wymiaru są tworzone przy użyciu bieżącego stylu wymiarowania:

- Jeśli styl wymiarowania jest opisowy, wymiary automatycznie otrzymują właściwość opisową i przypisywana jest bieżąca skala opisu.
- Jeśli styl wymiarowania nie jest opisowy, należy ręcznie nadać wymiarom opisowe.

Uwagi:

- ten **Całkowita skala przyciemnienia** (zmienna systemowa DIMSCALE) nie można edytować w przypadku opisowych stylów wymiarowania i jest pomijane w przypadku wymiarów opisowych.
- Zaleca się ustawienie **Całkowita skala przyciemnienia** = 1 dla opisowych stylów wymiarowania.

14.3.6.1 Aby uczynić jednostkę wymiaru opisowym

1 Wybierz element wymiaru.

1 Właściwości wybranej jednostki wymiaru są wyświetlane w **Nieruchomości** płyta.

2 Wybierz **Opisowy** własność.

3 Wybierz **Nie** z listy opcji.

Jednostka wymiaru jest skalowana w odniesieniu do bieżącej wartości zmiennej systemowej DIMSCALE.

14.3.7 Liderzy opisowi

Wielolinie odniesienia są tworzone przy użyciu bieżącego stylu wielolinii odniesienia:

- Jeśli styl wielolinii odniesienia jest opisowy, Mleader jednostka automatycznie pobiera właściwość opisową i przypisywana jest bieżąca skala opisu.

- Jeśli styl wielolinii odniesienia nie jest opisowy, należy ręcznie nadać elementowi wielolinii odniesienia opisowy.

14.3.7.1 Aby nadać elementowi wielolinii odniesienia opisowy

1 Wybierz **Wielolinia odniesienia** podmiot.

Właściwości wybranego **Wielolinia odniesienia** podmiot są wyświetlane w **Nieruchomości** płyta.

2 Wybierz **Opisowy** własność.

3 Wybierać **tak** na liście opcji.

Przypisana jest bieżąca skala opisu.

ten **Wielolinia odniesienia** jednostka jest skalowana w odniesieniu do bieżącej skali adnotacji.

14.3.7.2 Aby usunąć właściwość opisową

1 Wybierz **Wielolinia odniesienia** podmiot.

Właściwości wybranego **Wielolinia odniesienia** podmiot są wyświetlane w **Nieruchomości** płyta.

2 Wybierz **Opisowy** własność.

3 Wybierać **Nie** z listy opcji.

15. Dostosowanie

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 "/".

Uwaga:

- 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 BricsCAD user files

In BricsCAD user files are stored in specific folders:

- **PlotConfig**: plotter configuration files (*.pc3)
- **PlotStyles**: plot style table files (*.stb) and plot color table files (*.ctb)
- **Support**:
  - hatch pattern files (*.pat)
  - line style files (*.lin)
  - alias files (*.pgp)
  - unit files (*.unt)
  - custom user interface files (*.cui)
  - BIM library files (*.bsyslib) see BIMPROJECTINFO command.
  - Structure tree configuration files (*.cst) see STRUCTUREPANEL command.

- Library panel configuration files (*.ccf) see LIBRARYPANELOPEN command.

- **Templates**: template drawings (*.dwt)

Platform	Folders	Subfolders
Windows	C:\Users\<UserName>\AppData\Local \Bricsys \BricsCAD\<Version>\en_US	\Templates
Windows	C:\Users\<UserName>\AppData\Roaming \Bricsys\BricsCAD\<Version>\en_US	\PlotConfig \PlotStyles \Support
Mac	/Users/<UserName>/Library/Preferences/ Bricsys/BricsCAD/<Version>/en_US	/PlotConfig /PlotStyles /Support /Templates
Linux	/home/<UserName>/Bricsys/BricsCAD/ <Version>/en_US	/PlotConfig /PlotStyles /Support /Templates

15.2.1 The BricsCAD user file manager

When you update, the first time you launch BricsCAD after the update, the content of the **UserDataCache** subfolder of the BricsCAD installation folder is compared with the content of the **user file folders**. If you have customized files in these folders and if one or more of the corresponding files in the **UserDataCache** folder are more recent, the **BricsCAD user file manager** dialog box appears, asking you whether to keep or to overwrite your files with the new version.

Do one of the following:

1 Select a file in the file list, then click the **Yes** button to overwrite or click the **No** button to keep your file.
   - Repeat this procedure for each file in the list.
   - Click the **Yes to all** button to replace all your files with the more recent files from BricsCAD.
   - Click the **Stop Copying** button to keep all your files.

2 If you have decided not to update one or more files, an alert window appears, listing all files that you want to keep and that therefore will not be updated. You are asked whether or not to be reminded the next time you start BricsCAD.

3 Do one of the following:
   - Click the **Yes** button if you want to update one of these files later.

     In this case this procedure is restarted the next time you start BricsCAD.
   - Click the **No** button if you do not want to be reminded.

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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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 Customizing the ribbon

The ribbon is composed of a series of tabs, which in turn are composed of panels. Ribbon panels are composed of rows with tools, controls and sub-panels.

Ribbon tabs control which ribbon panels are displayed on the ribbon.

To display a ribbon tab on the ribbon, you must add the tab to a workspace.

15.12.1 Creating a ribbon tab

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Do one of the following:

- Expand a menu group, then right-click **Ribbon tabs** and choose **Append ribbon tab** in the context menu.

  The new ribbon tab will be added below the last existing ribbon tab.

- Select an existing ribbon tab in a menu group, then right-click and choose **Insert ribbon tab** in the context menu.

  The new ribbon tab will be added above the selected ribbon tab.

The **Add ribbon tab** dialog box appears.

3 Define the ribbon tab:

- **Label**: Name of the tab as displayed in the ribbon.
- **Title**: Long name of the tab.
- **ID**: Unique tag to identify the tab in the menu group. The ID is used to reference the ribbon tab programmatically.

4 Click the **OK** button to create the ribbon tab.

15.12.2 Creating a ribbon panel

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Do one of the following:

- Expand a menu group, then right-click **Ribbon panels** and choose **Append ribbon panel** in the context menu.

  The new ribbon panel will be added below the last existing ribbon panel.

- Select an existing ribbon panel in a menu group, then right-click and choose **Insert ribbon panel** in the context menu.

  The new ribbon panel will be added above the selected ribbon panel.

The **Add ribbon panel** dialog box appears.

3 Define the ribbon panel:

- **Label**: Name of the panel as displayed in the ribbon.
- **Title**: Long name of the panel.
- **ID**: Unique tag to identify the panel in the menu group. The ID is used to reference the ribbon panel programmatically.

4 Click the **OK** button to create the ribbon panel.

15.12.3 Adding a ribbon panel to a tab

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Expand **Ribbon tabs** in a menu group.

3 Do one of the following:

- Right-click a ribbon tab and choose **Append panel** in the context menu.

  The ribbon panel will be added below the last existing ribbon panel.

- Expand a ribbon tab, then right-click and choose **Insert panel** in the context menu.

  The ribbon panel will be added above the selected ribbon panel.

The **Select ribbon panel** dialog box appears.

All existing ribbon panels in the menu group are listed.

4 Select a panel in the list, then double-click or click the **OK** button.

The panel is added to the tab.

15.12.4 Adding a row to a panel

1 In the **Customize** dialog box, click the **Ribbon** tab.

2 Expand **Ribbon panels** in a menu group.

3 Select a panel, then do one of the following:

- Right-click and choose **Append row** in the context menu.

  The row will be added below the last existing row in the selected panel.

- Select an existing row then right-click and choose **Insert ribbon row** in the context menu.

  The row will be added above the selected row.

15.12.5 Adding items to a row

When you right-click a row, the following options are available in a context menu:

- **Add row panel**: Adds a subpanel containing three subpanel rows by default. You can then add one or more items to these rows and add/remove rows.
- **Add command button**: See **Adding tools to user interface controls**.
- **Add split button**: Adds a button to which you can add multiple items. Only the first item or the most recently used item is displayed.
- **Add toggle button**: Adds a button to toggle a setting or an option. E.g. entity snaps or selection priorities. A toggle button indicates the state of the setting or option.
- **Add Separator**: Adds a space between rows.

15.12.6 Defining the properties of an item

Select the item, then define the properties in the property grid in the bottom section of the **Customize** dialog box.

15.12.7 Repositioning items

To reposition any item in a panel or a row: select the item, then press and hold the left mouse button and move the item to the desired position. Release the mouse button to place the item.

15.12.8 Deleting items

Select the item, then right-click and choose **Delete** in the context menu.

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

Uwaga: 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 (').

Uwaga:

- 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.

Uwaga: 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.

Uwaga:

- 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.

Uwaga:

- 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.

Uwaga: 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

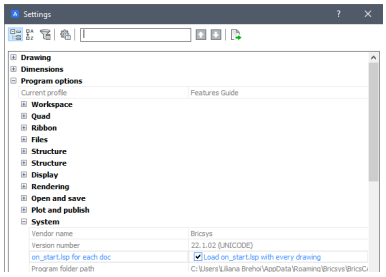
Migrating from AutoCAD

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.

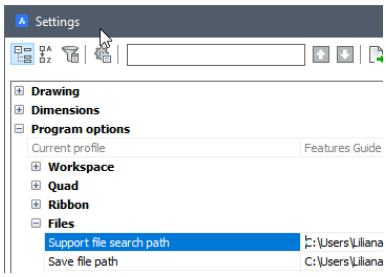
BricsCAD Settings Panel

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 out 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.

> ***Uwaga:*** 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"))
```

> ***Uwaga:*** **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")       ;If( file) tell us 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. Modifying entities

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

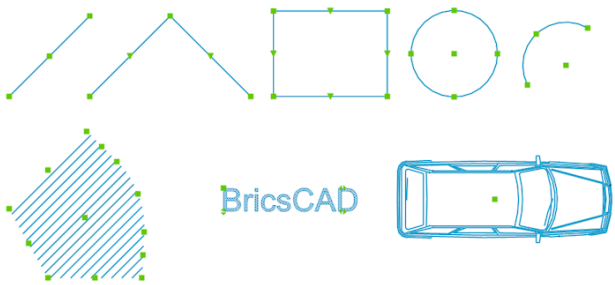
  BIM data, including IFC properties and user defined properties are copied, except:

  - 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.

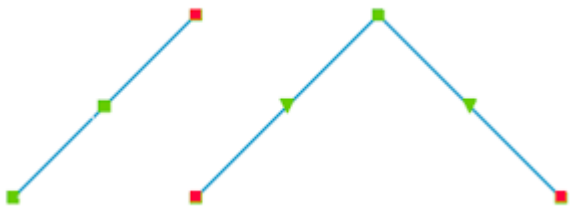
Uwaga: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

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.

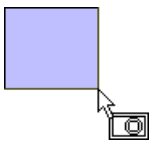
16.5.1 Composing a selection set before (pre-pick)

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.

Uwaga:

- 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.

Uwaga:

- 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.

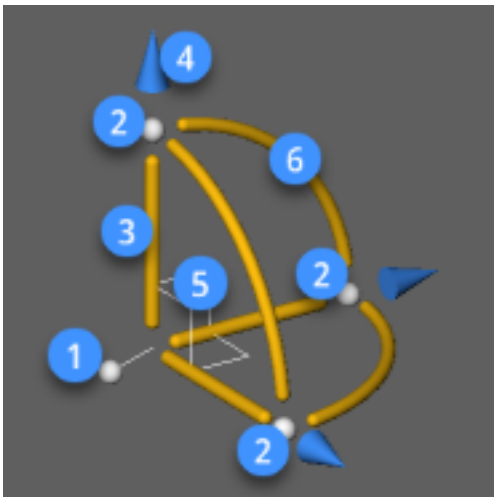
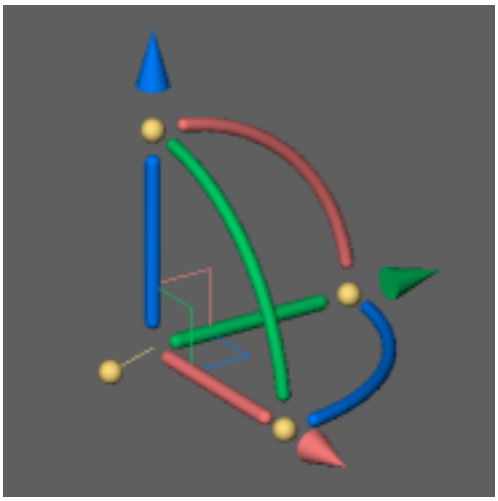
16.8.2 The manipulator settings

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

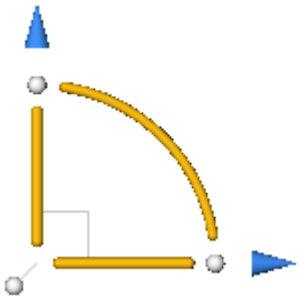
4 handles

- **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.

Uwaga: 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.

Uwaga: 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.

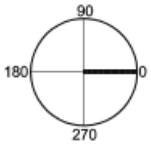
8 The selection is pasted as a block, using a default name.

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.



Uwaga:

- 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.

Uwaga: 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:

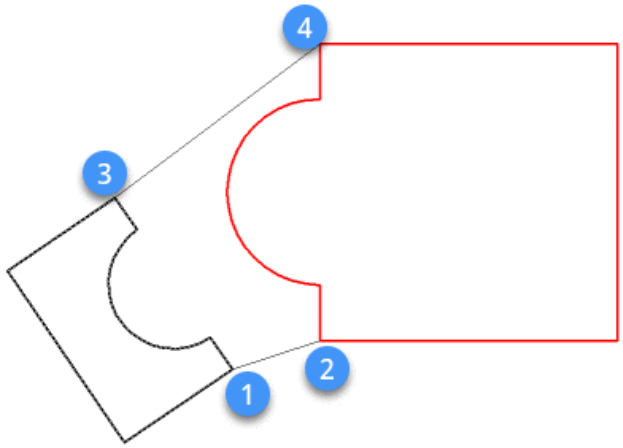
5 Snap to the first source point in the selection set (1).

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.

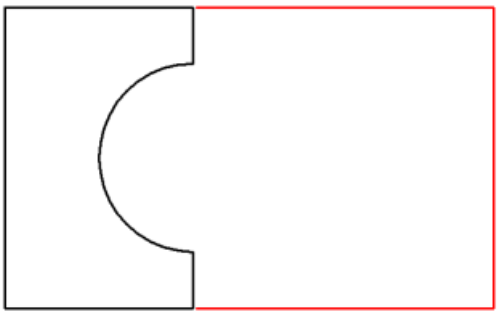
14 **You are prompted**: Scale objects based on alignment points [Yes/No] <No>:

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:

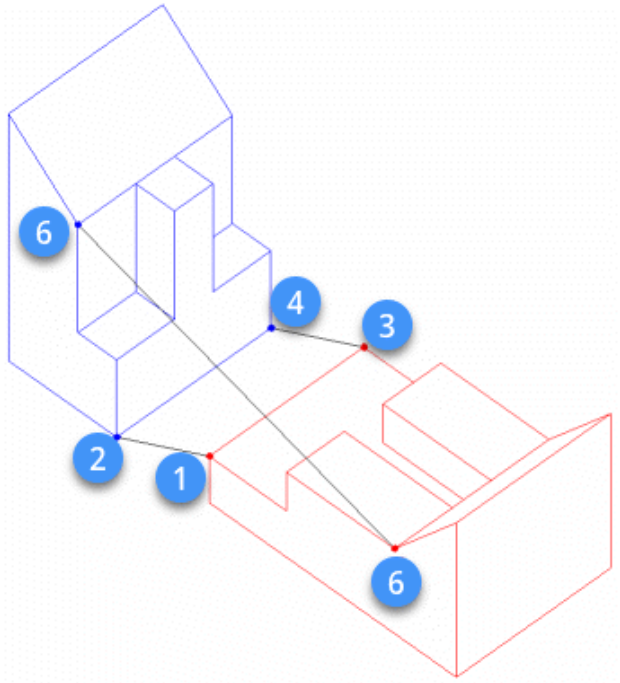
4 Snap to the first source point in the selection set (1).

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:

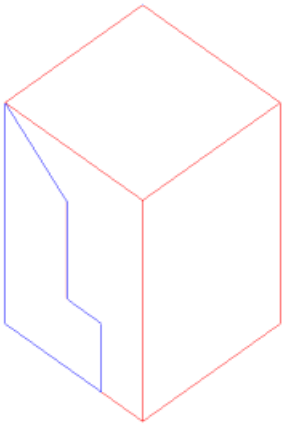
12 Snap to the third source point in the selection set (5).

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.

Uwaga: 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

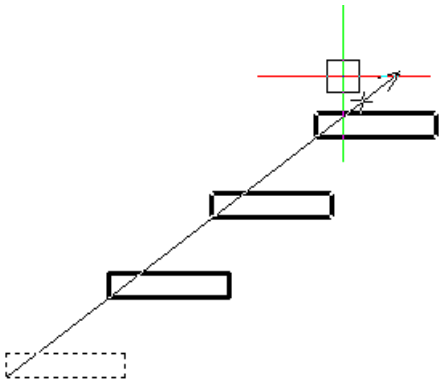
1 Execute steps 1-5 of the general procedure (see above).

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.

Uwaga: 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.

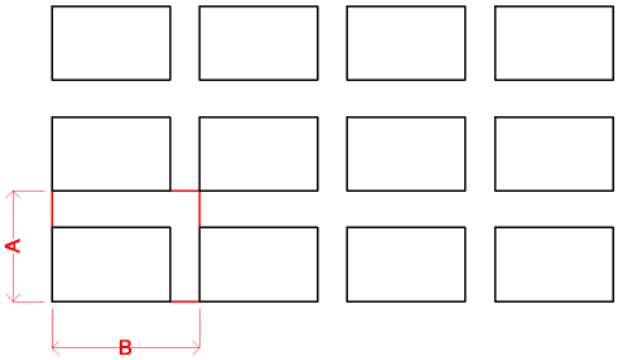
The **Array** dialog box closes temporarily to let you select entities.

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.

Uwaga: 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.

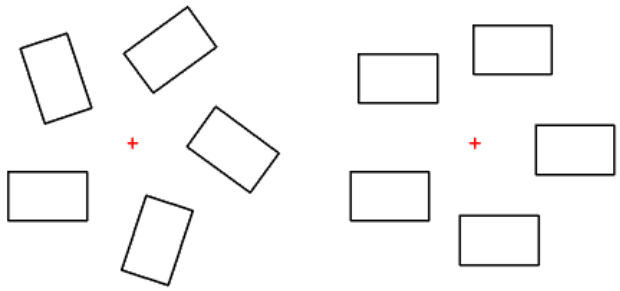
5 The **Array** dialog box reopens.

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.

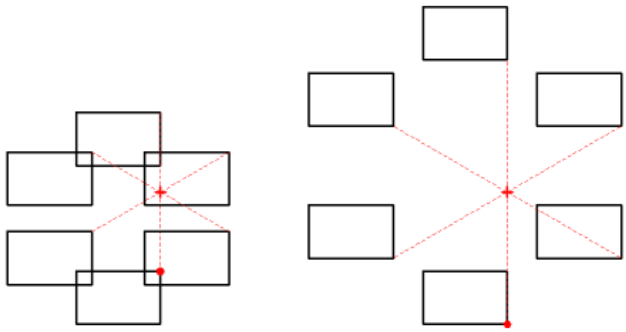
The **Array** dialog box closes temporarily to let you pick a point in the drawing.

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.

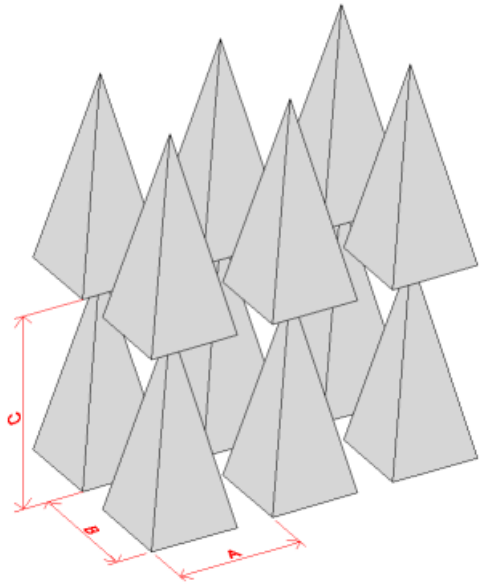
You are prompted: Number of rows in the array <1>:

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.

Uwaga: 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.

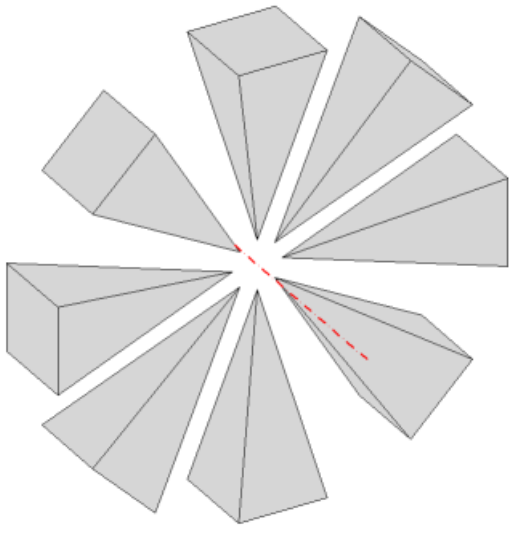
You are prompted: Center point of array:

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.

Uwaga:

- 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.

Uwaga:

- 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.

Uwaga: 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).

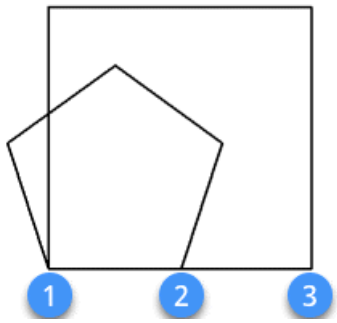
4 **You are prompted**: Second point:

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.

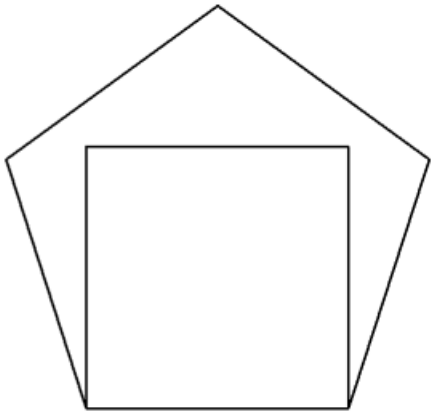
You are prompted: New scale factor or [Points] <2>:



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 (?)]:

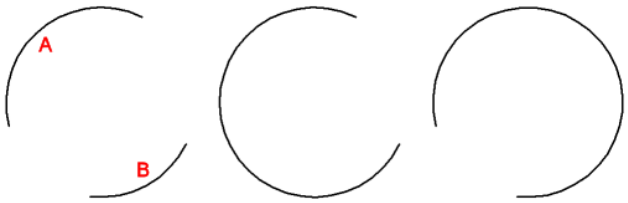
4 Select the second arc.

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.

Uwaga: 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.

Uwaga: 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

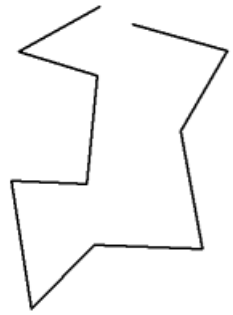
1 Launch the PEDIT command.

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

Uwaga:

- 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.

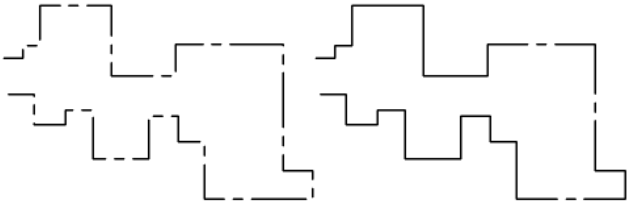
3 Click the arrow keys at the right hand side of the vertex property field to set the X marker at the start vertex of the segment you want to change the width of.

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.

Uwaga: 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>:

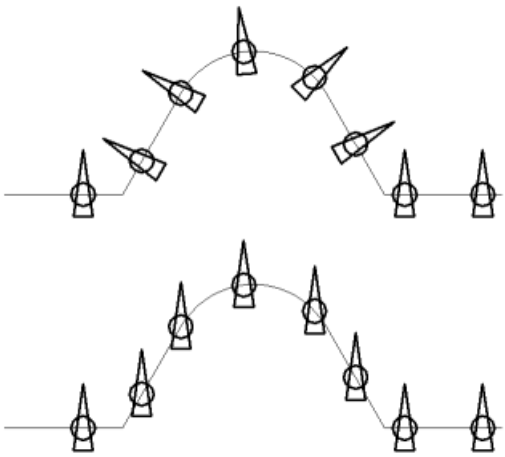
8 Choose whether or not to align the blocks.

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.

Uwaga: 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.

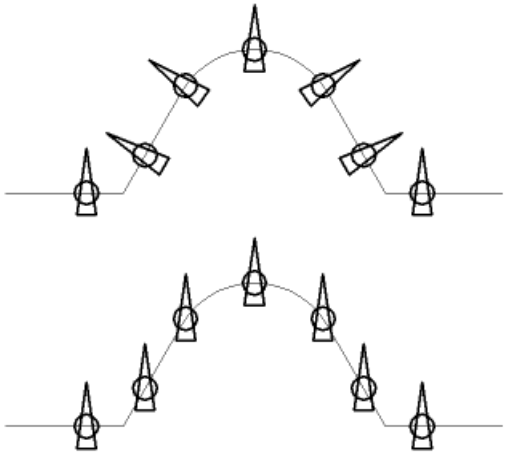
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

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. Printing and plotting

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.

Uwaga: All print settings and options are saved in the **Model** and each of the **Layouts**.



Uwaga:

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.

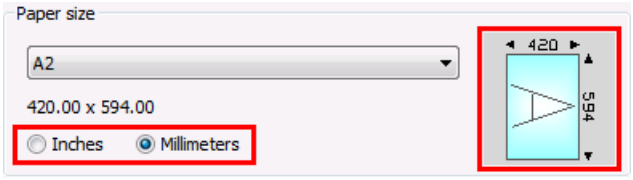
17.1.2 Choosing the paper size

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.

Uwaga: 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

Uwaga: 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. ***Uwaga:*** 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**.

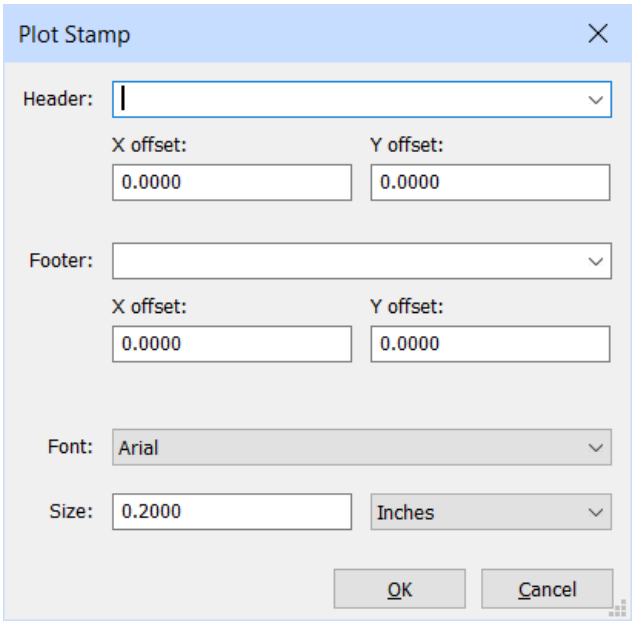
Click the list button, then select an option to plot using a different visual style.

Uwaga: 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.

Uwaga:

- 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.

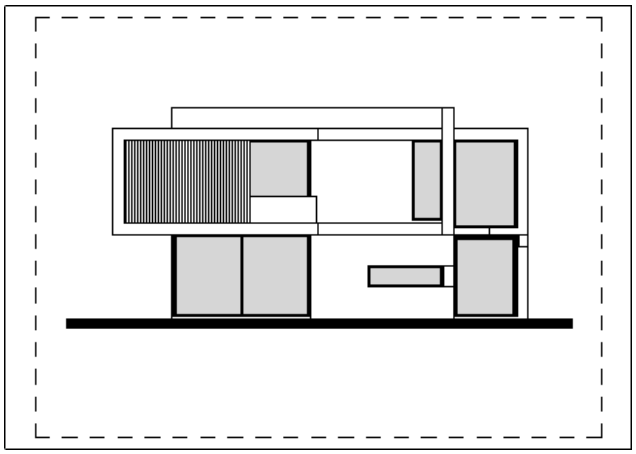
10 To create a custom paper size, do the following:

- 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.

Uwaga: 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.

Uwaga: 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.

Uwaga:

- 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. ***Uwaga:*** 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.

Uwaga: 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.

Uwaga: 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. Rendering

18.1 Using TwinMotion

Uwaga: 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:

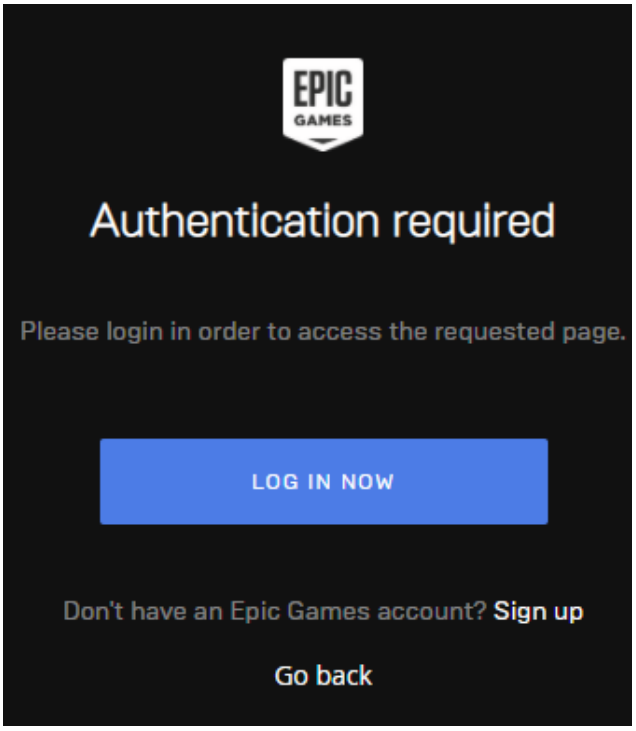
- Install the **Epics Games Launcher**.
- Download and install **Twinmotion**.
- Export your model to a ***.udatasmith** file.

18.1.1 Installing Twinmotion

From the Twinmotion Help pages:

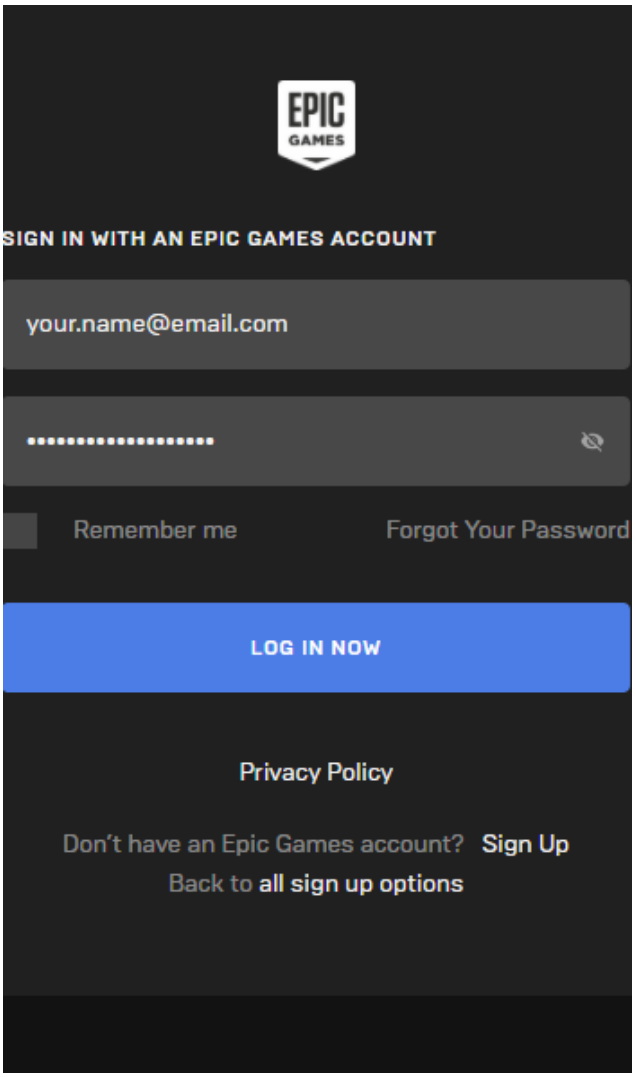
1 Go to https://www.unrealengine.com/en-US/twinmotion.

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.

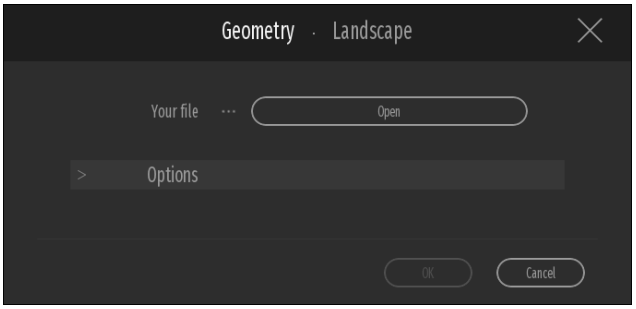
The exported model can now be imported in Twinmotion.

18.1.3 Importing your model in Twinmotion

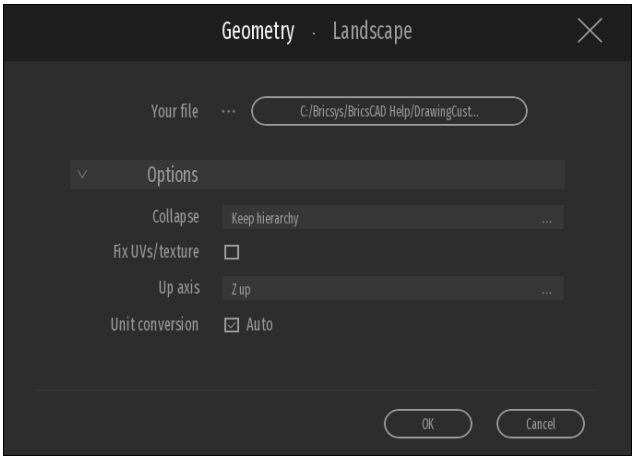
1 Launch Twinmotion.

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

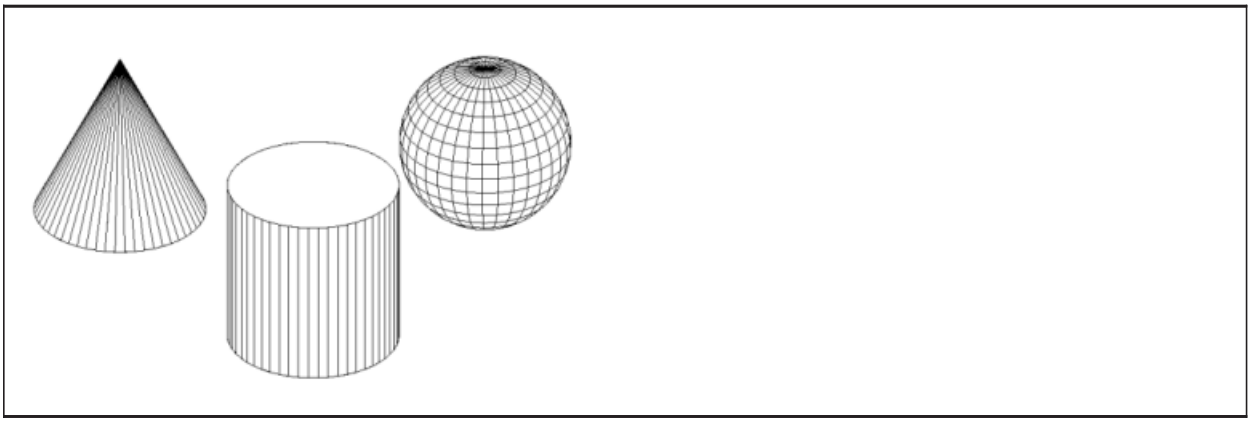
Rendering tools are available in:

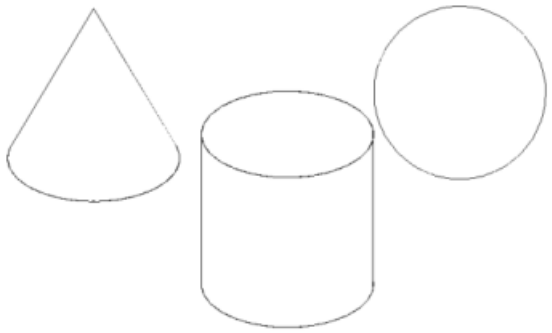
- The **Rendering** toolbar.
- In the **View** > **Rendering** menu.
- The **View** and **Visualize** ribbon tabs.
- In the **General** tab of the **No Selection** Quad.

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.

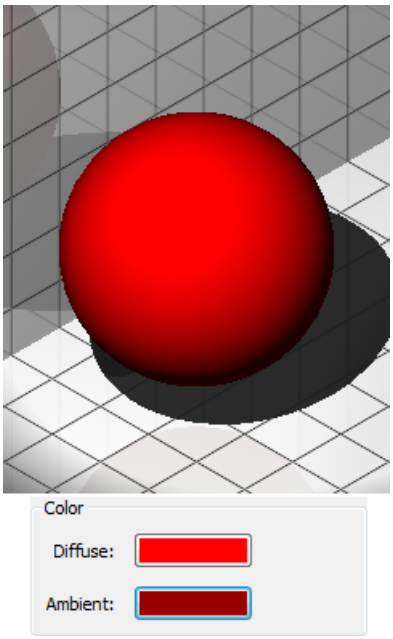
The MATBROWSERCLOSE command closes the **Render materials** panel.

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.

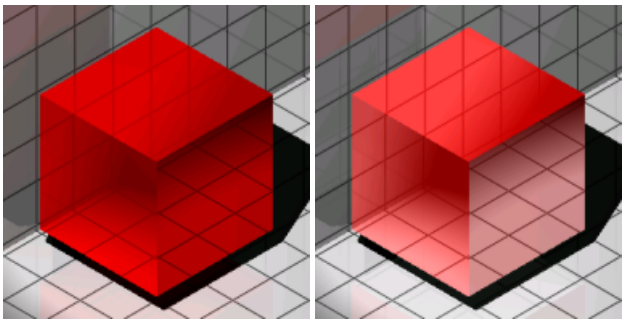
Material definitions are saved in the drawing. Each drawing contains the **Global** material, which you cannot to delete or rename. You can edit the properties of the **Global** material though.

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.

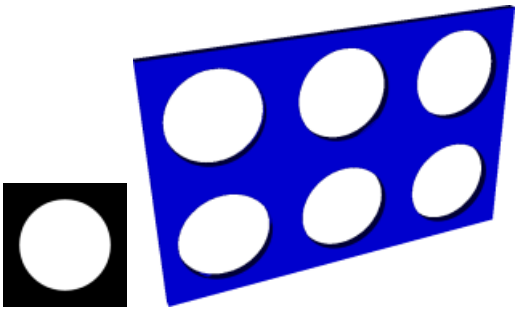


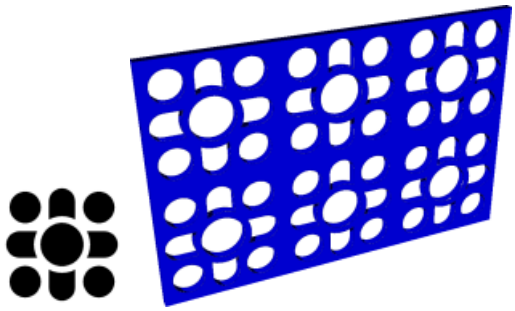
- **Highlights**: highly reflective highlights in the color of the material.



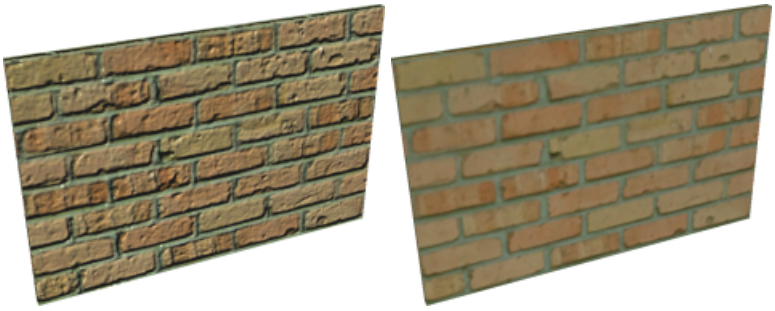
Metallic Highlights (left) and Non-metallic Highlights (right)

- **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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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

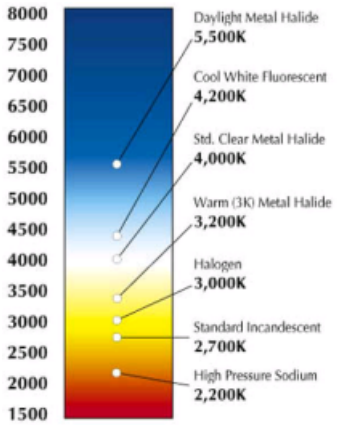
The general properties are common to all lights:

- **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.

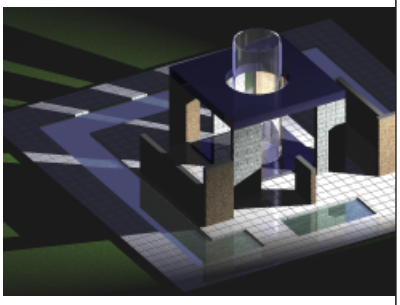
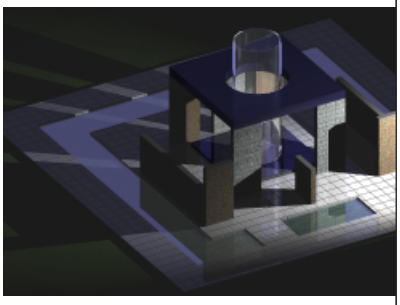
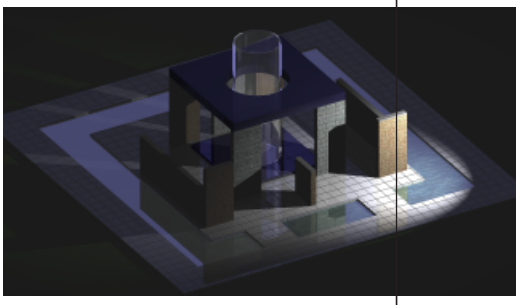
18.6.6 Attenuation properties

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

Uwaga:

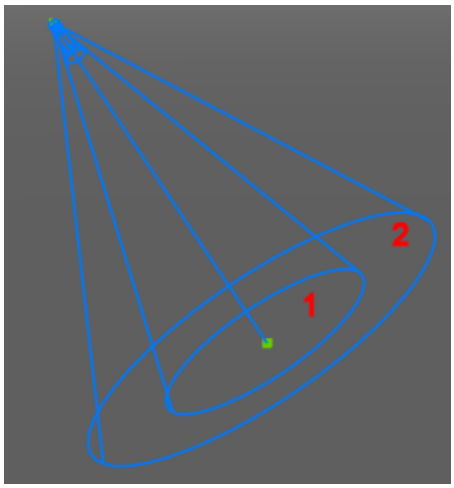
- The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0).
- Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

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.

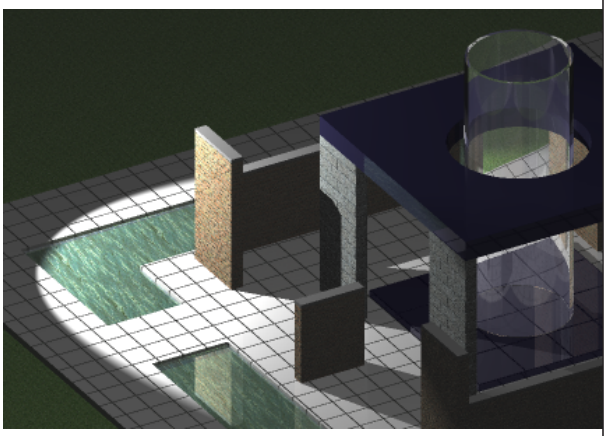
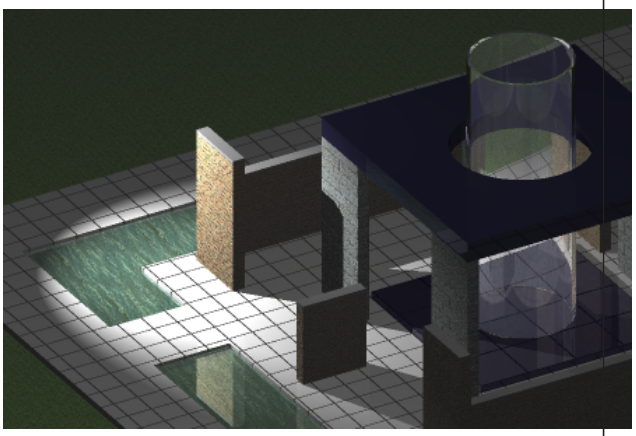
18.6.8 Spotlight hotspot and falloff properties

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.

Uwaga: 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.

Uwaga: 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.

Uwaga:

- 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. Odpowiedź zwrotna użytkownika

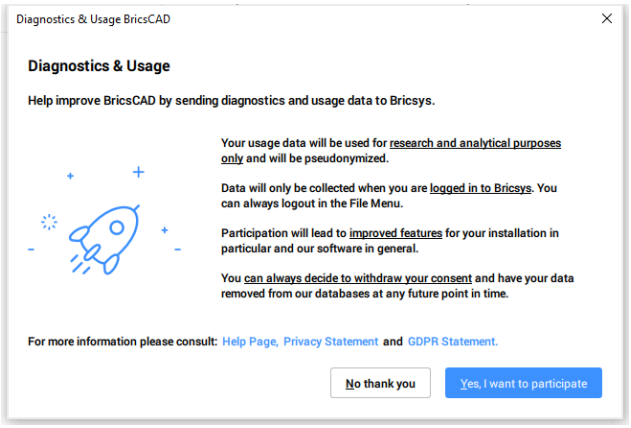
19.1 Diagnostics and usage data collection

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.

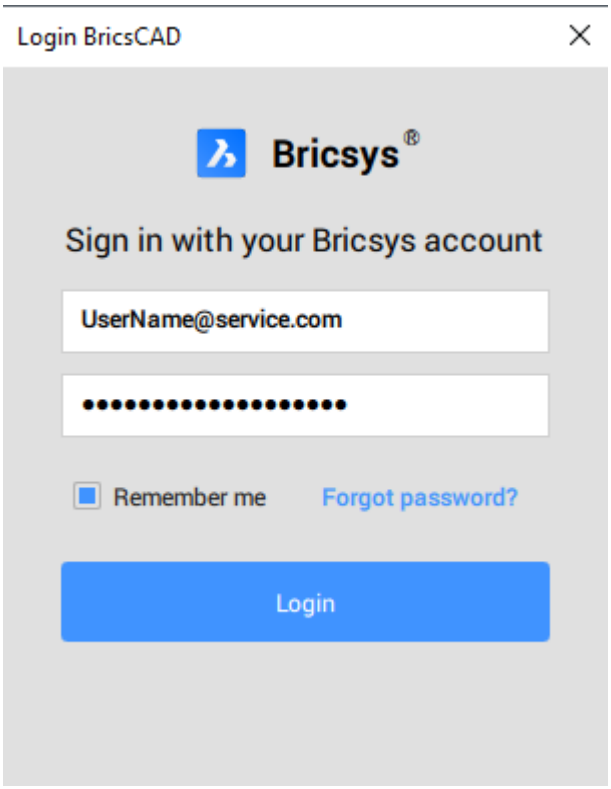
When transferring data:

- 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 Co nowego

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 stronie 959

Building data na stronie 966

Project collaboration na stronie 975

Design documentation na stronie 976

Point clouds na stronie 985

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 stronie 31

User interface na stronie 32

Productivity na stronie 35

Drafting na stronie 49

Modeling na stronie 51

Civil tools na stronie 60

Performance and compatibility na stronie 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:

Przegląd na stronie 1529

Menedżer BOM na stronie 1530

Biblioteka części na stronie 1533

Parametry i ograniczenia na stronie 1534

Części standardowe na stronie 1534

Arkusz Blachy na stronie 1534

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

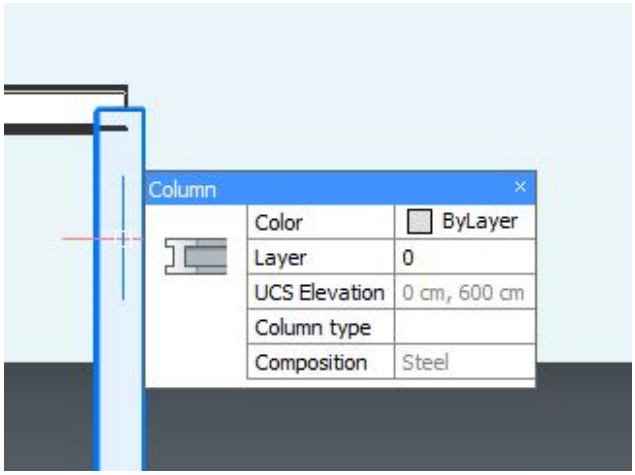
Diagnostics and Usage Data Collection program. The data collection process enables you to share anonymous diagnostic and usage data with Bricsys to help improve BricsCAD. You can withdraw your consent and have your data removed from our databases at any time.

20.1.2 Modeling techniques

20.1.2.1 BIM Profiles

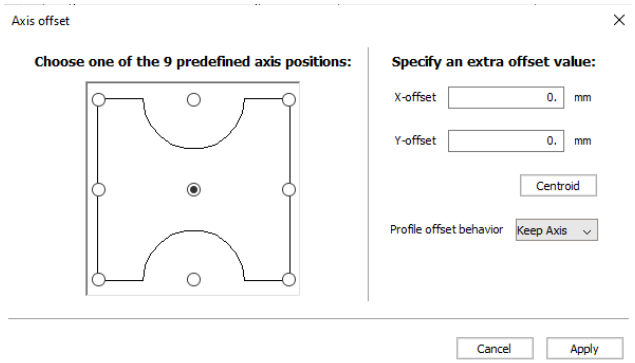
BIMAPPLYPROFILE / BIMLINEARSOLID

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

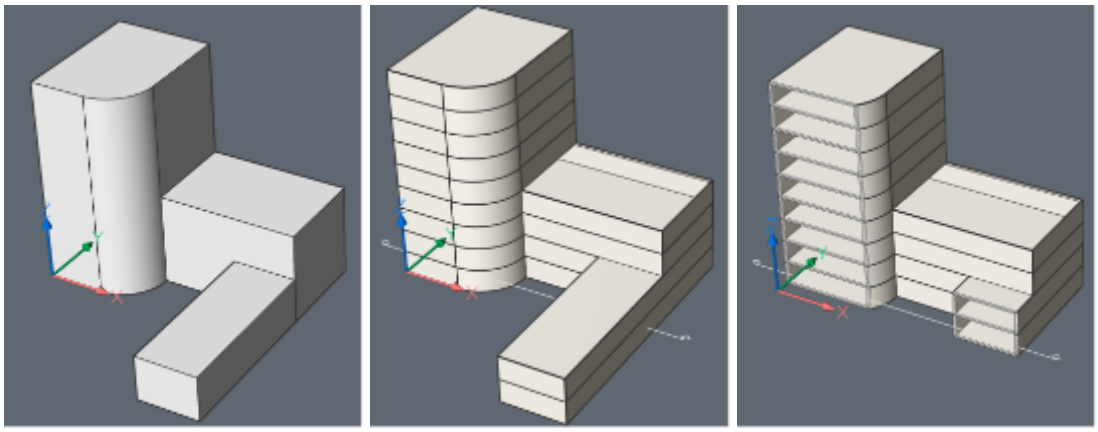
The BIMPROFILESTANDARDS setting is no longer case sensitive. In case there are no profiles matching the value of the BIMPROFILESTANDARDS setting, a warning message displays.

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.



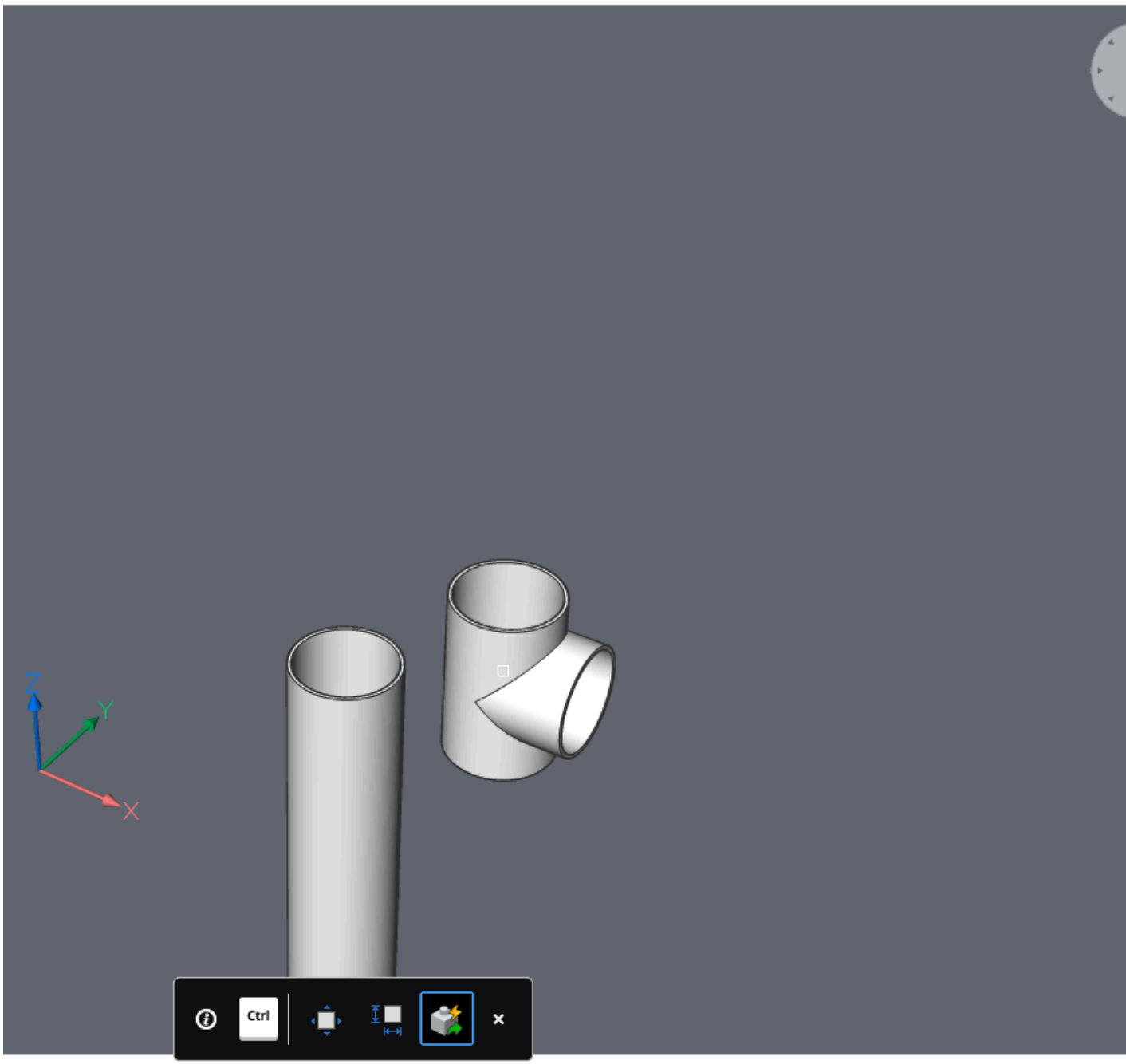
20.1.2.3 Insert BIM Component

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.

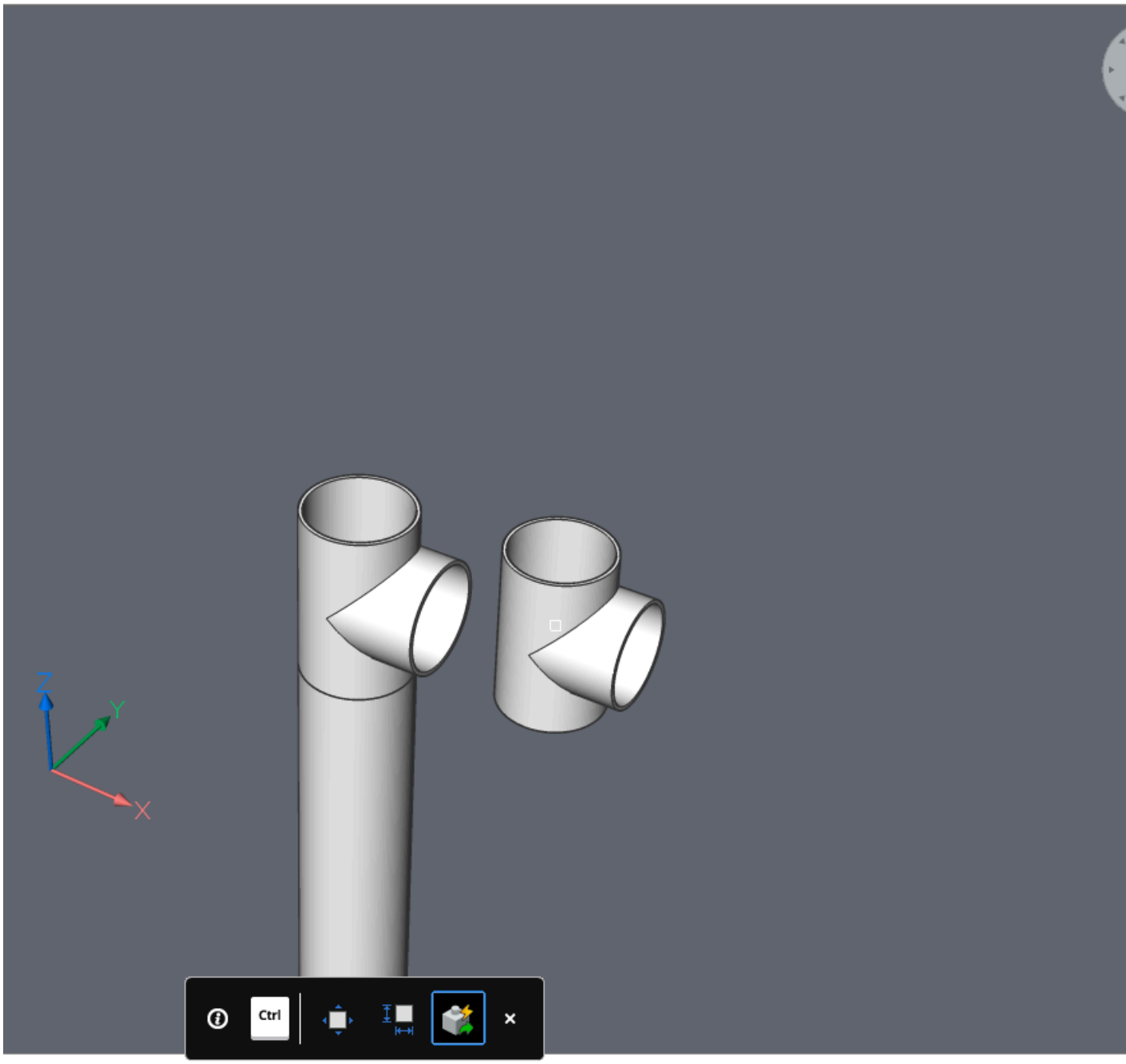
You can select SMART insert by clicking the option in the Command line, or toggle CTRL to the rightmost icon when prompted in the HotKey Assistant.

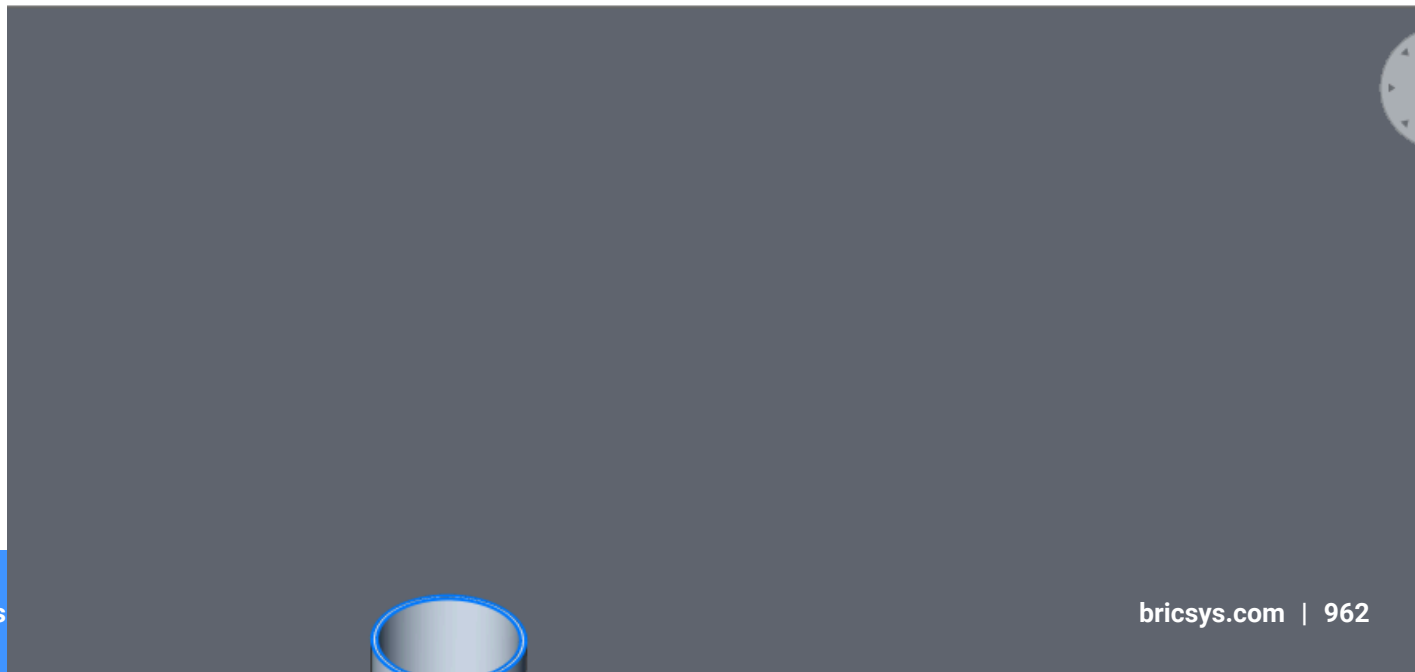


By detecting and creating 3D constraints between the relevant connection points, SMART insert resizes and attaches your desired component onto the current component/s with assembly constraints and common parameters to form a bigger part of a parametric assembly. This is particularly useful in modeling parametric MEP piping systems.


Z
Y
X
Ctrl

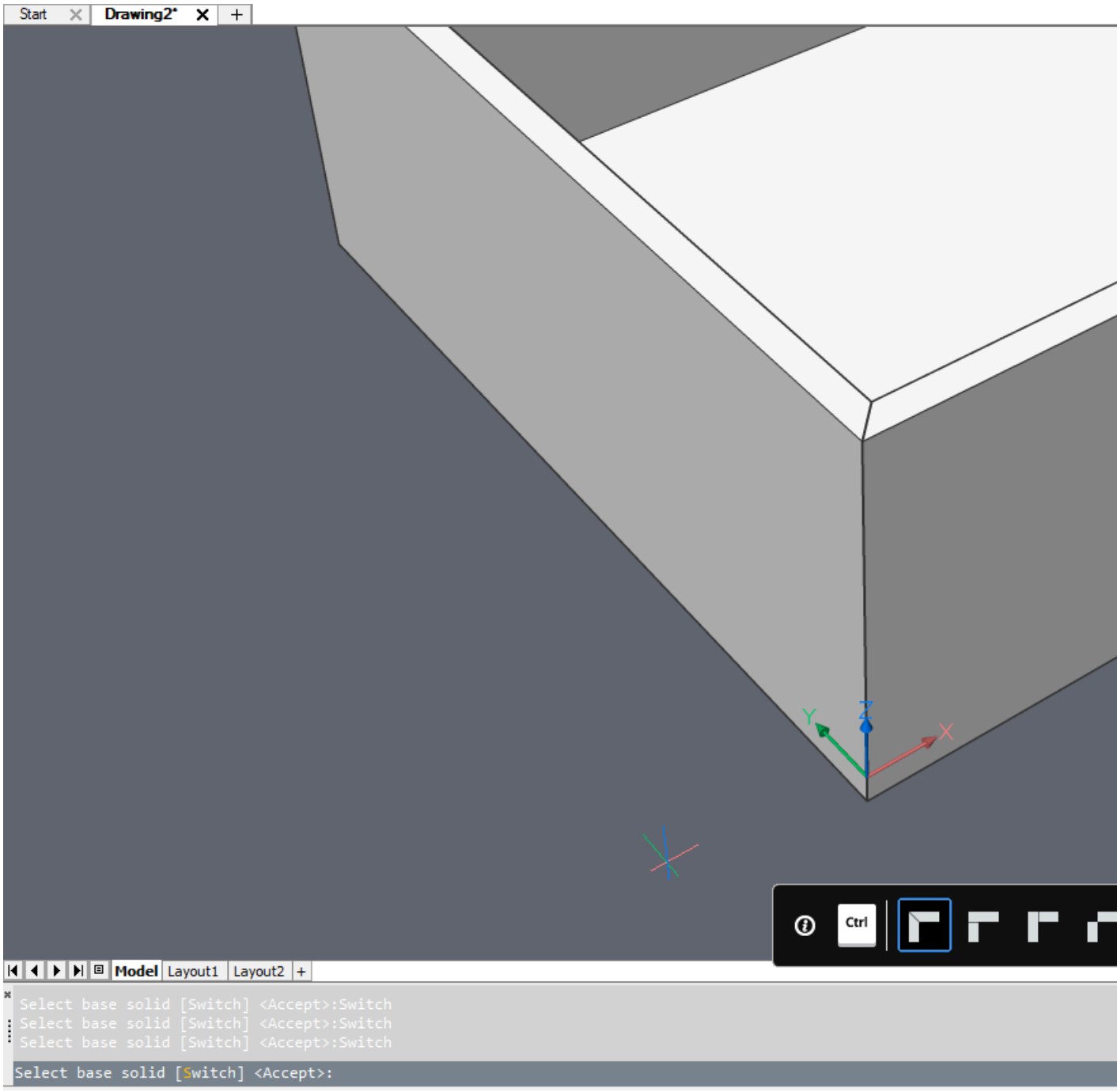


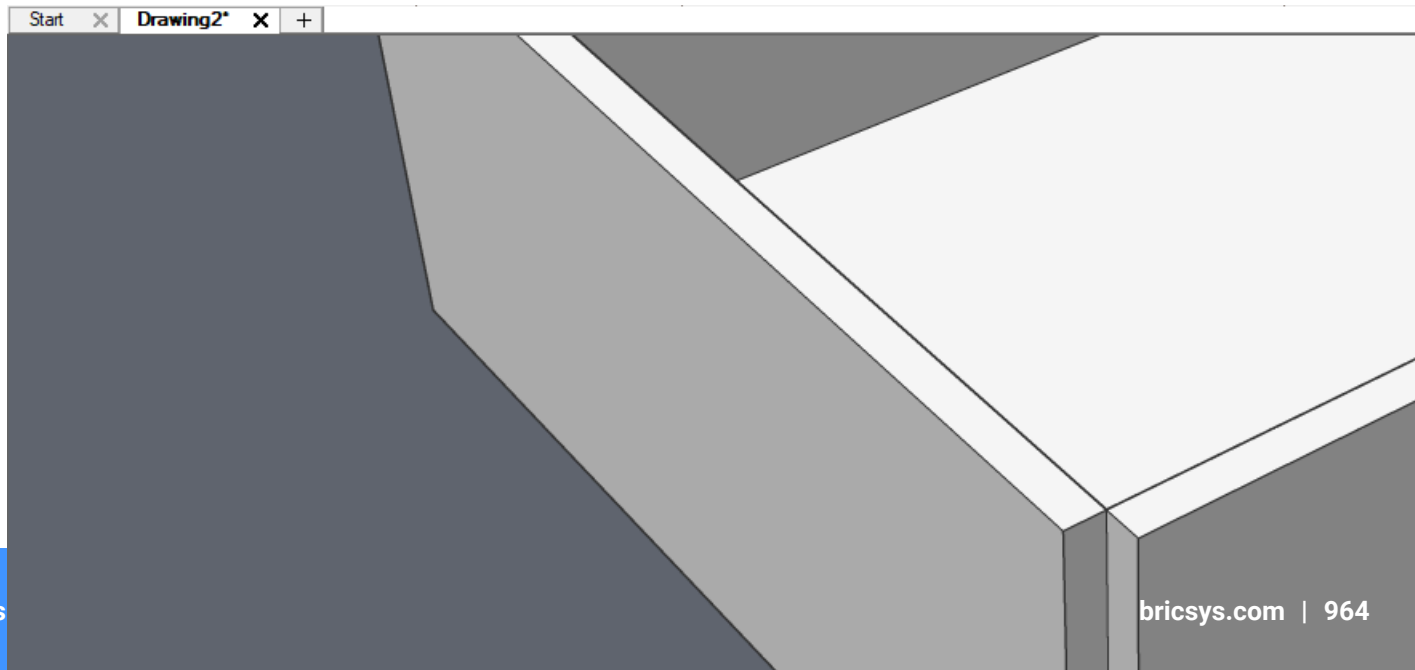

Z
Y
X
Ctrl

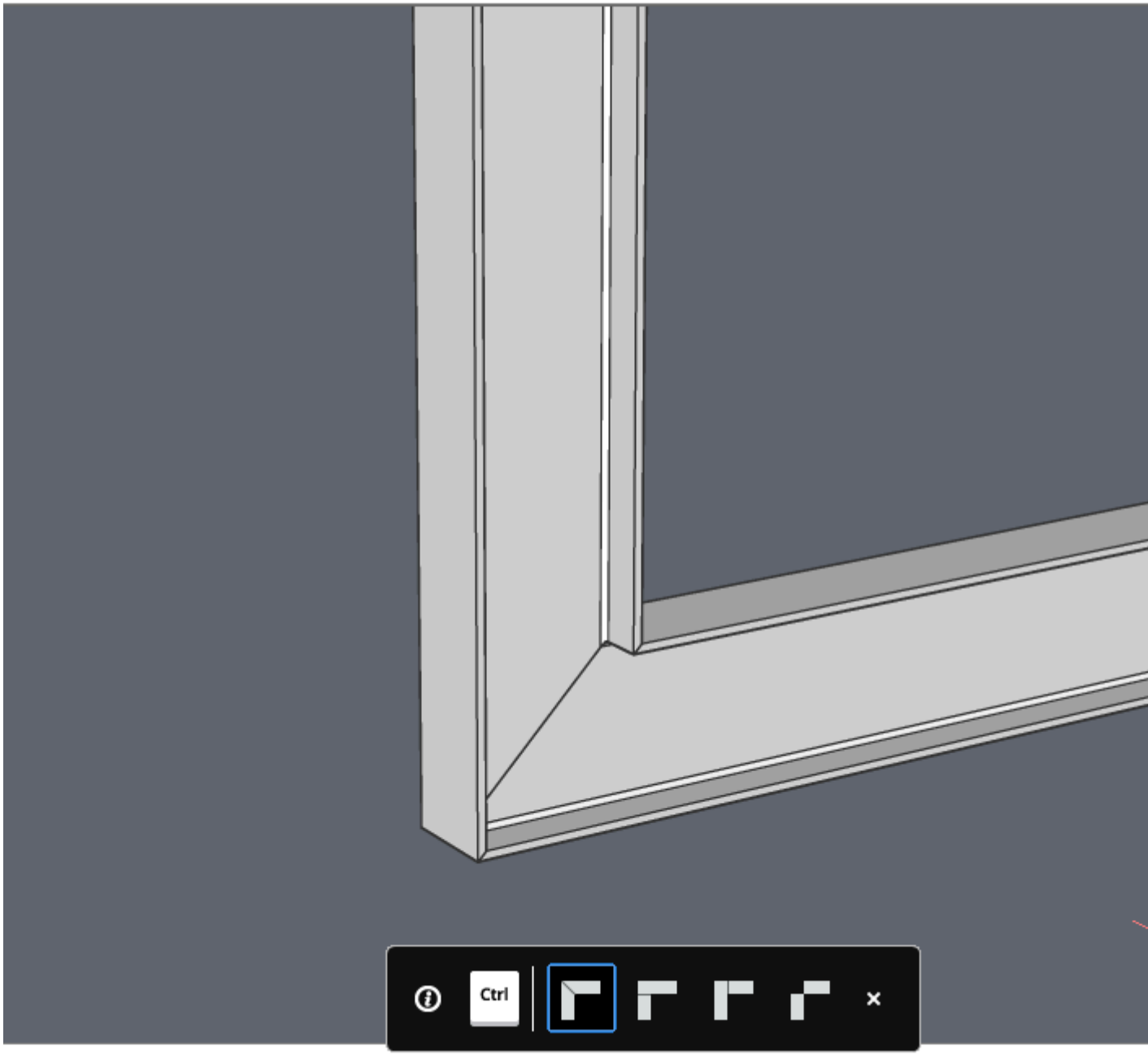


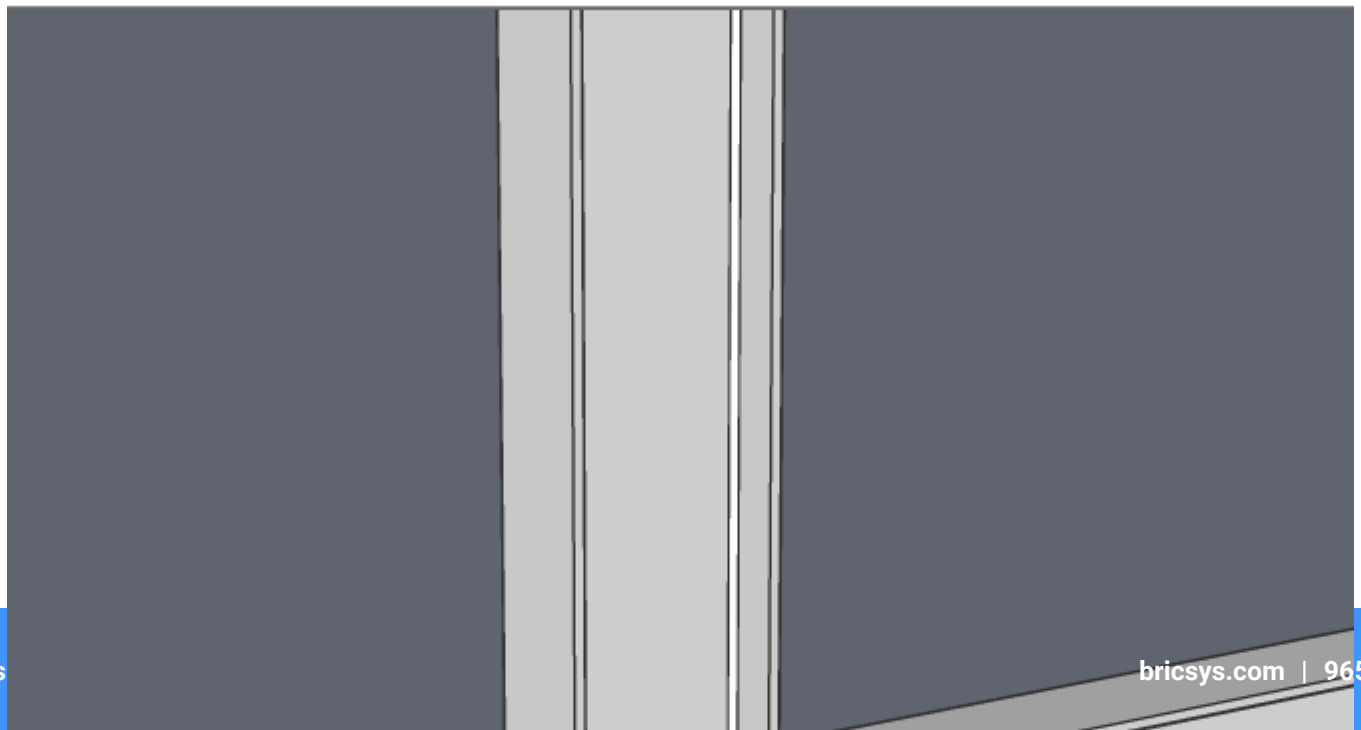
20.1.2.4 Disconnect

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

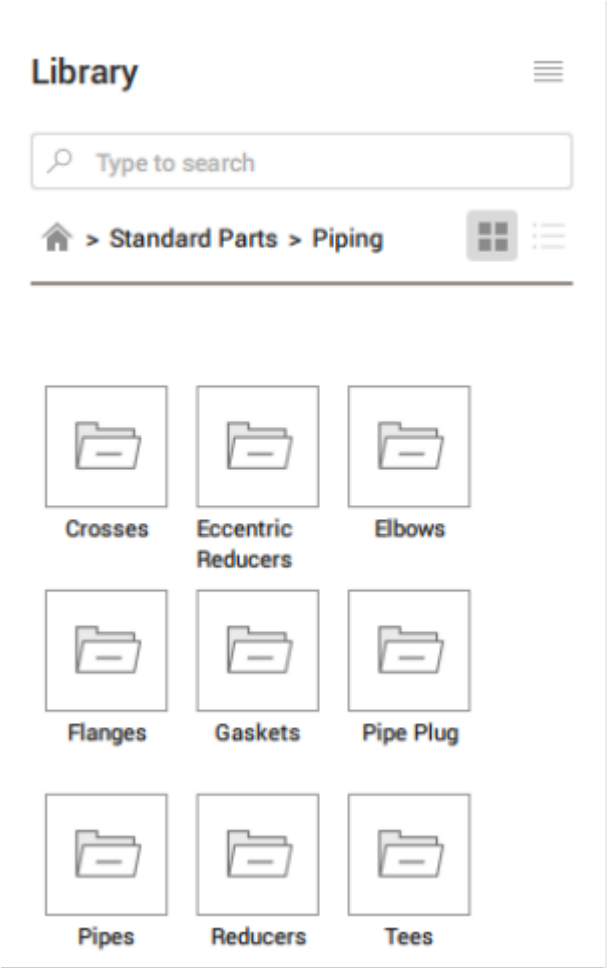


20.1.2.5 Quickdraw

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

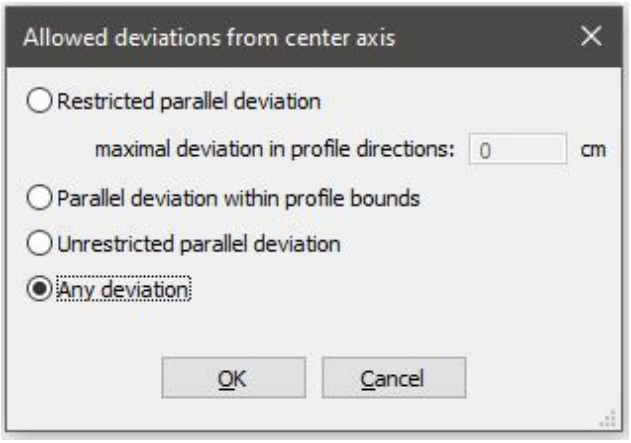
Piping parts were added to the standard parts category of the Library panel.



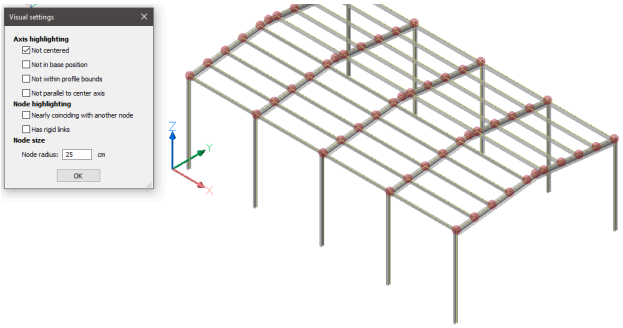
20.1.3 Building data

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.

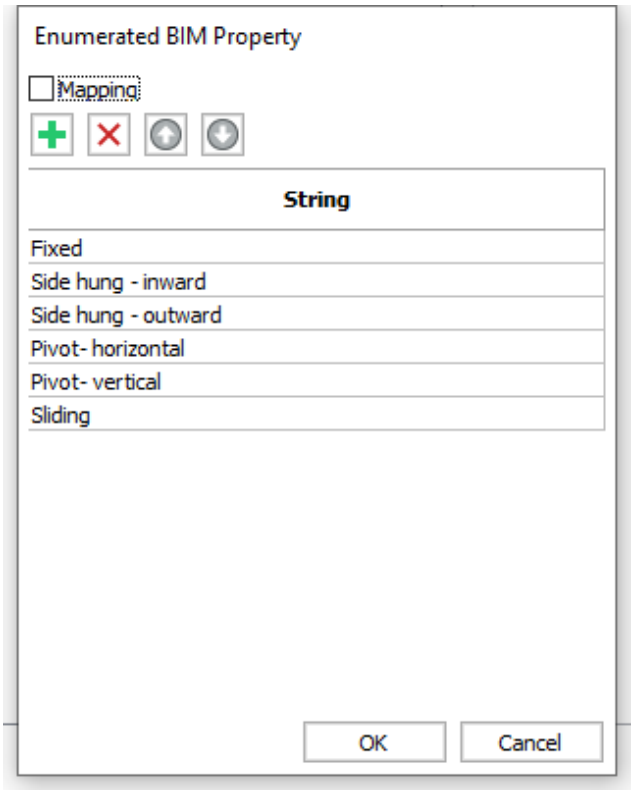


LOD settings control whether ply solids are shown or not. If ply solids are not visible and render composition materials is on, only major faces of the multi ply solids will get render materials.

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.



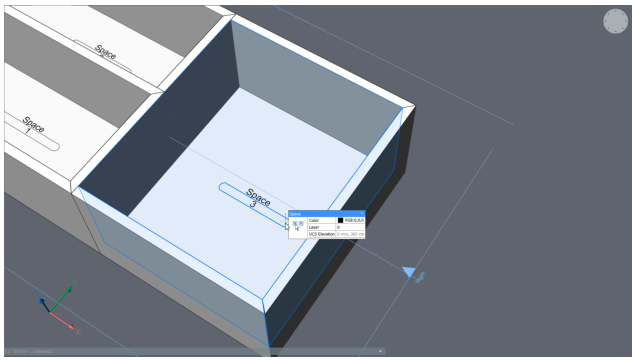
20.1.3.5 Spatial Elements

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

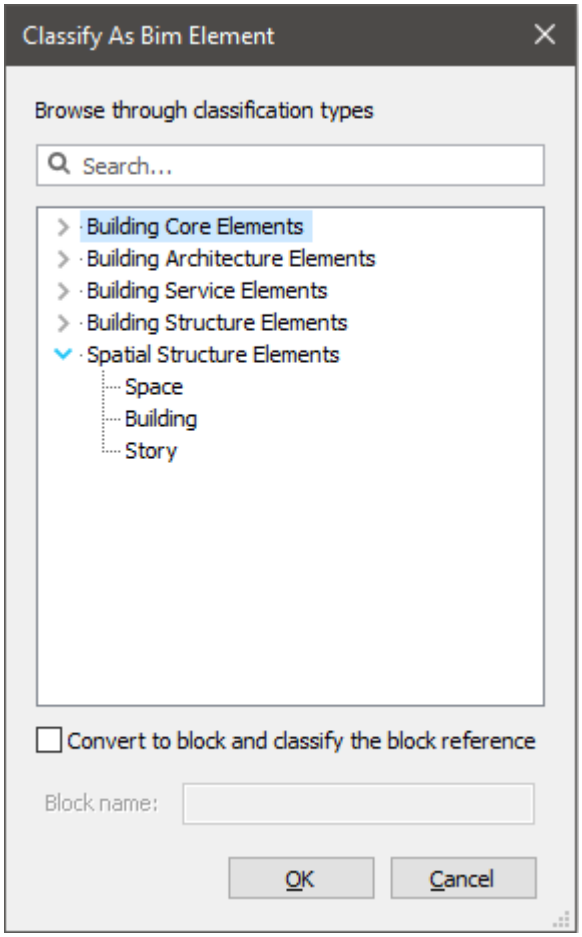
Space icons

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.

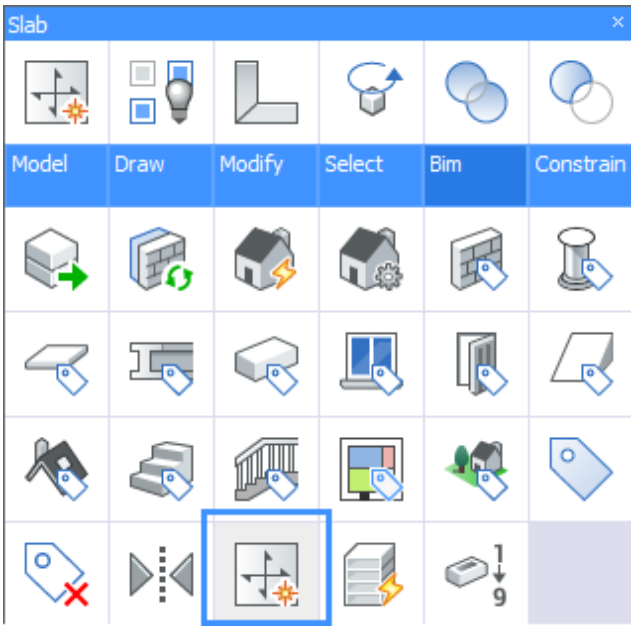


DefaultSpaceHeight

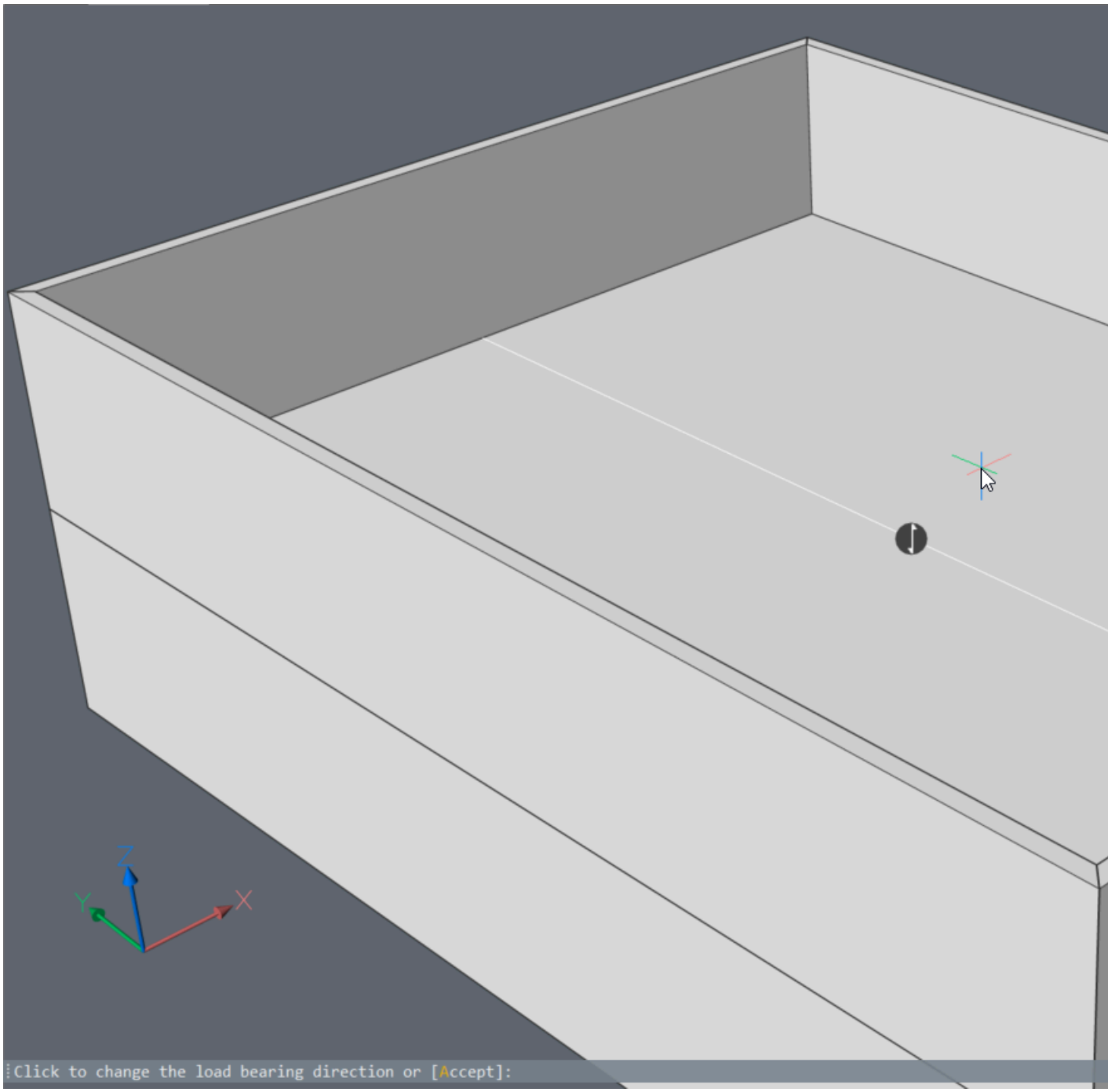
The ROOMHEIGHT system variable is replaced by the DefaultSpaceHeight user preference, the setting applies to Spaces.

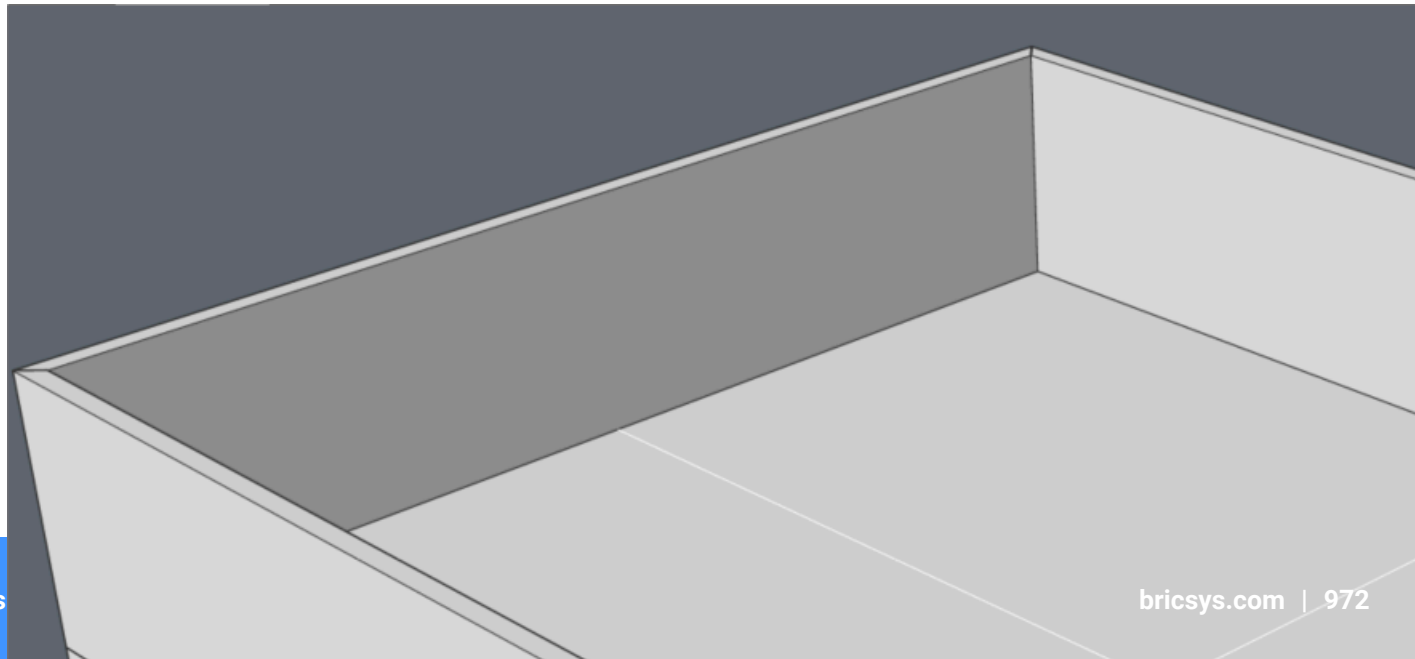
20.1.3.6 BIMSETLOADBEARINGDIRECTION

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

Uwaga: 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.

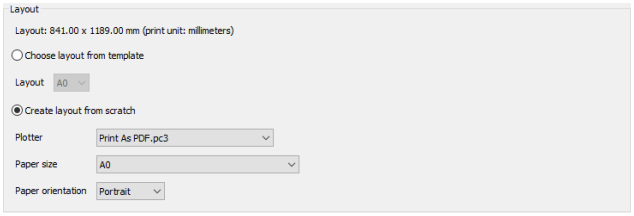
The Python Programming Language, together with its standard libraries, is embedded in BricsCAD BIM so there is no need for you to install them separately, unless you have custom packages and libraries which you wish to use in your scripts.

20.1.3.8 Project Browser

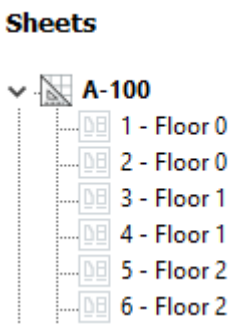
BIM project data are now stored in an external project **.bsyslib**, instead of the previously used **.projectroot**. When opening a pre-V21 BIM project, you can convert the **.projectroot** to a V21 project **.bsyslib**. To simplify the opening of a BIM project at a new location, paths to **.bsyslib** 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.



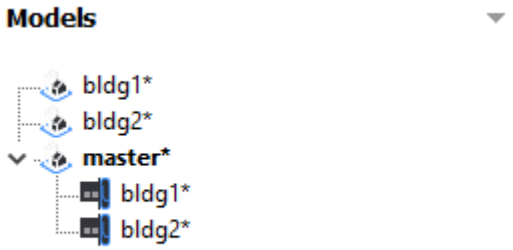
The Project Browser displays the Sheet Number (if it exists). The Sheet Number is assigned by the **Create Sheets** procedure. Sheet View Numbers are assigned starting from 1 for all section results created by the BIMSECTIONUPDATE command.



Models

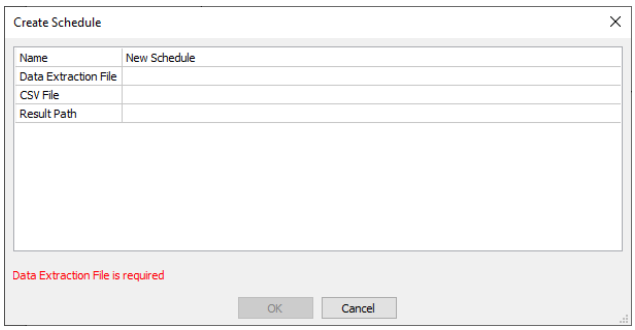
Master drawings and external references are listed in a tree-structured manner in the Models tab of the Project Browser:

A balloon is available to fix a project libraries' connection to a drawing sheet.

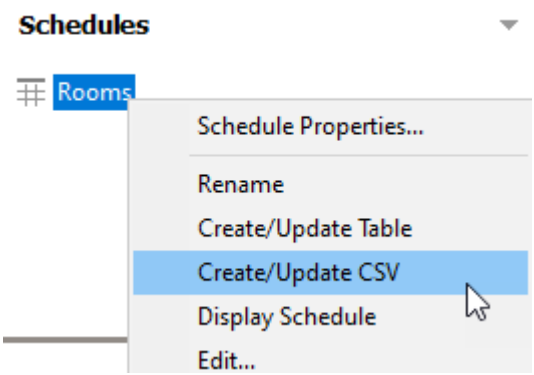


Schedules

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

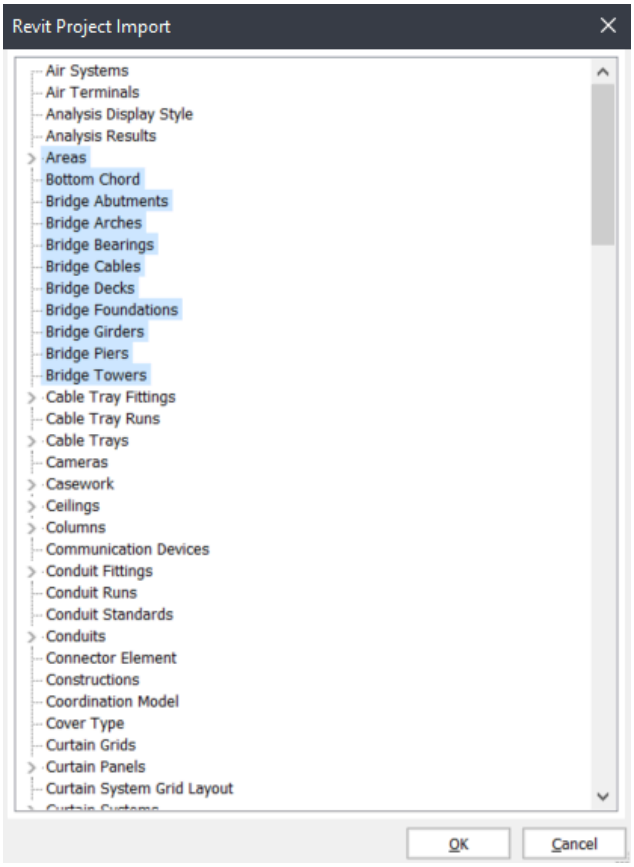
In this case, walls will be imported as slabs and slabs will be imported as walls. No other entities will be imported, as the asterisk means that only listed entity types will be taken into account, all unlisted entity types will be skipped.

The IfcCreateUniqueGuid user preference has been added to enable/disable the creation of unique GUIDs for nested elements inside classified blocks and Xrefs.

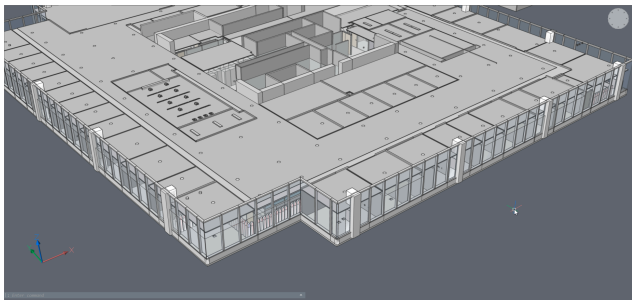
The flow connection point is exported to IFC as IfcBuildingElementProxy. Only 2D geometry or a block reference with 2D geometry can be classified as a flow connection point.

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

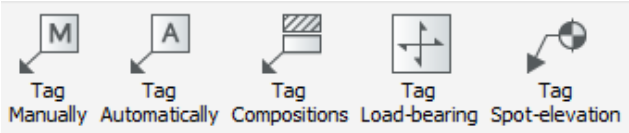
The associative data for 3D solids, needed for section generation and tagging of the section results by the BIMTAG command, are generated automatically in BricsCAD BIM V21 as you model, unless the new GENERATEASSOCATTRS (Generate Associative Attributes) system variable is set to Off.

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.

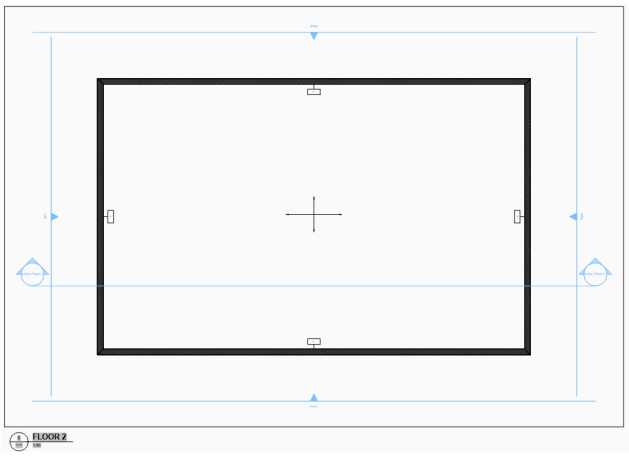
Tag Spot-elevation

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.

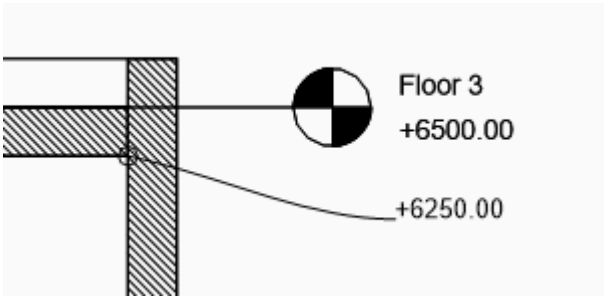
Tags in general

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.



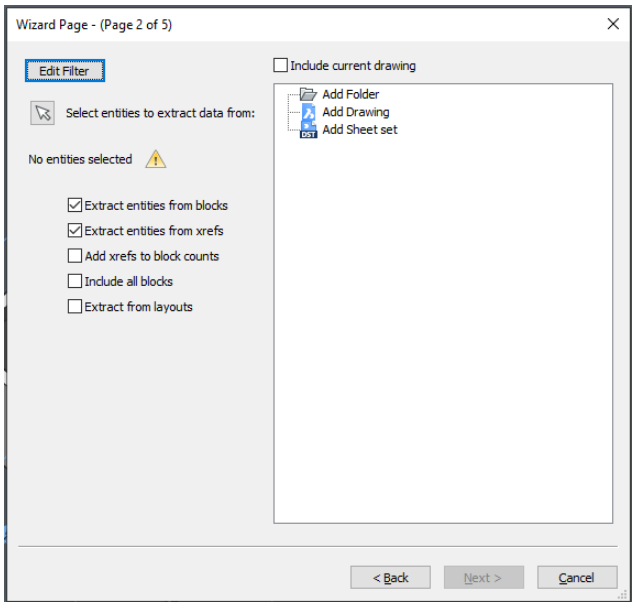
Or you can have spot elevations displaying elevation levels relative to the WCS.



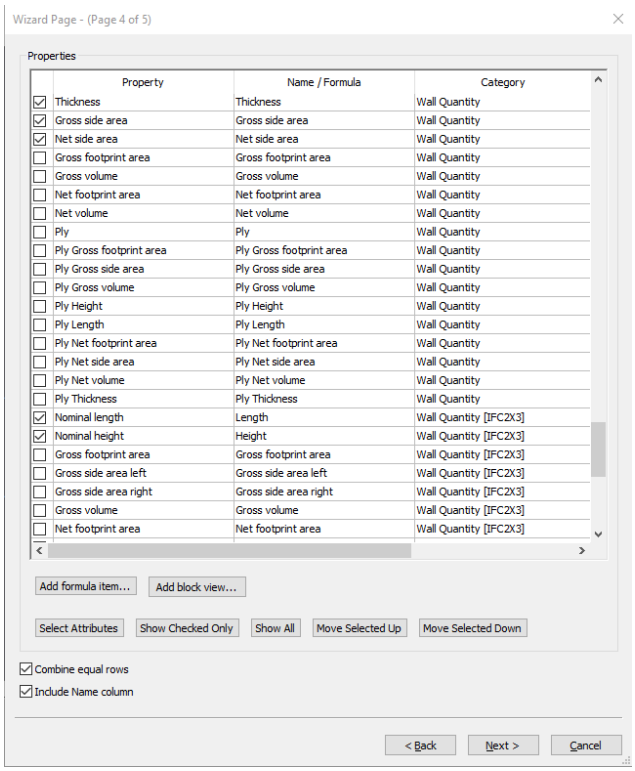
Like all other BIM tags, they are customizable multileader blocks. Therefore, for example, Spot Elevation tags can be further adjusted to suit your graphic and information preferences.

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.



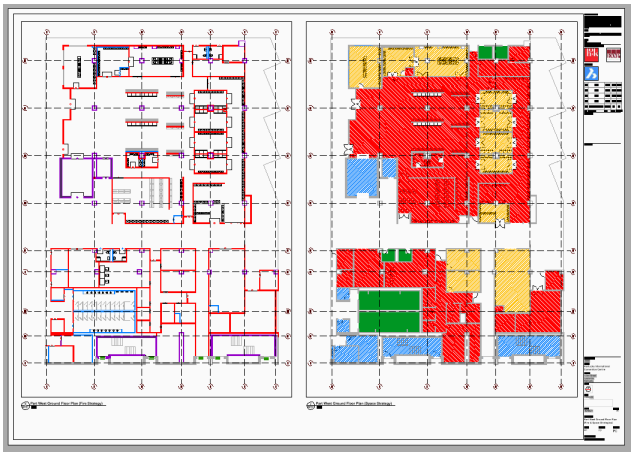
You can now extract and schedule quantity data based on IFC2x3 Element Types.



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.

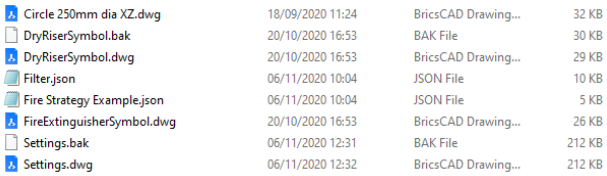
It allows you to create view templates to control and customize the visibility settings of your drawings on layouts. The Drawing Customizations tool in BricsCAD is operated by 3 principles – Entity Customizations, Styles and Filter Rules.



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.

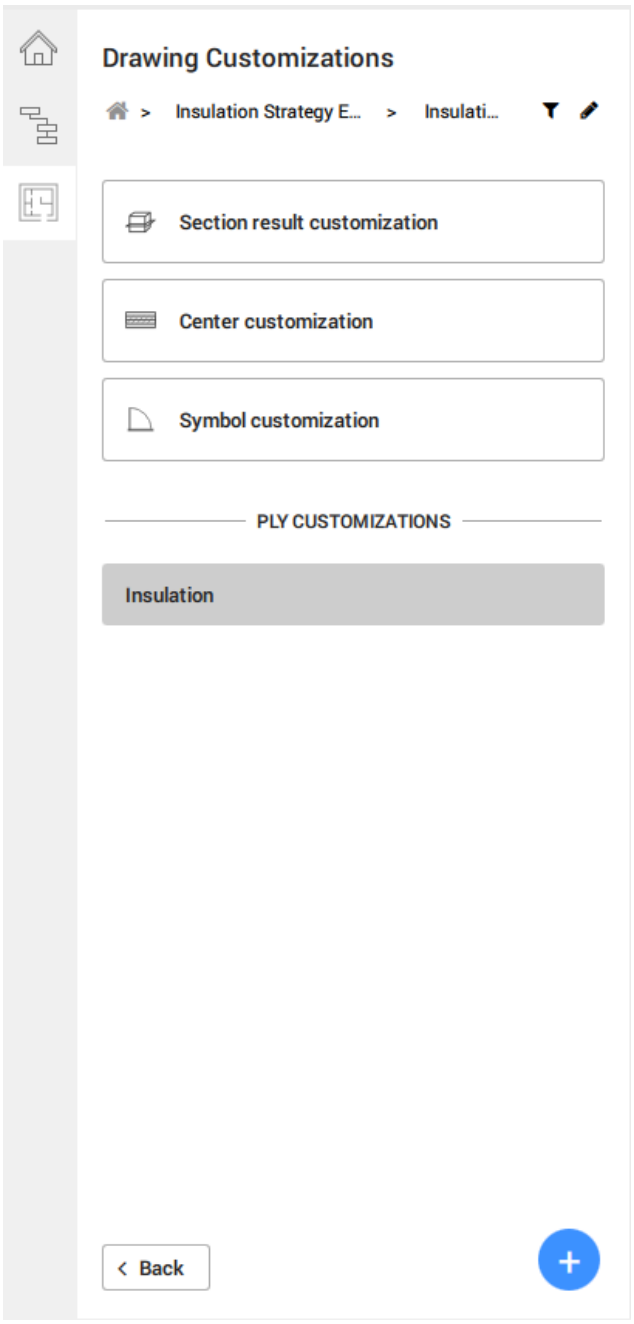
Settings.dwg contains the values created in the Style tab in the form of layers and if you have created any custom properties that you wish to use as a Filter parameter, you have to ensure they are also reflected in the Settings.dwg.



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.

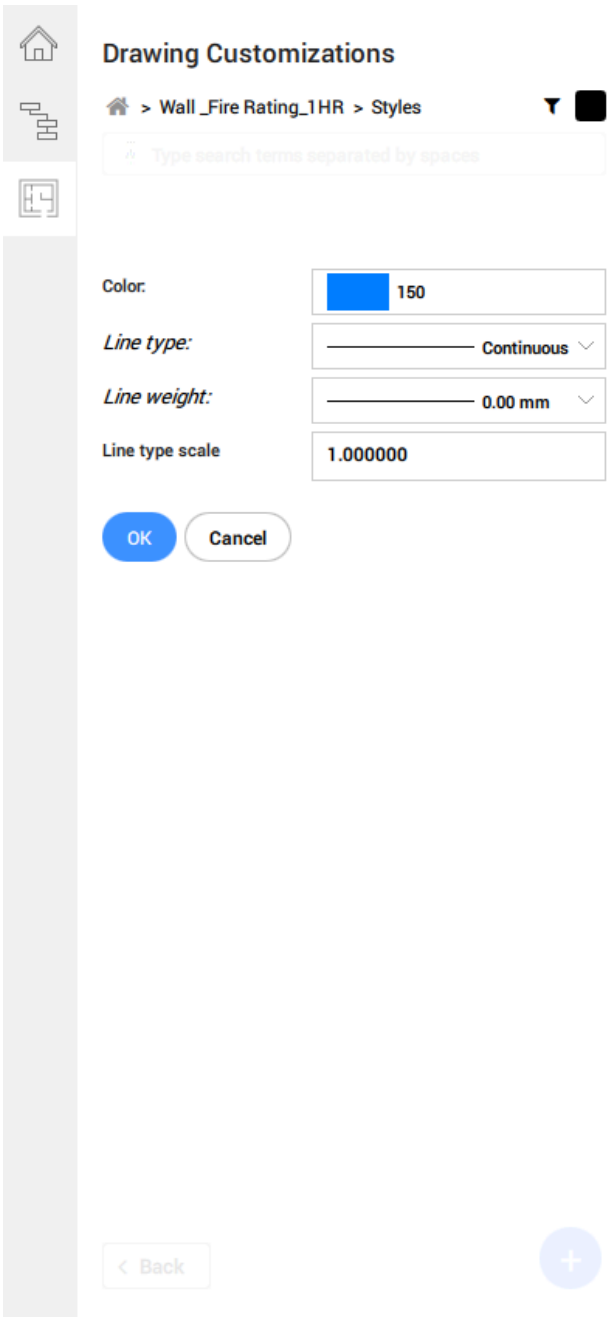
You can further apply your saved Styles to specific layers within your external symbol source, thereby allowing you to retain a general symbol drawing across several customization templates with different output results.

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.

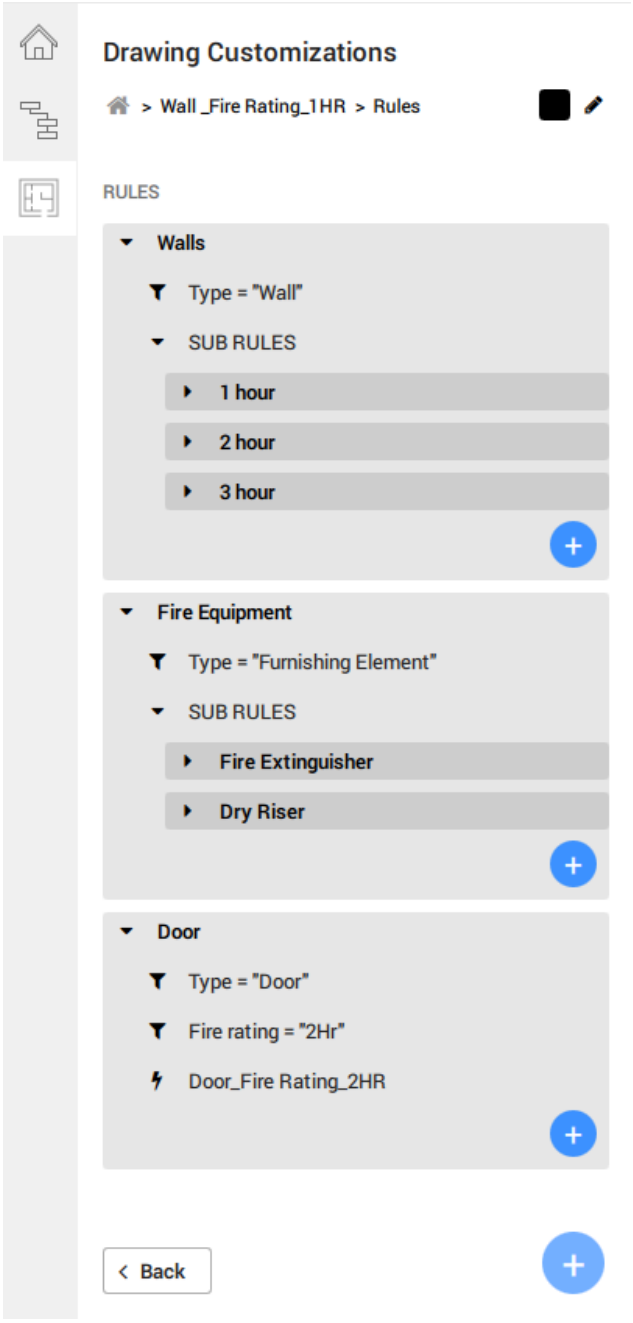
The operation of the Styles tabs is like how Entity Customization tabs can be controlled.

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.

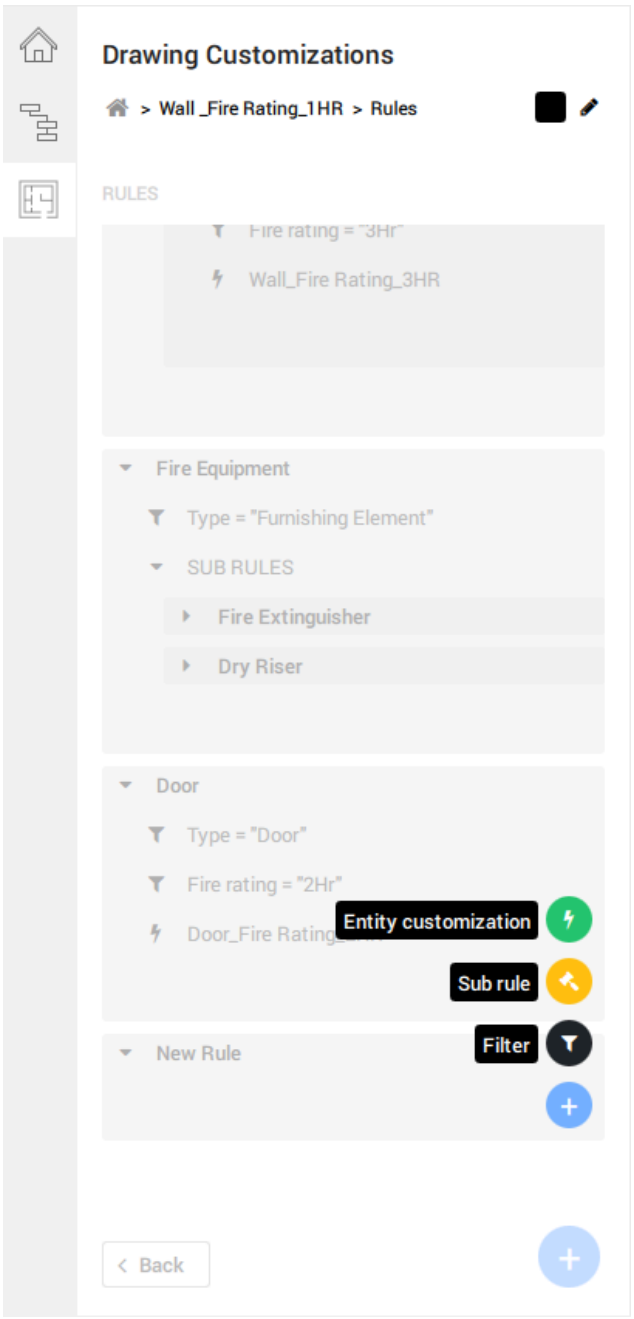


Filter Rules

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.

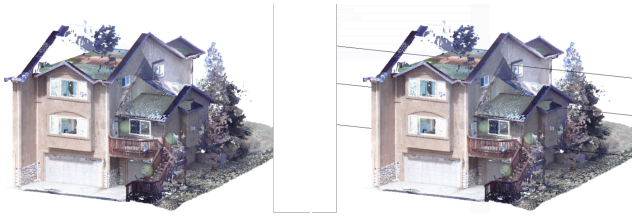
20.1.6 Point clouds

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.

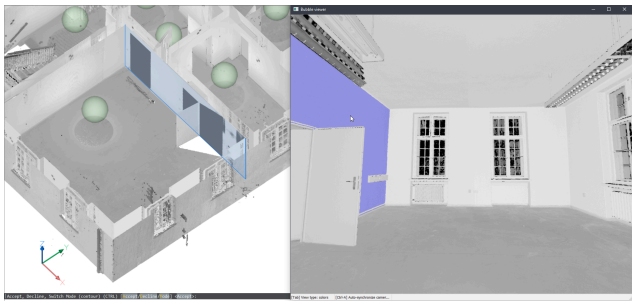


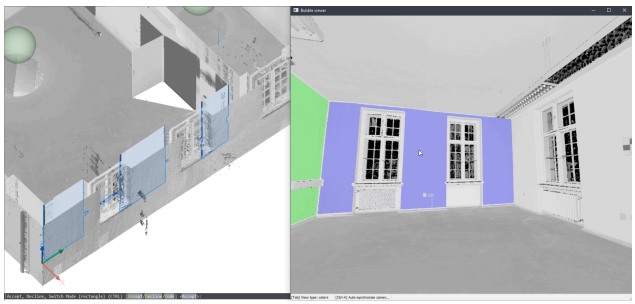
20.1.6.2 Planar Fit

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.

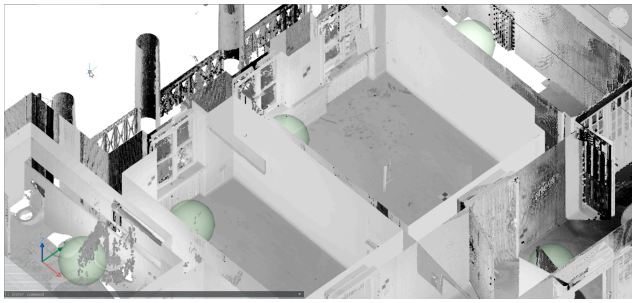


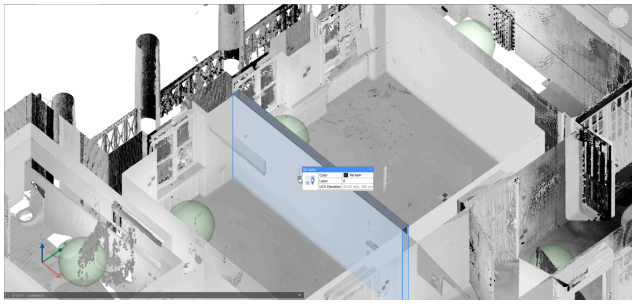


In model space

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:

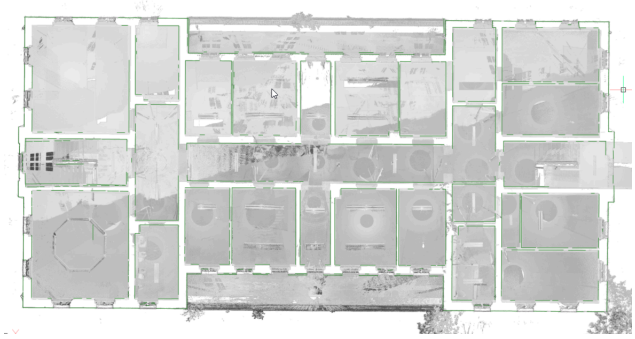
- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.





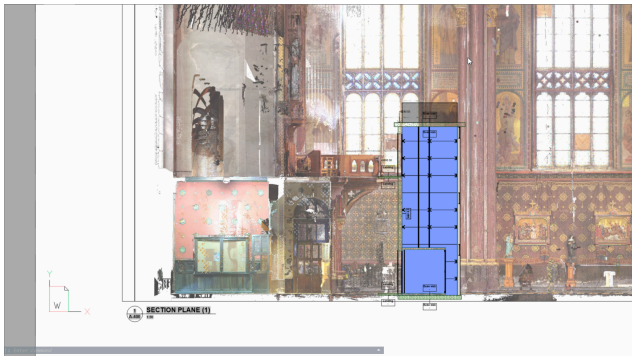
20.1.6.3 Point cloud projection

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 Rozpocznij pracę

20.2.1 Creating a BIM model

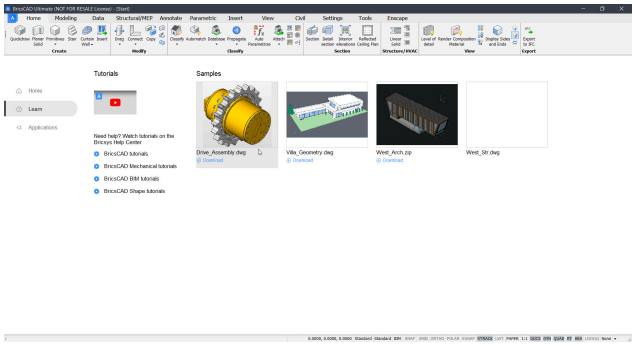
Do the following:

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.

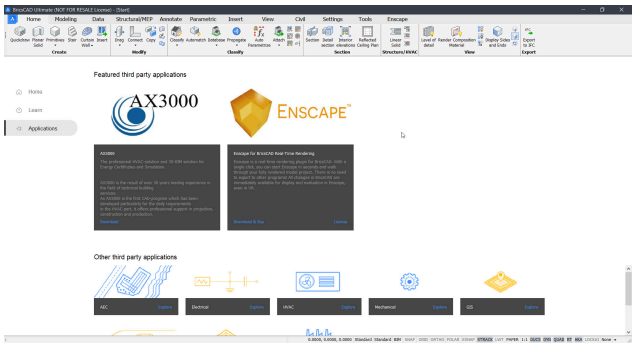
20.2.1.1 Learn



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

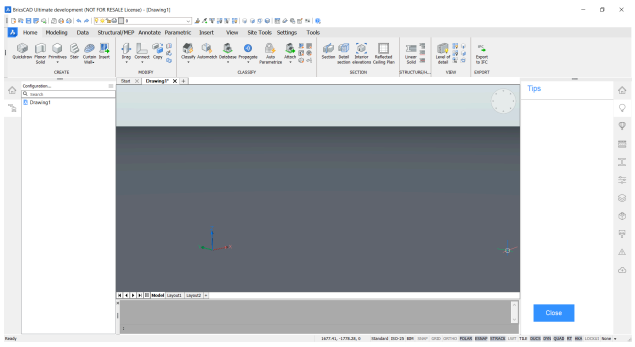
Here you can find some of the third party applications that are available for BricsCAD. You can find more applications on our Applications Store website.



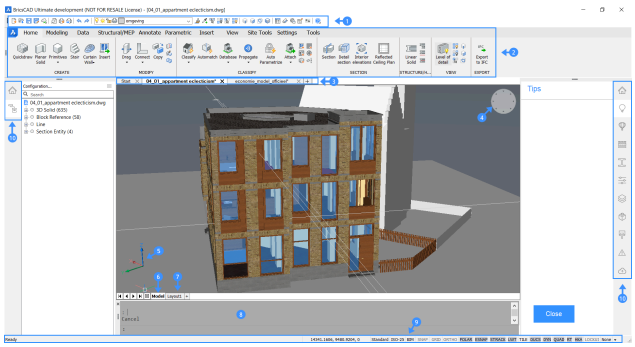
20.2.2 Exploring the interface

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

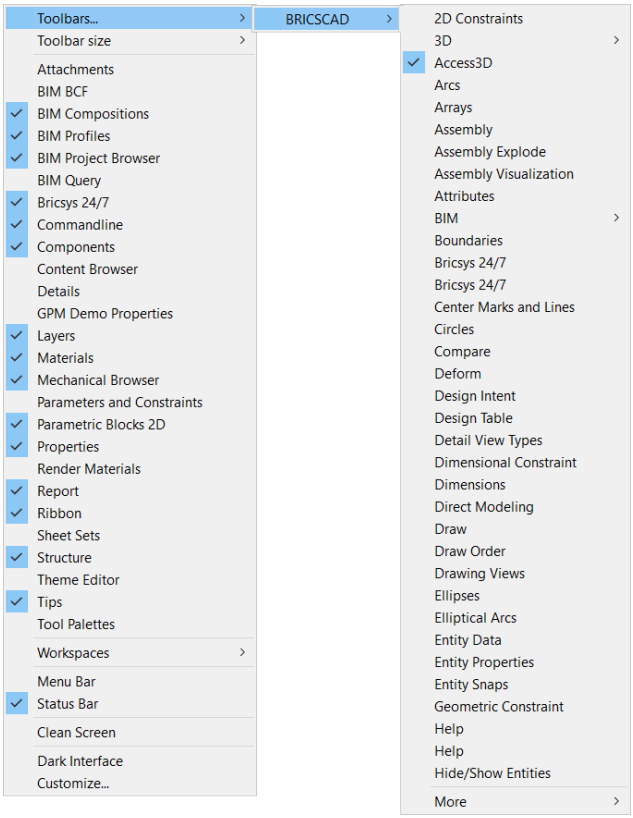


1. Toolbar

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.

Uwaga: To open more toolbars you can right-click, choose toolbars and check the topics you want.

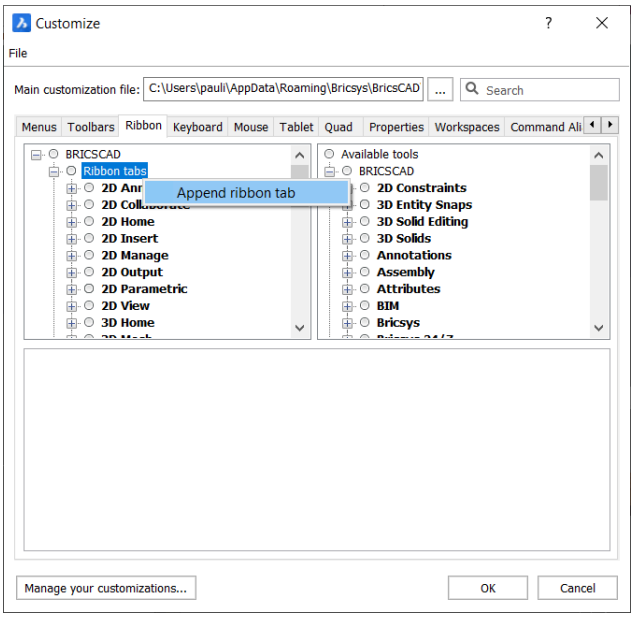
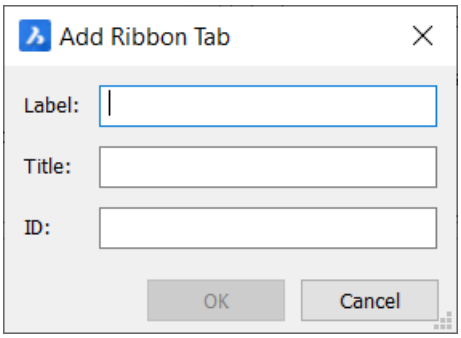


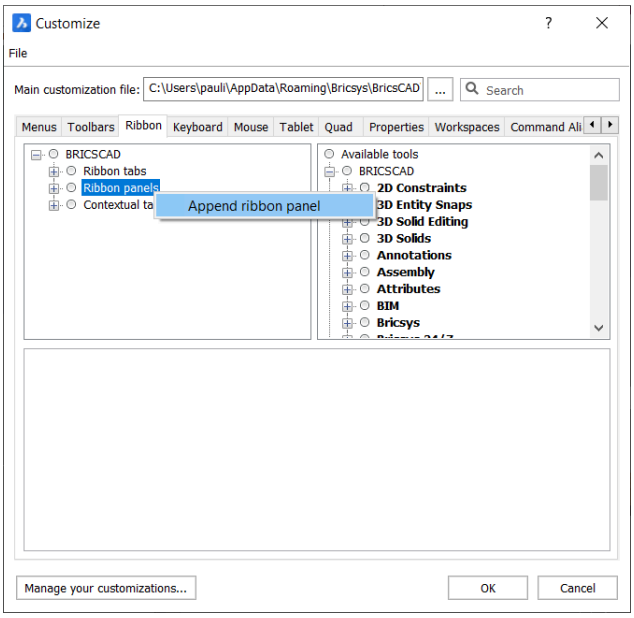
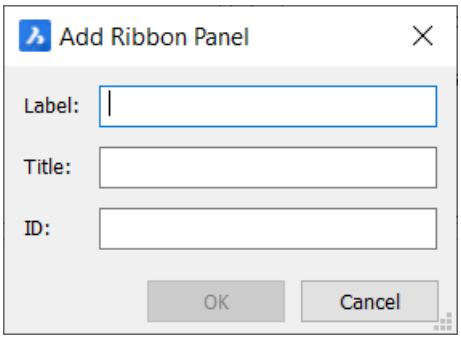
2. BIM Ribbon

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.

Uwaga: The content of the ribbon depends on the current workspace (see Editing In Workspace).

Uwaga: 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

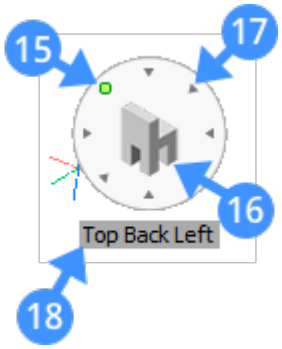
Document tabs allow you to:

- 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

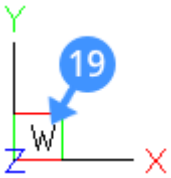
Each drawing has at least one layout and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale.

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.

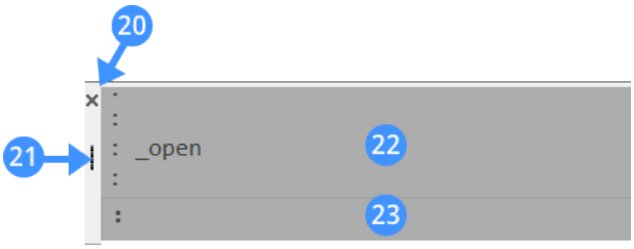


Uwaga: WCS stands for World Coordinate System.

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.



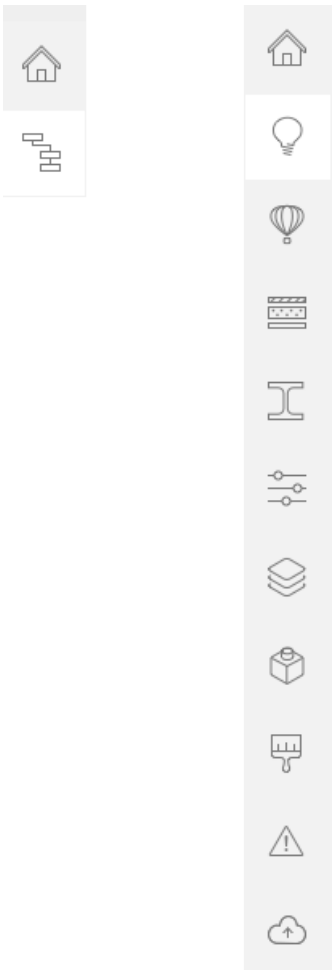
Status field (24)	Displays the status of the software.

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**.



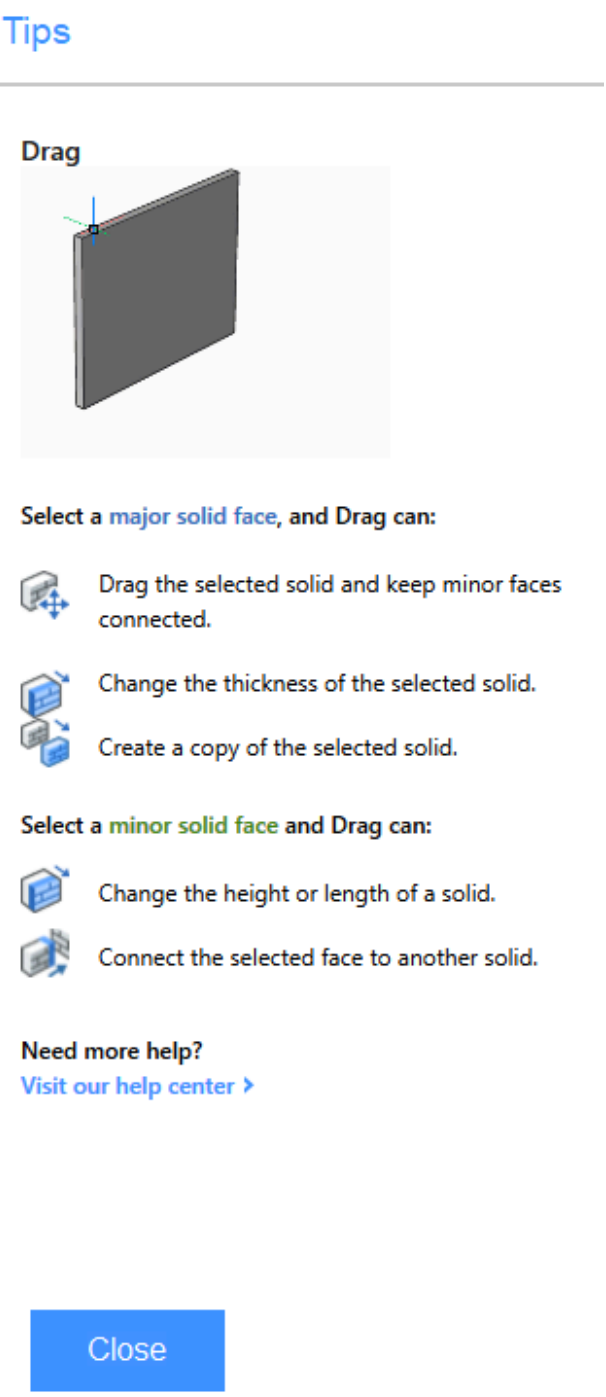
Uwaga: You can always add panels in the Tool Panel by right-clicking and checking those you want.

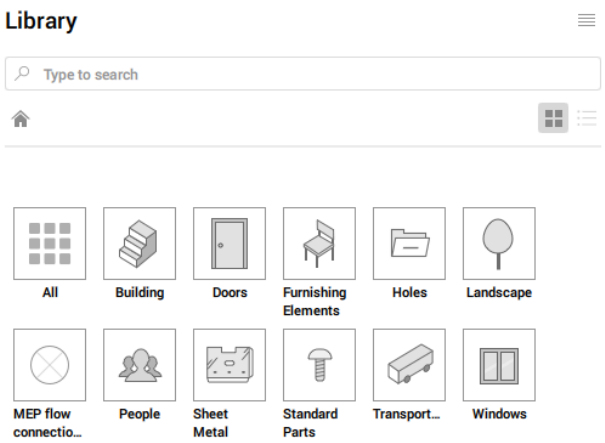
We will now have a look at the functions of the different dockable Panels from the Tool Panel:

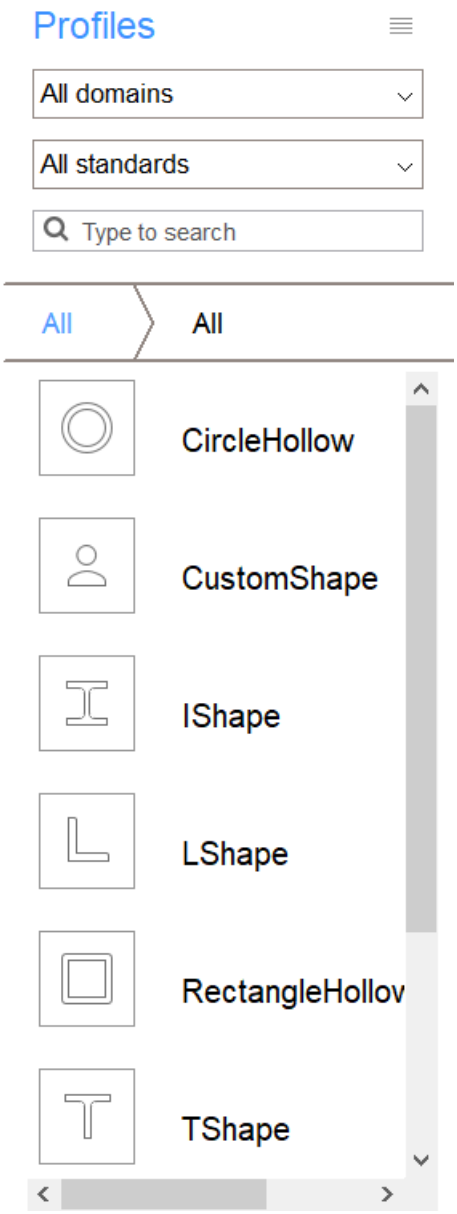
Project Browser	The BIM Project Browser allows you to easily navigate through the entire content of a project: models, sections, drawings, and schedules. See The **BIM Project Browser** article on how to use this panel.
Structure Browser	The **Structure Browser** is an interactive tree that displays 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:

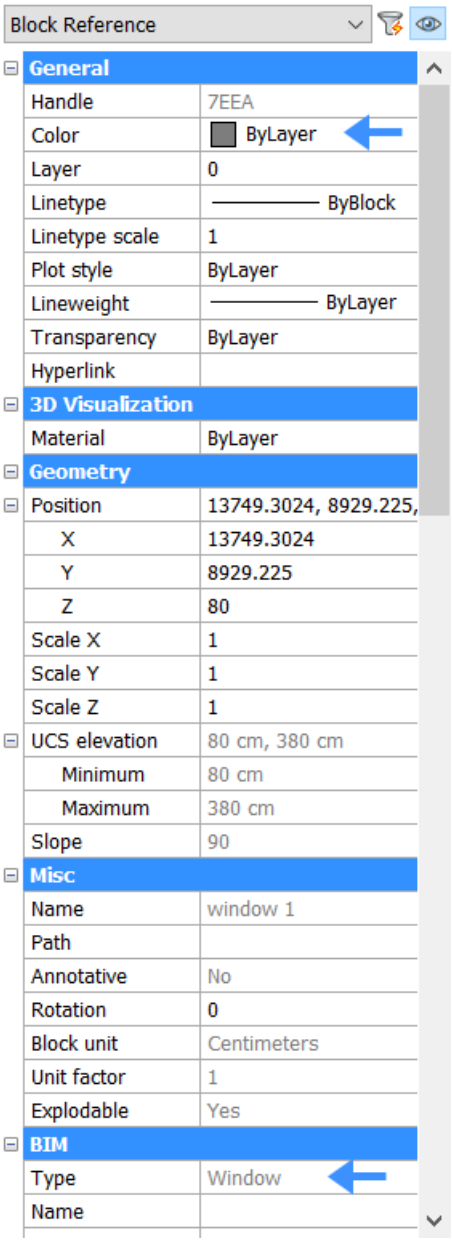


Library	The **Library** panel gives a list of components that you have by default. You can simply click and drag these components into your drawing.  At the bottom of the panel, you have the button . When you have made for example a drawing of a window, you can click on this button and add your own drawing 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.

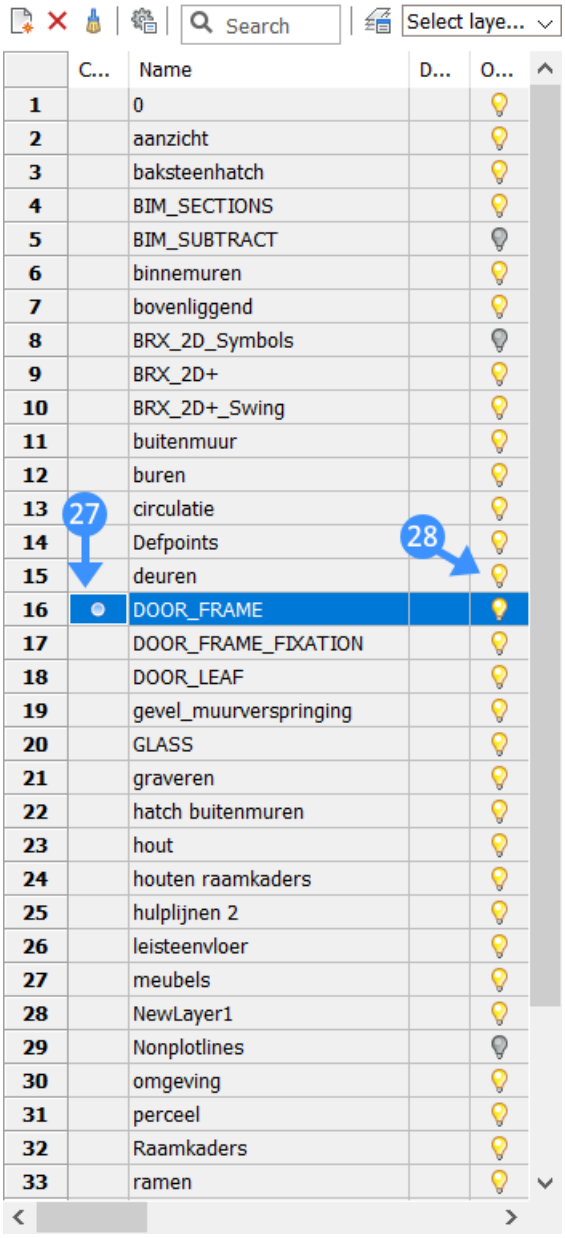


For example, here we can see that the selected item is a window and is colored by layer. You can adjust the properties as you like.

See the **Properties** article to see how to work with this panel in more detail.

Layers

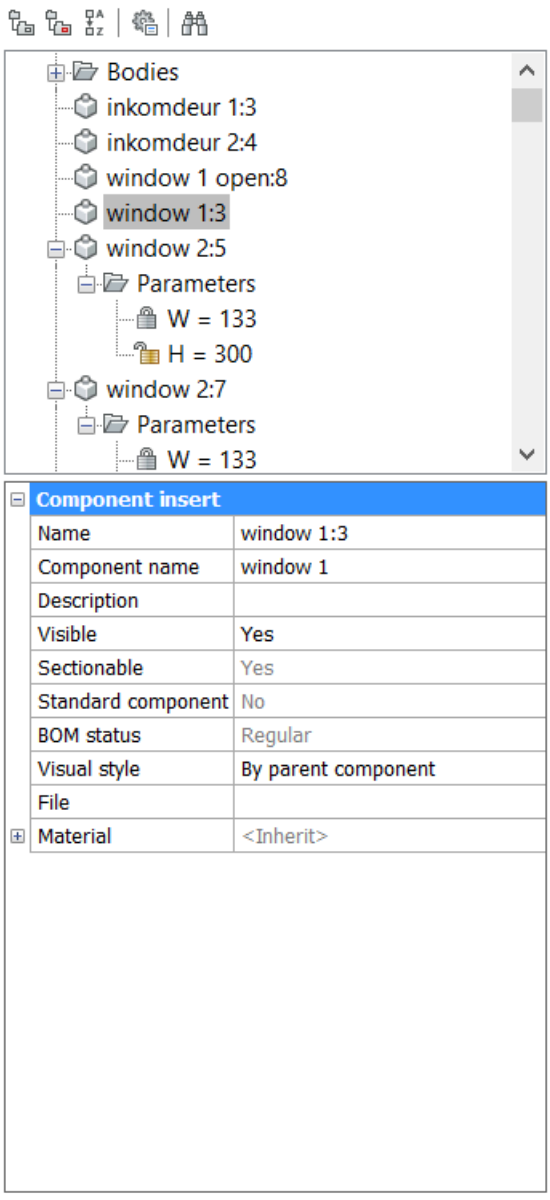
The **Layers** panel shows all the layers of your drawing.
It shows you the current layer (27), you can change it by clicking left of another layer. It also shows if a layer is ON or OFF by the state its lightbulb icon (28). Other layer properties such as color, line type,... can be found and adjusted 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).

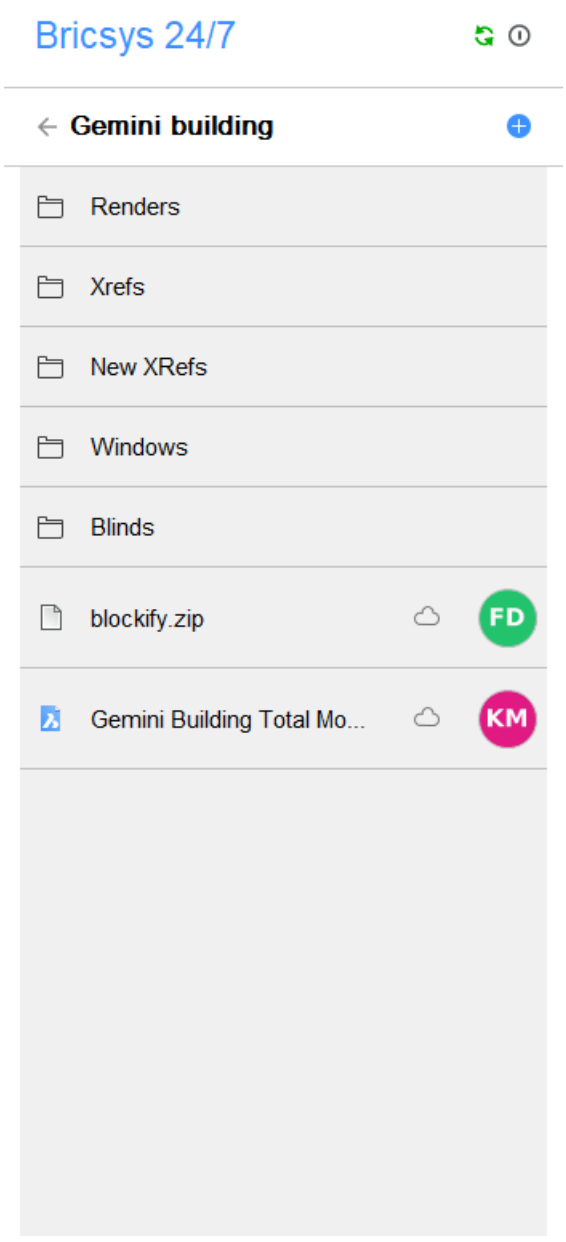


The **Mechanical Browser** allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right click on a particular node. For example, in the image above you could add a description for the selected window.

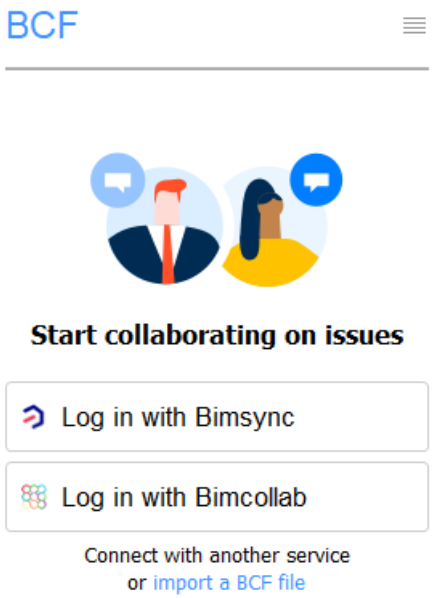
Render materials	The **Render materials** Panel contains a library of predefined materials that you can assign to solids by dragging them on. You can also see a list of all the materials that you are currently using in your drawing. See the **Materials** article to see how to edit and create materials.
Report	Some commands use the **Report** panel to report the command output; e.g. all SHEETMETAL (Sm*) commands, DMAUDIT and DMSTITCH.

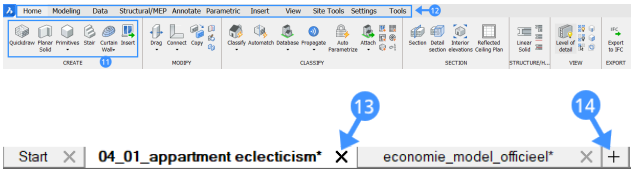
Bricsys 24/7

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.

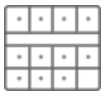
BIM BCF	Here you can import BCF (Bim Collaboration Format) files and connect to a BCF server. The BCF topics are listed in this panel.  The top part of the new **BCF** panel shows all the topics and can look similar to the **Library** panel. The lower part of the new **BCF** panel shows the currently 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:

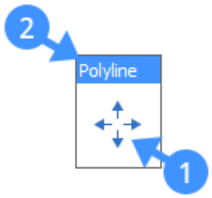
- Click the **RT** field in the Status bar.
- Type ROLLOVERTIPS in the Command line then set a value.

20.2.3.2 Quad layout

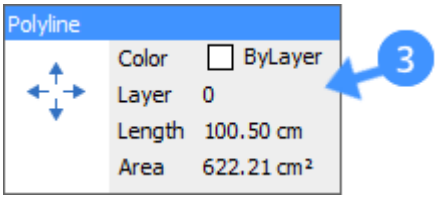
The **Quad** has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace.

- Based on the highlighted entity type, the **Quad** displays the most recently used tool (1) with the entity name (2).

  Uwaga: When multiple entities are under the cursor, hit the TAB key to cycle through the different entities.

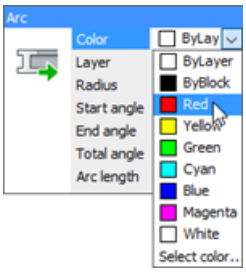


- When the **Rollover Tips** setting is on, a limited set of properties (3) of the entity under the cursor display in the **Quad**.



  The property values in the **Rollover Tips** can be edited.

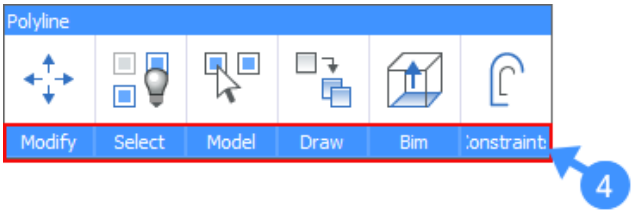
  The following figure shows the color of an arc being changed using the **Rollover Tips**.



Uwaga: To display the **Rollover Tips**, simply click the title bar of the **Quad** after expanding it.

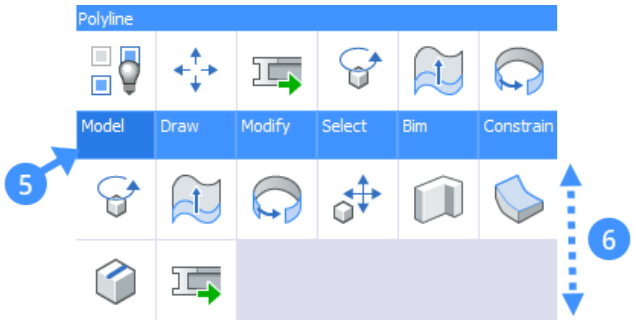
- When the cursor is moved on the **Quad**, a number of tool tabs display at the bottom (4).

  Uwaga: The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6).

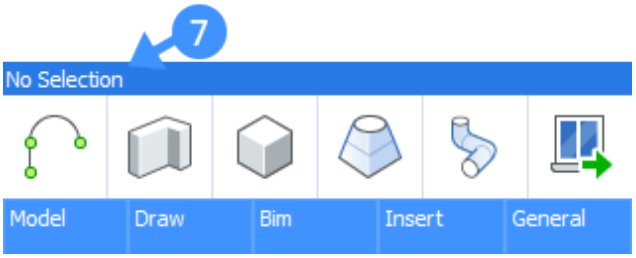


- When you move the cursor over the tool tabs title bar (5), the tool groups expand (6) under the cursor, which provides you with more tool options.

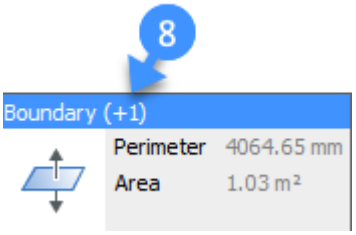
  Uwaga: The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor.



- 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.

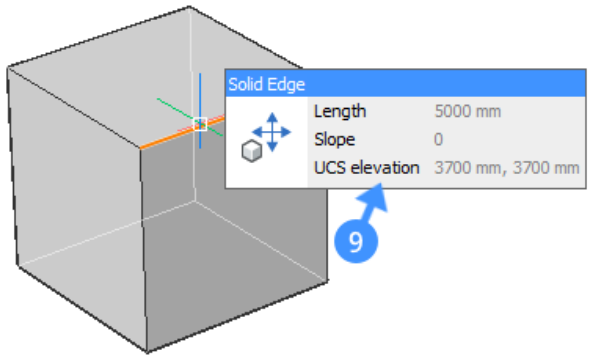


Uwaga: The **Quad** also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

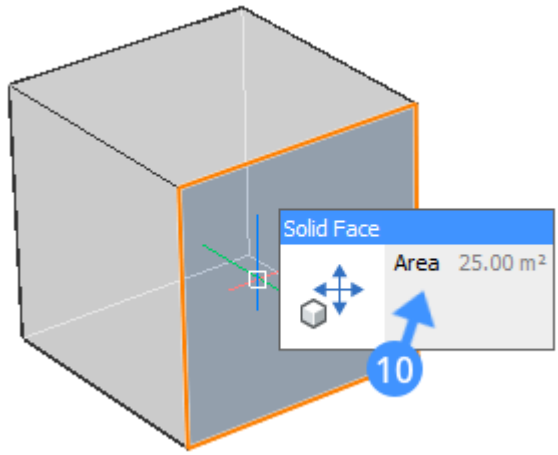
20.2.3.3 About: selecting entities

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.

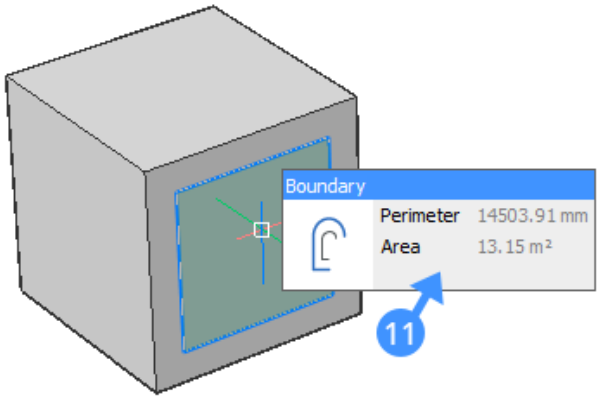
- Selecting an **edge**, the **Quad** appears showing the edge length, slope, and elevation (9).



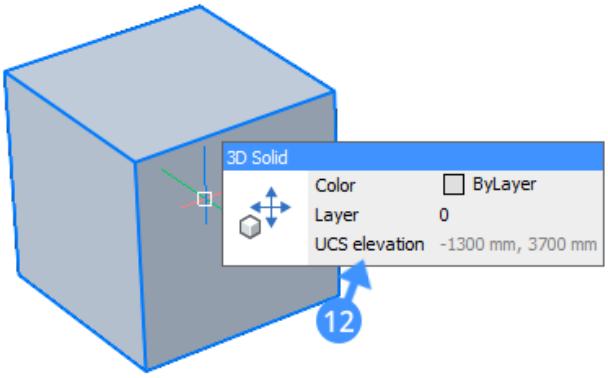
- Selecting a **solid face**, the surface area is shown in the **Quad** (10).



- 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

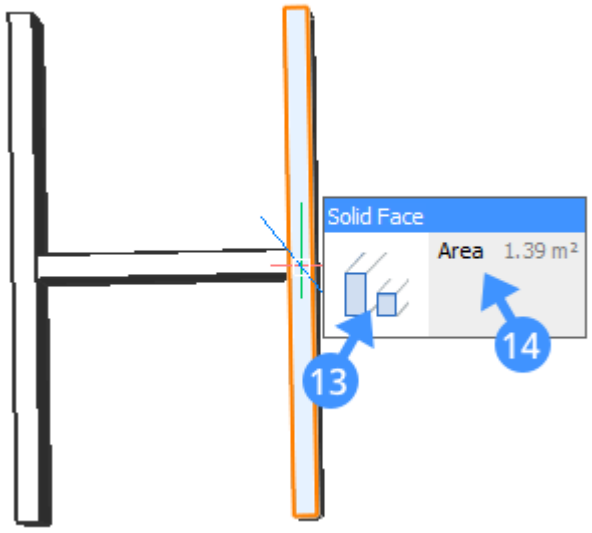
The following are general steps to use the **Quad** cursor.

Uwaga: The **Quad** and **Rollover tips** are both enabled.

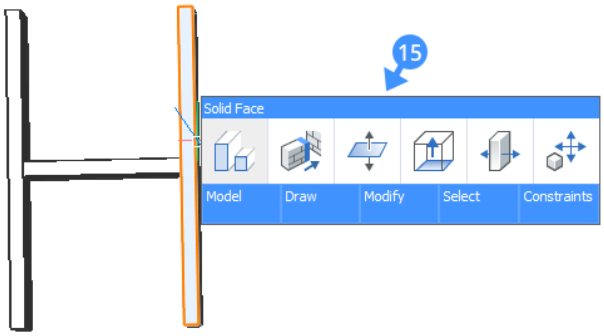
1 Hover the cursor over the entity you want to manipulate.

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

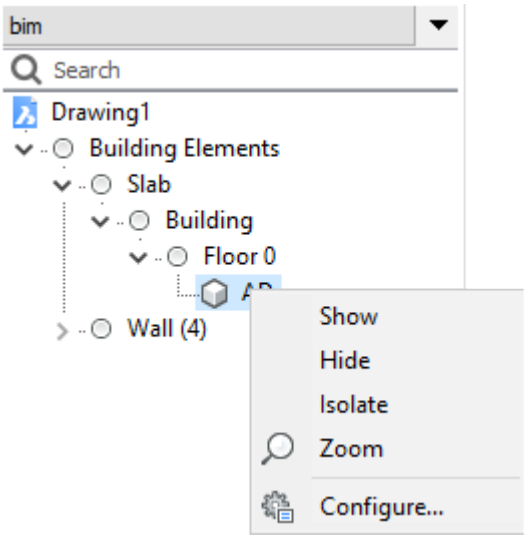
The **Hide** tool temporarily hides the selected entities in the view.



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).



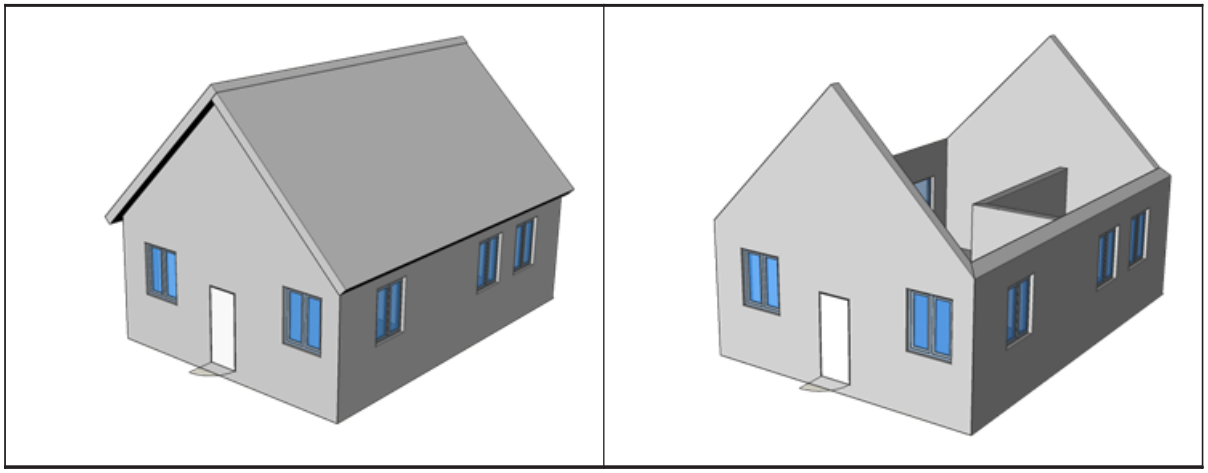
20.2.4.4 Procedure: hiding entities

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.

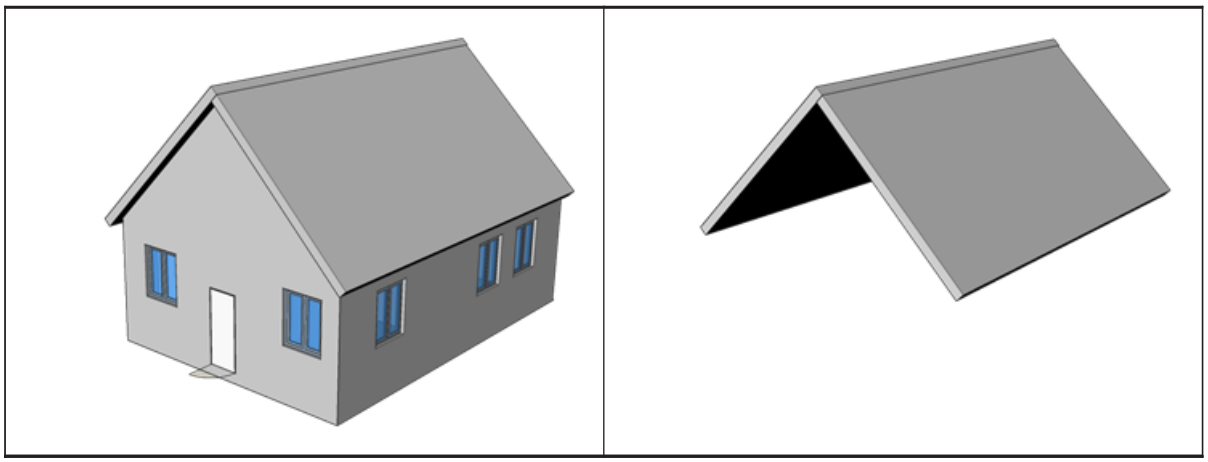
20.2.4.6 Procedure: isolating entities

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

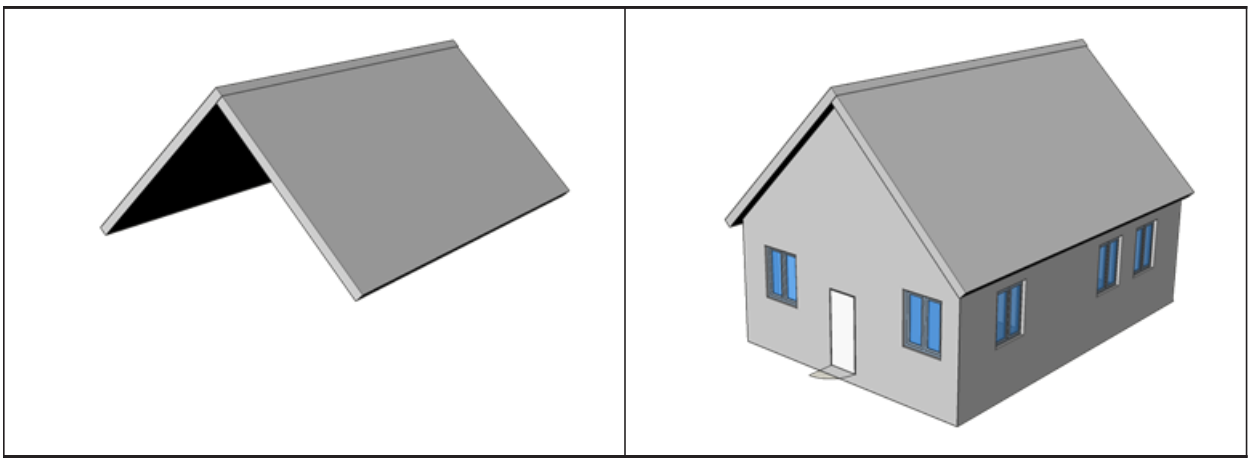
The **Show Entities** tool brings back all hidden entities in the view.



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.

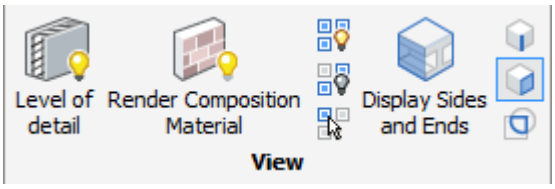
Selecting: you can select as many entities as you want.

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**



Uwaga: 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.

Uwaga: BIM Sides/Ends are only available on classified linear elements with the attached library profile.

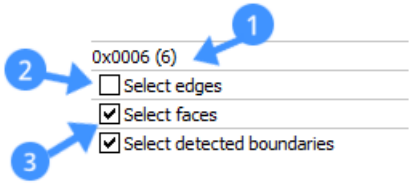
20.2.5.3 Changing selection modes setting

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:

Uwaga: 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**.

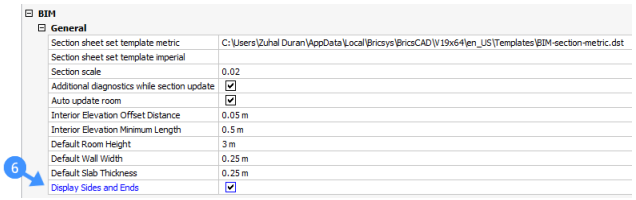
  The following illustration shows the sum of the values (1) of the SELECTIONMODES system variable when the **Select edges** option is disabled (2) after clicking the checkbox once, the **Select faces** and **Selected detected boundaries** are enabled (3).



- 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.

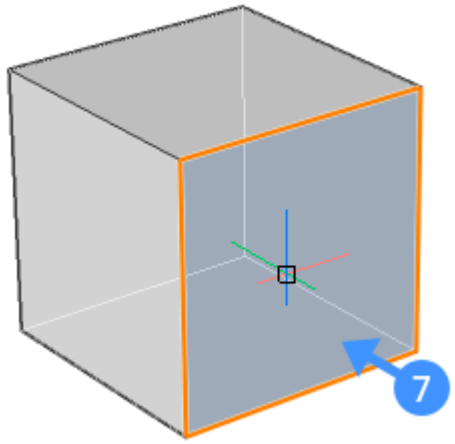
  The following illustration shows the **BIM** > **General** system variable when the DISPLAYSIDESANDENDS is enabled (6) after clicking the checkbox once.



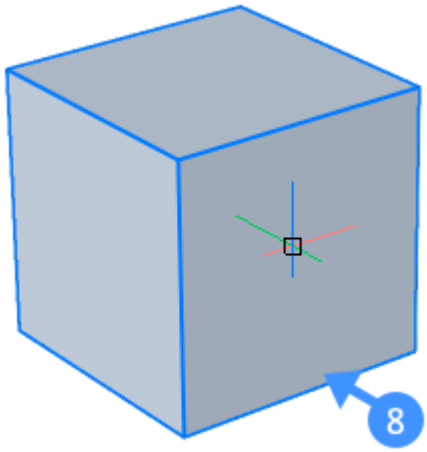
20.2.5.4 Selecting sub-entities

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default), do one of the following:

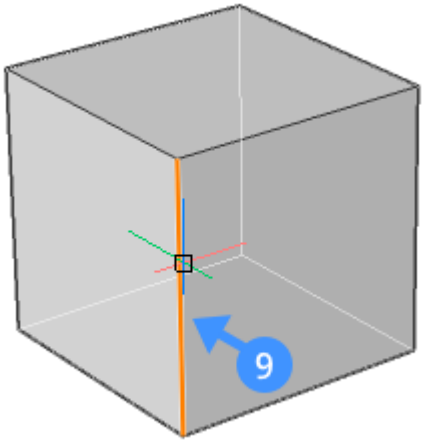
- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face displays in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.



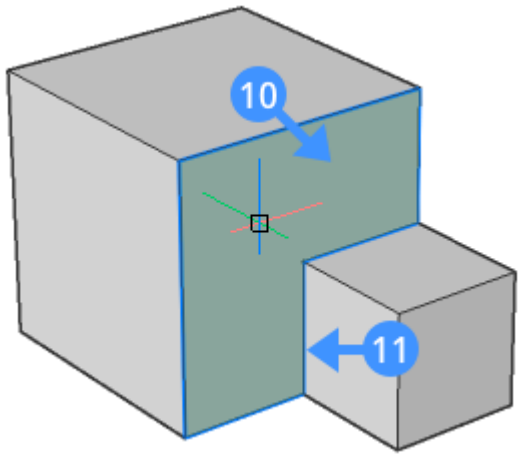
- To highlight a solid, hover over one of its faces while holding down the CTRL-key. The solid displays in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.



- 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.



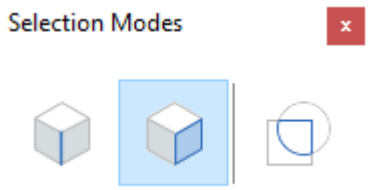
- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.
- When the detected boundary is highlighted, it displays with a color. By default, inside of the boundary displays in green (10) and the outer edge of the boundary displays in blue (11).



Uwaga: 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).

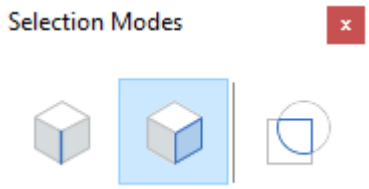
7 (**Optionally**) Press Tab to change the face selected as the reference face for the dynamic dimension.

Uwaga: 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:

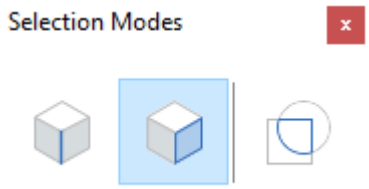
5 Type a value in the dynamic dimension field and press Enter.

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.

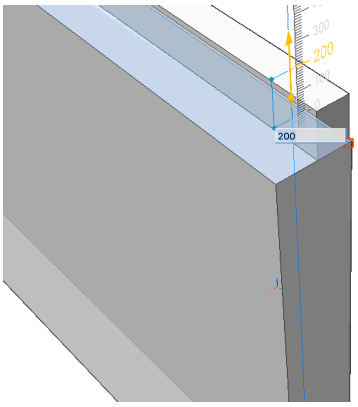
Modify the height ply-by-ply

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.

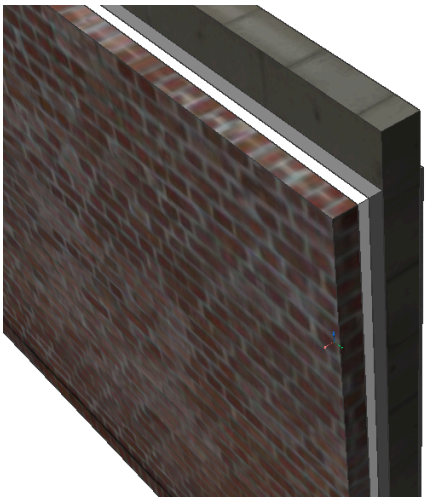
1 Move the cursor over the ply you want to edit and choose **Push/Pull** in the **Model** command group of the Quad when the ply highlights.



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.

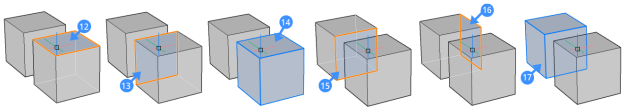
1 In the example below, the top face of the front solid is highlighted (12).

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

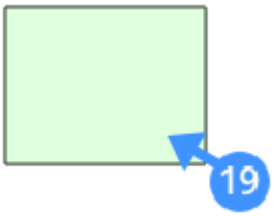
The selection boxes allow you to select one or more than one entity at a time. To use a selection box, left-click on an empty space in the drawing area, and move the cursor until the selection box covers the entities that you want to select.

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.

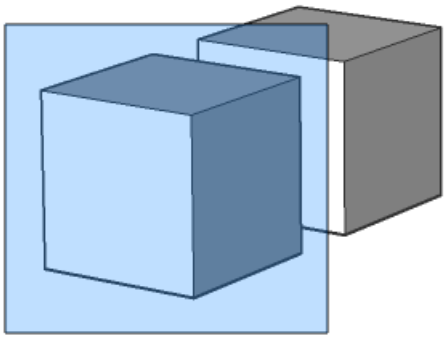


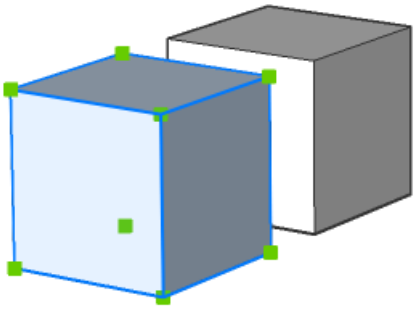
The following steps show how to select multiple entities using the blue selection box:

Uwaga: 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.

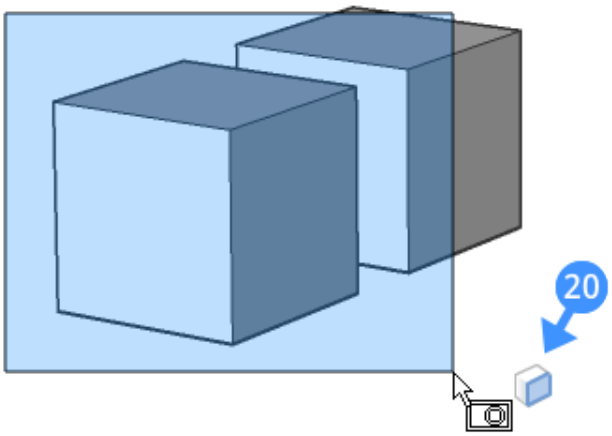
1 The example below shows the blue selection box usage when the selection mode is set to **Solids** (default).

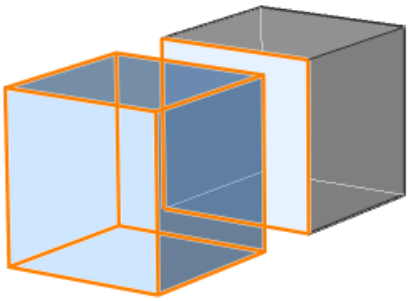




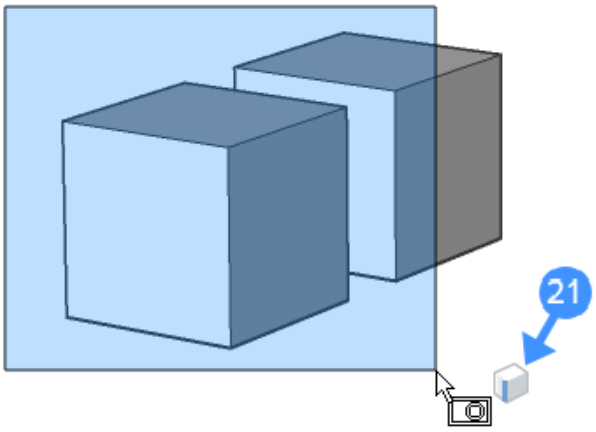
The blue selection box covers the left cube completely and thus only this cube is selected in the model.

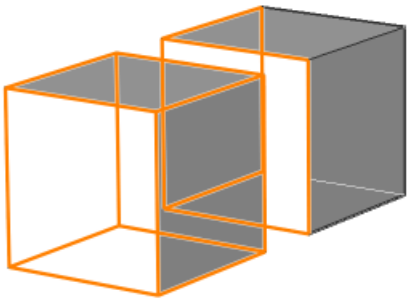
2 To select faces of 3D solids and surfaces, press CTRL key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to **Faces**.





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**.

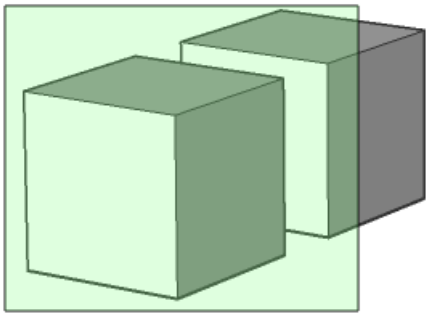


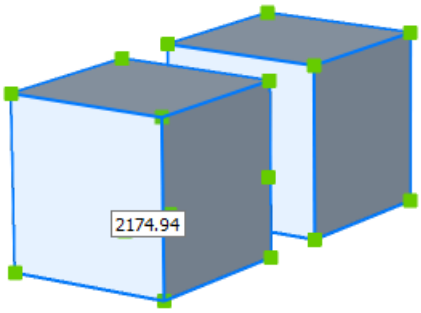


The following steps show how to select multiple entities using the green selection box:

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.

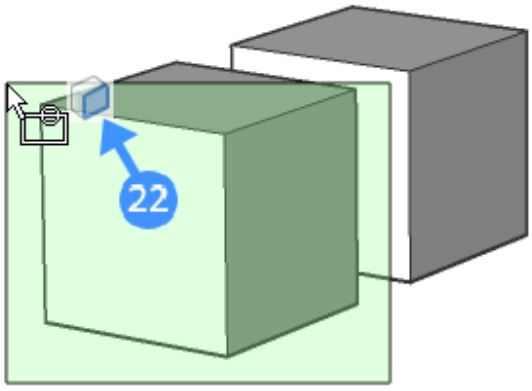
5 The example below shows the green selection box usage when the selection mode is set to default.

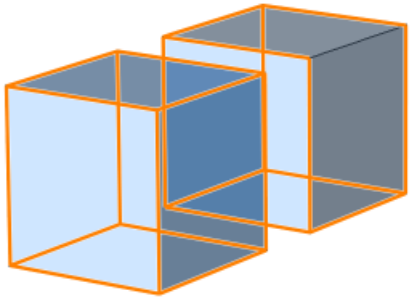




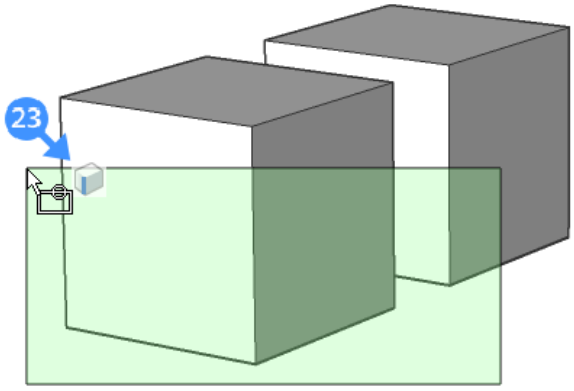
The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

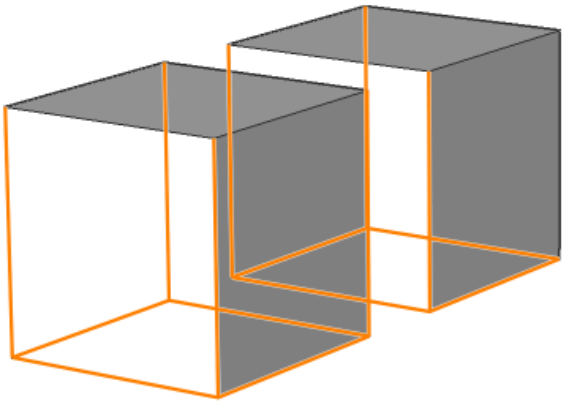
6 To select faces of 3D solids and surfaces, making use of the green selection box, press the CTRL key once during window selection. The ‘Faces’ icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected.





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.



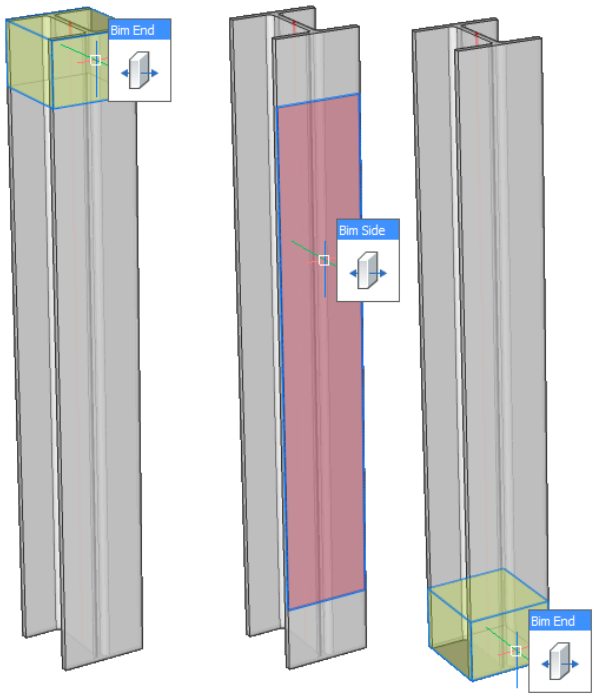


20.2.5.9 Selecting/Highlighting BIM sides and ends on profiles

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.
 [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 Modeling techniques

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.

Uwaga: 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.

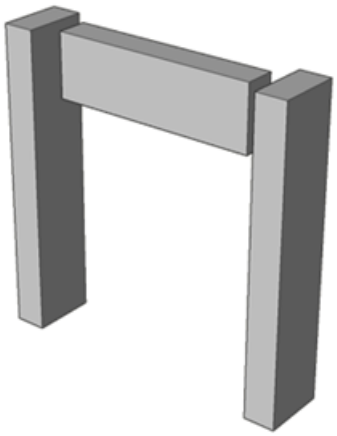
3 You are prompted: Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/ adJust profile]:

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.

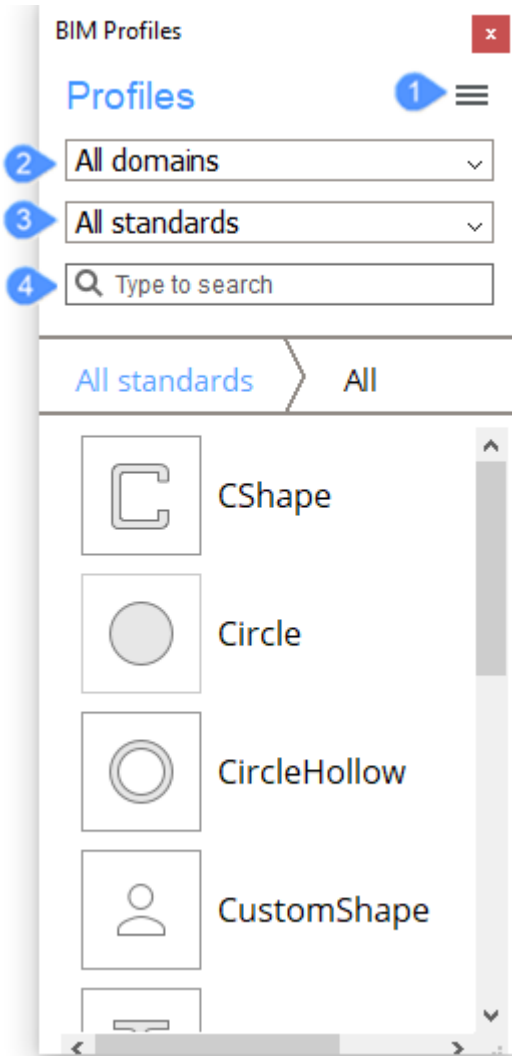


20.3.1.3 About BIM profiles

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**.

Uwaga: 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**.

Uwaga: 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.

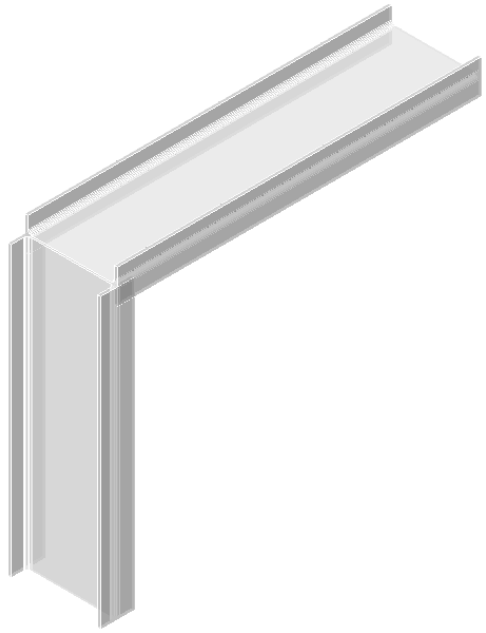
20.3.1.4 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

Uwaga: 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.



Uwaga: 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.

Uwaga: 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.

Uwaga: 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).

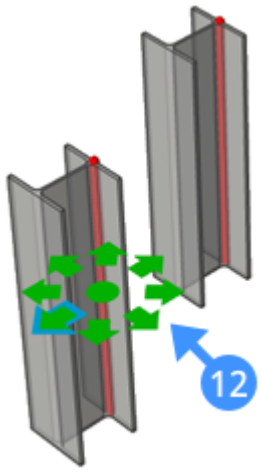
The eccentricity widget appears.

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

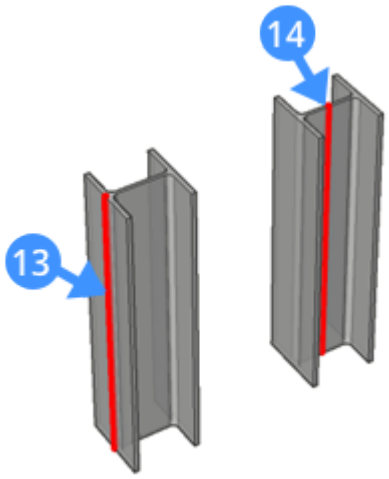
1 Select any linear building element in your model.

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

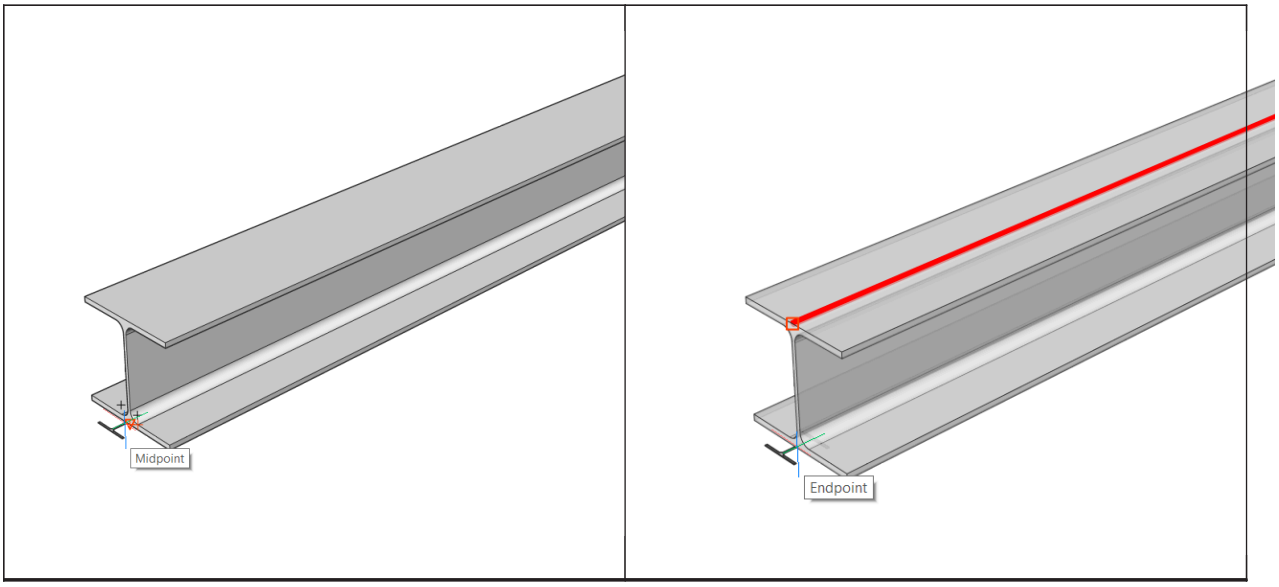
The DISPLAYAXES variable system can hide or view the Axes of linear solids in order to edit the structure more easily.



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.

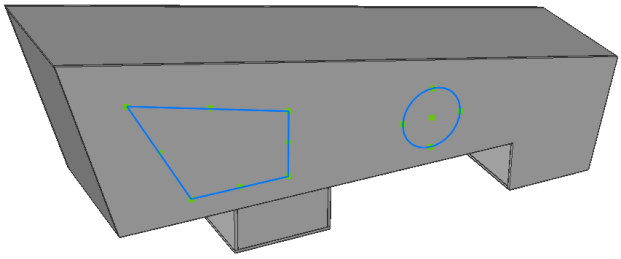
When you have a closed 2D entity or boundary on the face of a solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box and change the properties of this created window in a BIM model.



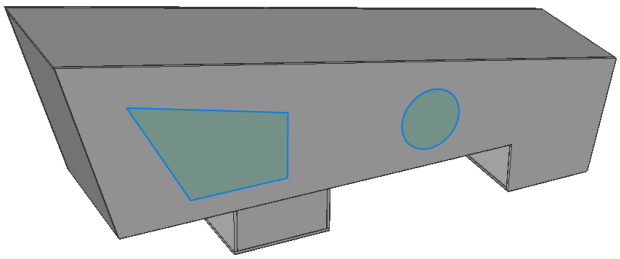
20.3.2.3 Creating a parametric window using boundaries

1 Draw a polygon on the face of a solid (wall).

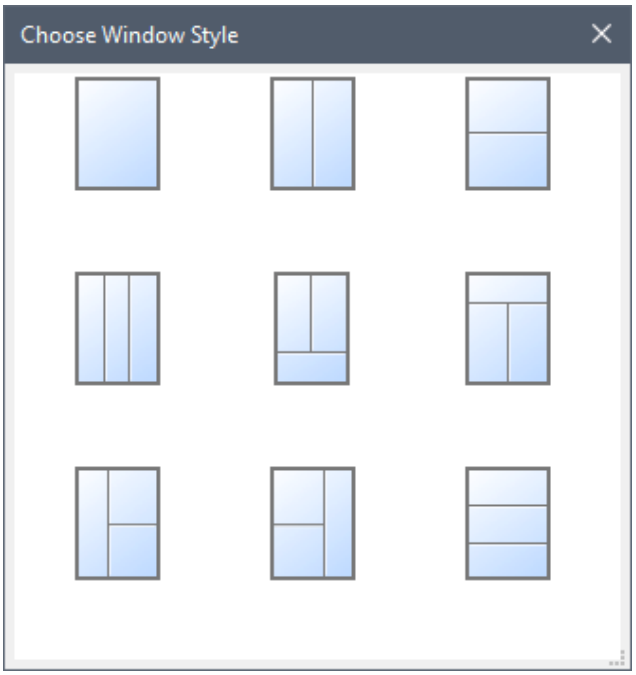
1 Based on this base profile, the window opening will be created.



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.

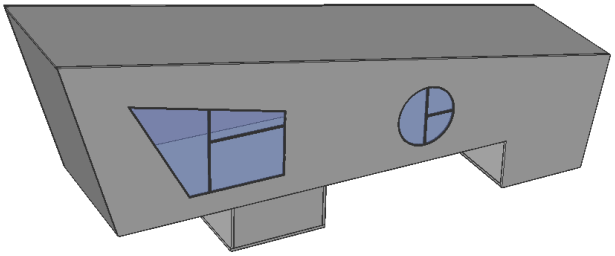


A **Choose Window Style** dialog box displays:

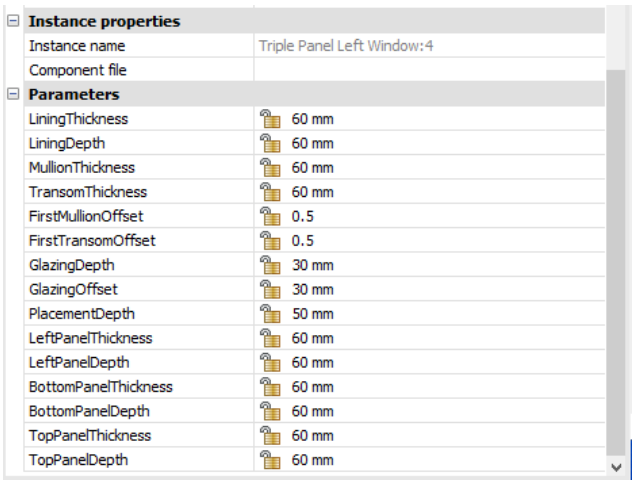


3 Select a window style in the dialog box.

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.



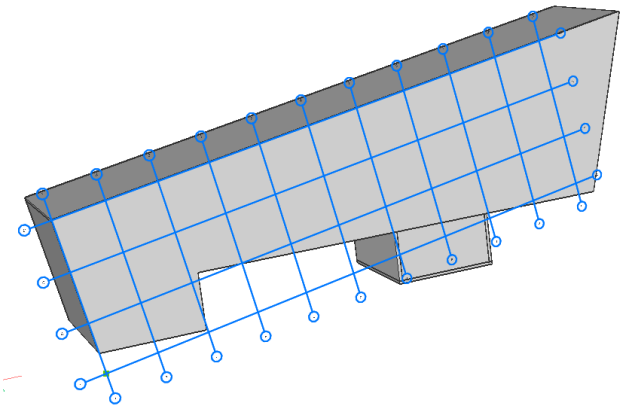
To change one of the default parameters of this window, type a new value in the property field.

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

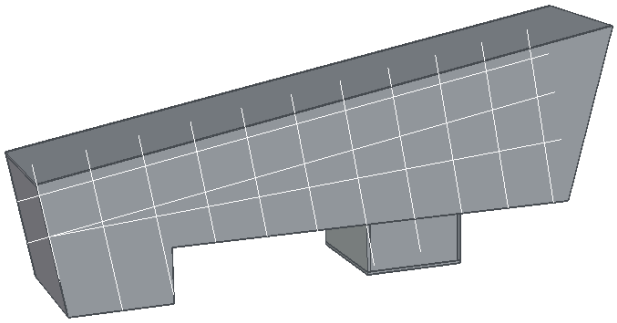
1 Use BIMGRID command to create a Grid on the face of the wall. Choose the basic dimensions in the Command line. Changing the **U-line** and **V-line** panel options allows you to specify the desired size and number of panels on the surface.

Uwaga: 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.

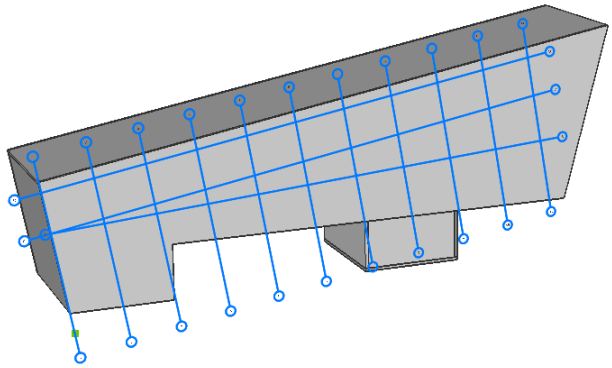


2 Use BEDIT or REFEDIT to edit the Grid.

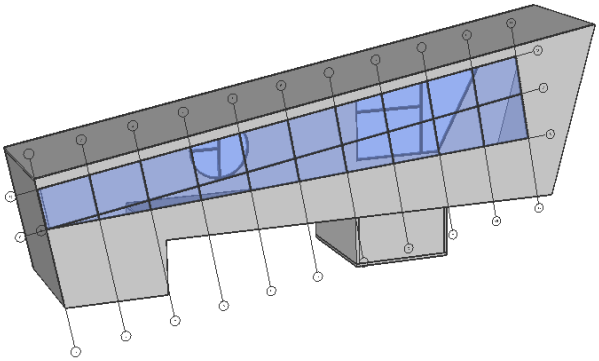
3 Use the Manipulator to change the position of the gridlines on the wall.



4 Select the Grid and select **Create Window** in the Quad.



5 The window is generated based on the gridlines.



Uwaga: 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.



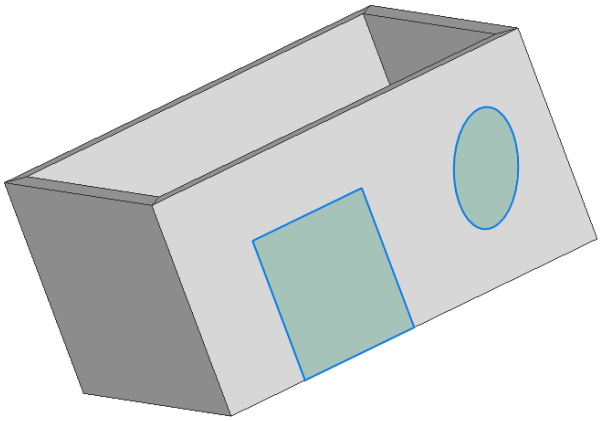
For more information about this command, visit the Command Reference article BIMWINDOWCREATE.

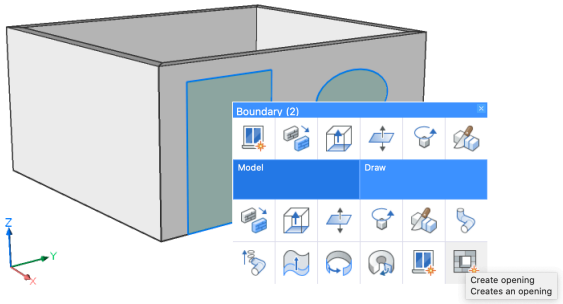
1 Draw one or more closed polylines on the face of a wall.

2 While boundary detection is on, hover over the polygon.

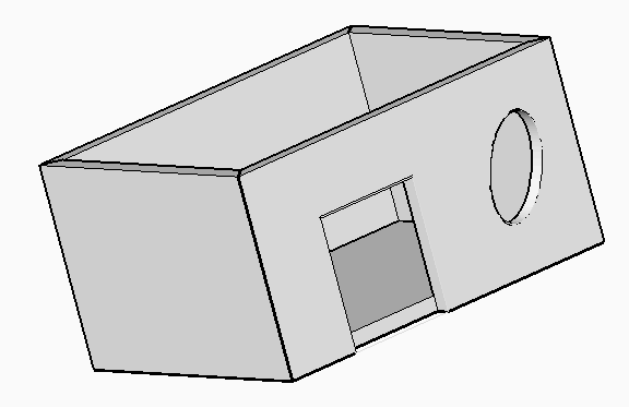
3 Or select the closed polyline.

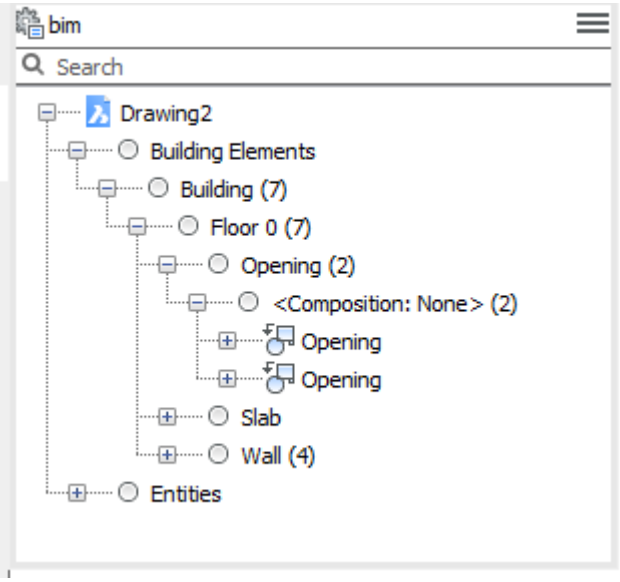
4 Select the **Create Opening** tool from the Quad.



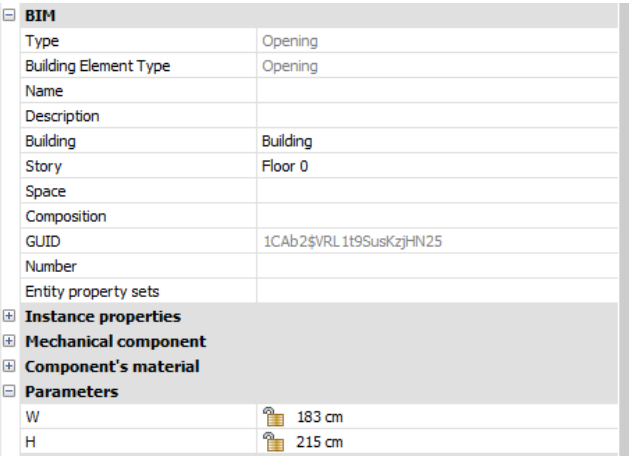


5 The opening is created based on the closed contour. In the **Structure Browser**, the opening is added as a block reference. The block reference is classified as an opening.





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.



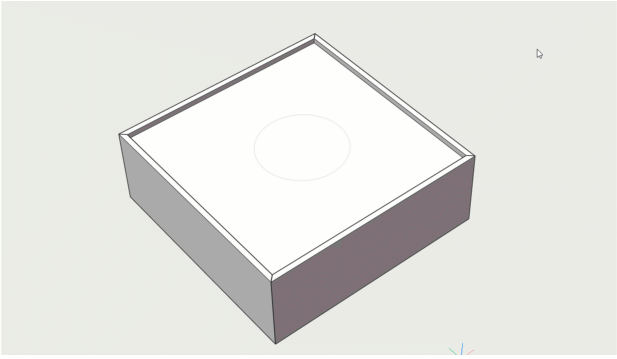
BIM	
Type	Opening
Building Element Type	Opening
Name	
Description	
Building	Building
Story	Floor 0
Space	
Composition	
GUID	1CAb2$VRL1t9SusKzjHN25
Number	
Entity property sets	
Instance properties	
Mechanical component	
Component's material	
Parameters	
W	183 cm
H	215 cm

Uwaga: If your closed contour has a rectangle shape, the opening element will include Width and Height parameters.

Uwaga: Opening elements contain subtractor solids. To display the subtractor solid, open the **Layers** panel and select the **BIM_Subtract** layer.

Uwaga: 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.

Uwaga: 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.

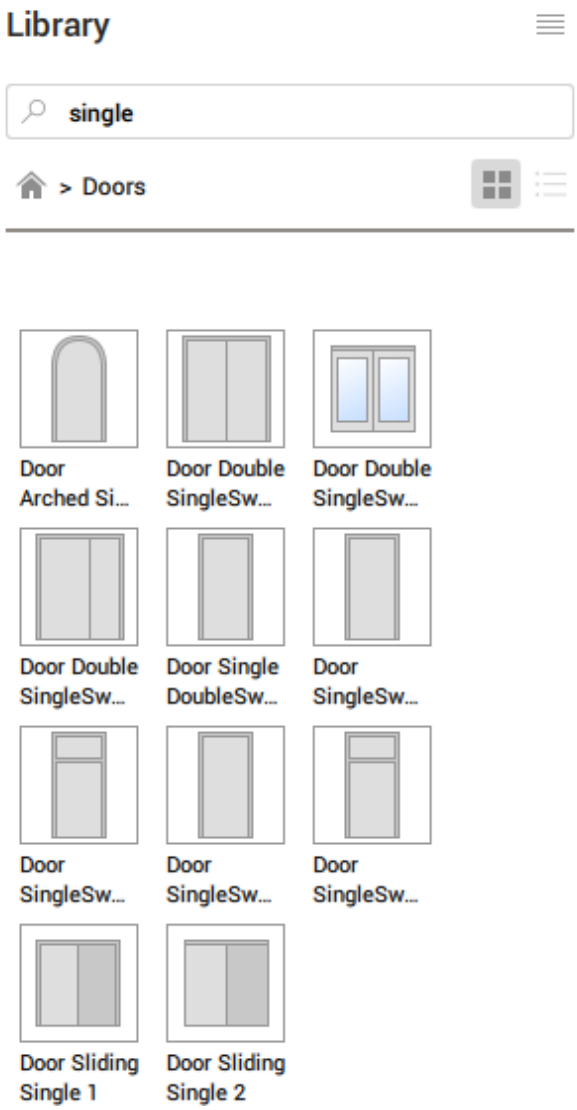
4 (**Option**) Click the **Delete** icon (2) to delete the block file.

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.

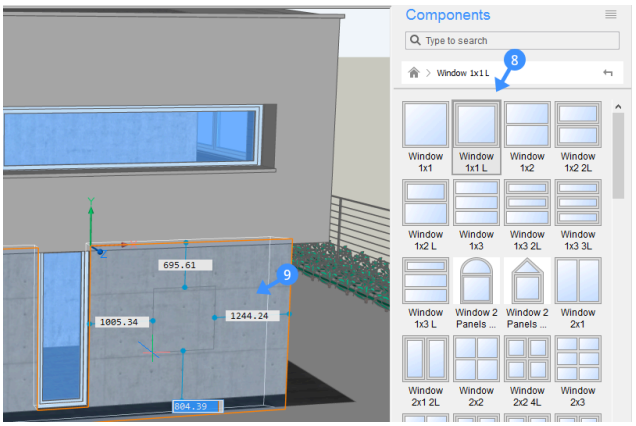


20.3.3.7 Inserting a BIM window into your 3D model

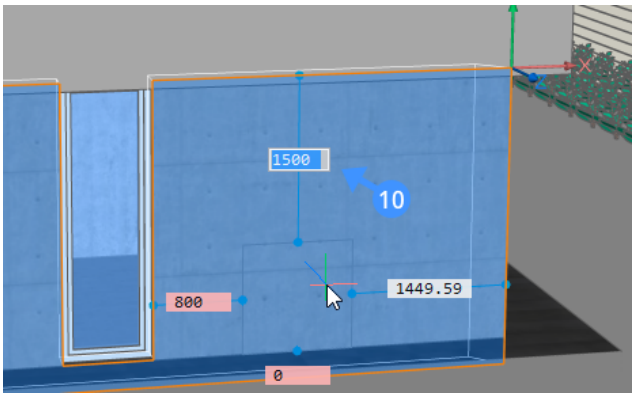
1 Click the **Windows** category tile in the **Library** panel.

2 Click a window (8).

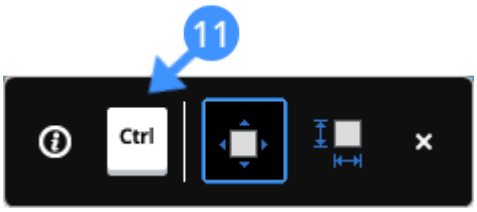
3 Drag and drop the window onto the wall (9).



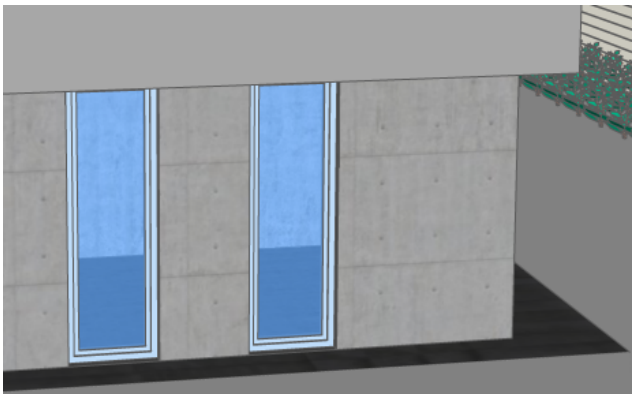
4 If Dynamic dimensions (**DYN**) is switched on, set the window location by entering the dimensions into the DYN fields (9). Press the Tab key to cycle through the DYN fields (10). A DYN field turns red to indicate the distance is locked.



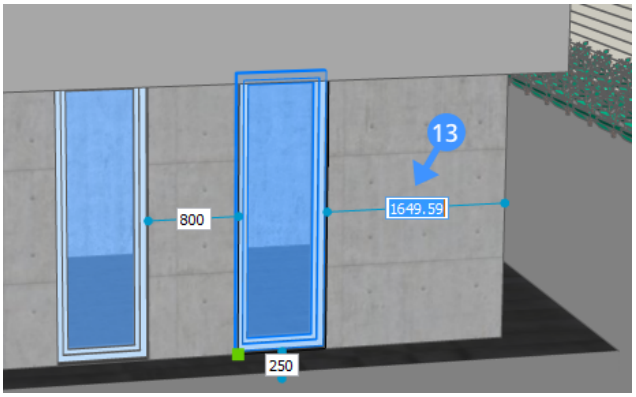
5 To edit the dimensions of the window press Ctrl (11) once and then use the Tab-key to cycle through the DYN fields (12).



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

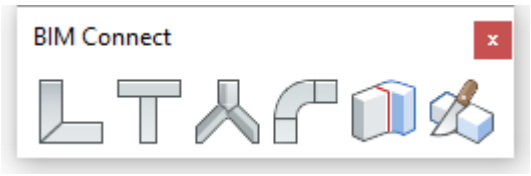
20.3.4 Connecting solids

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.



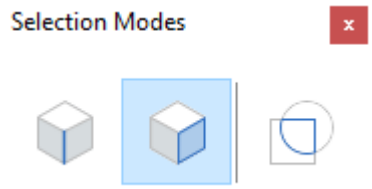
Use the CTRL key to cycle between the connection types **or**
You are prompted: Select base solid [Switch] <Accept>. Type **A**to use the selected connection, type **S** to switch to the desired connection type.
The 3D model updates dynamically when cycling the L-connection types.

Uwaga: 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.

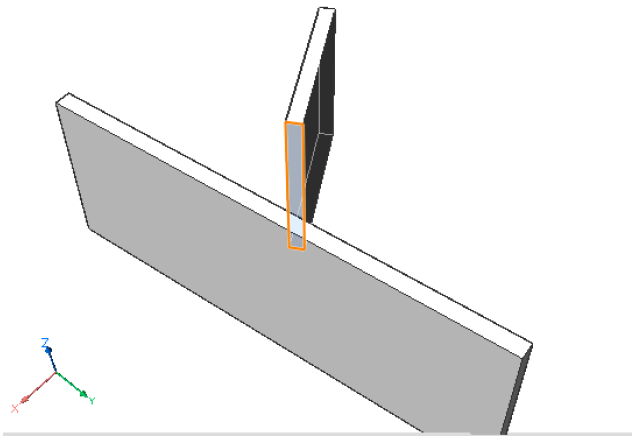


2 Type TCONNECT in the Command line or select it via the Quad.

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.

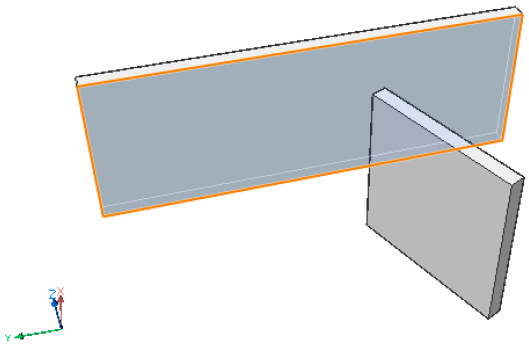
Wskazówka: The SELECTALIGNEDFACES command in the **Select** command group in the Quad selects all faces in the 3D model that are in the same plane (coplanar) with a selected face.

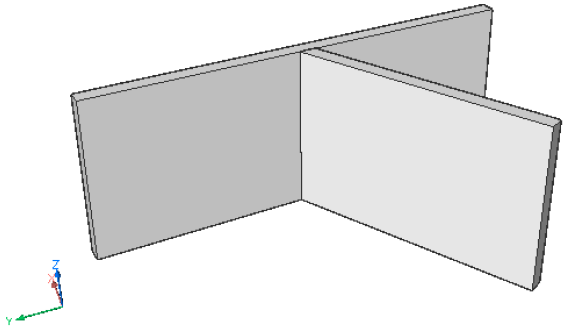
6 You are prompted: Select entities to connect to or [Connect to nearest/Disconnect/selection options (?)] <Connect to nearest>:

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.

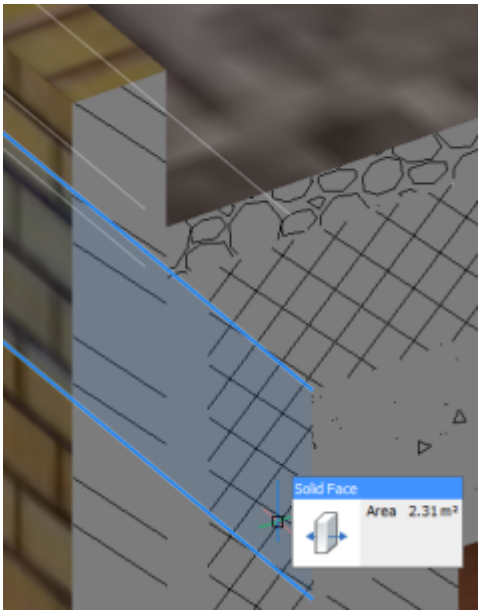
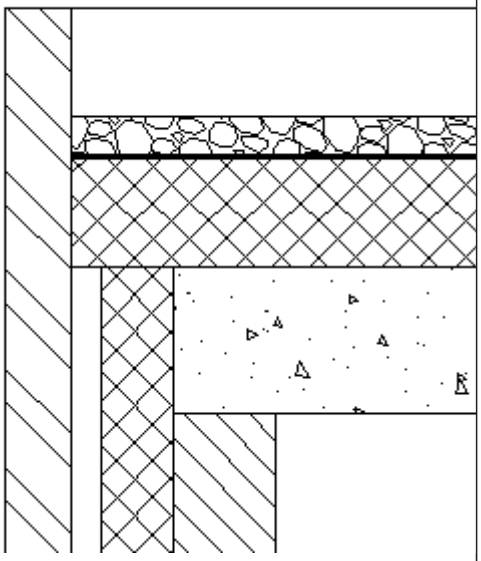
  Right click to stop selecting faces.





Wskazówka: 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.

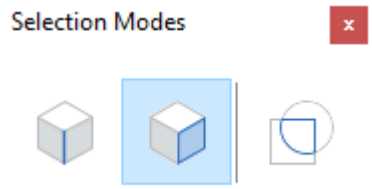
3 The nearest distance between the solids displays.

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.

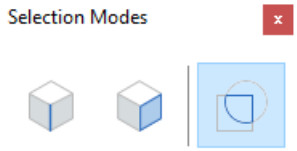
3 When the face highlights, choose **BimCopy** in the **Model** command group of the Quad.

A copy of the solid displays dynamically. The ruler indicates the current distance.

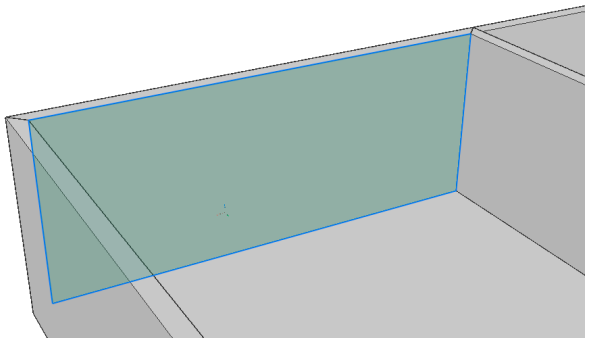
4 Type a value in the dynamic entry field and press Enter.

Creating a partial copy of a solid

1 Make sure the **Enable boundary detection** option of the SELECTIONMODES system variable is selected.



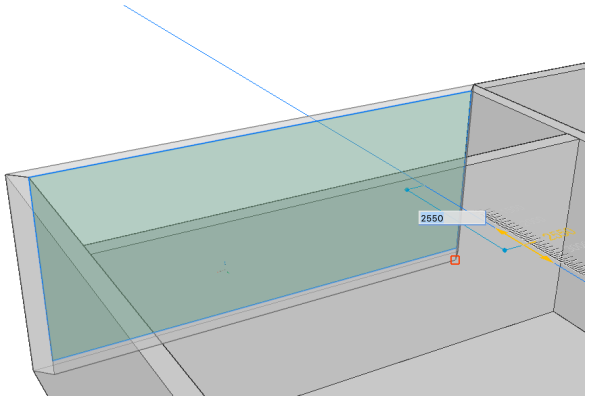
2 Highlight the part of the boundary for the part of the solid you want to copy.



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.

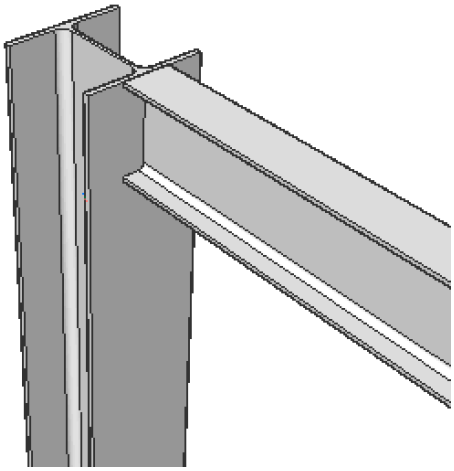
4 Type a value in the dynamic entry field and press Enter.



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.



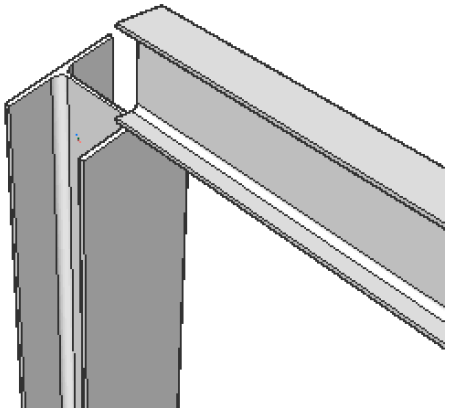
1 Type LCONNECT in the Command line.

2 Select the two solids.

3 The connection is automatically made, press CTRL until the disconnect icon is highlighted.



4 Press Enter.



Uwaga: You can use Propagate to disconnect similar nodes.

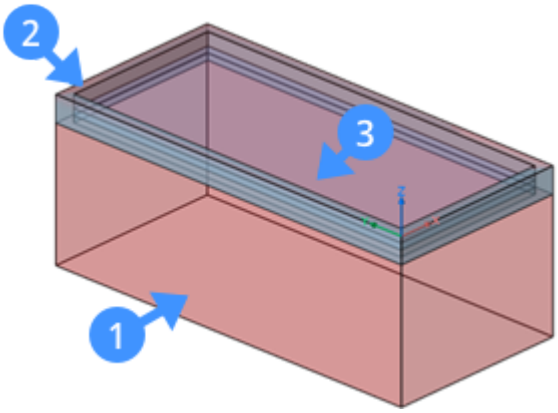
20.3.5 Creating a custom window

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)

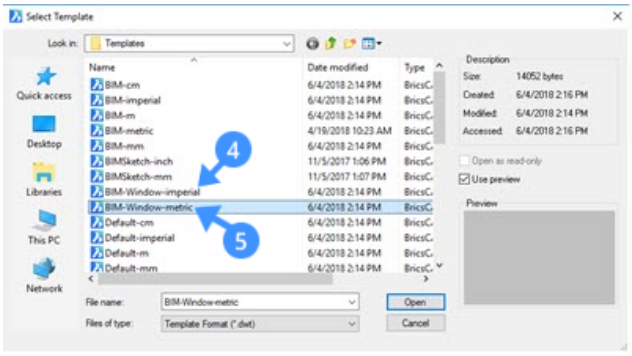


Step 1: Creating the file

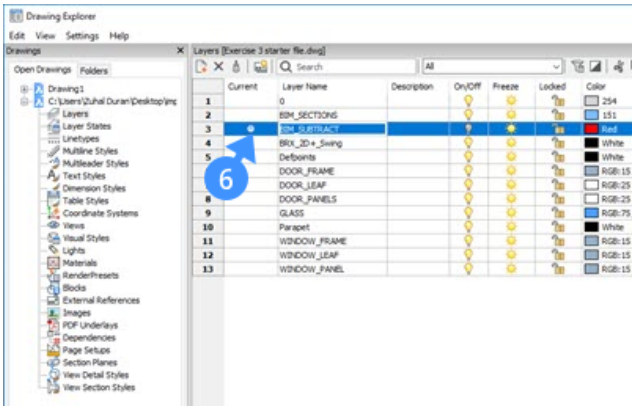
1 Click **New** in the **File** menu.

2 Select one of the pre-defined templates to create a simple window specifying the unit system of your drawing.

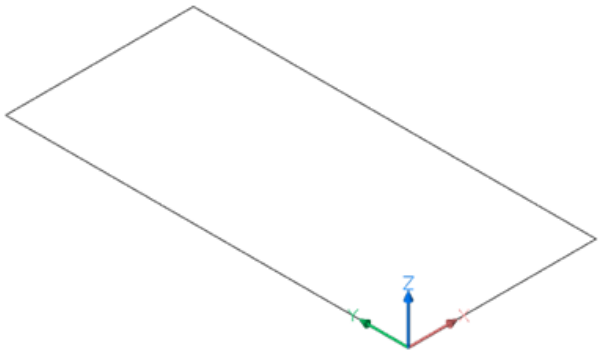
Uwaga: The pre-defined templates contain the layers needed to create the geometry of the window or door. These .dwt file format templates are **BIM-Window-imperial** (4) and **BIM-Window-metric** (5) which assure the required layers and properties are displayed in your drawing.



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.

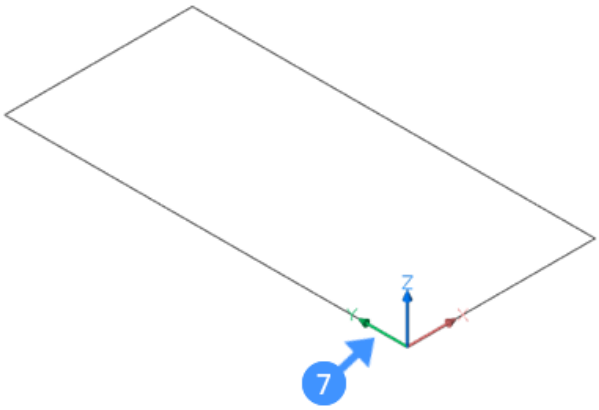
Step 2: Creating the subtractor solid

The subtractor solid will be used to create an opening.

Uwaga: **BIM Subtract** layer must be a current layer.

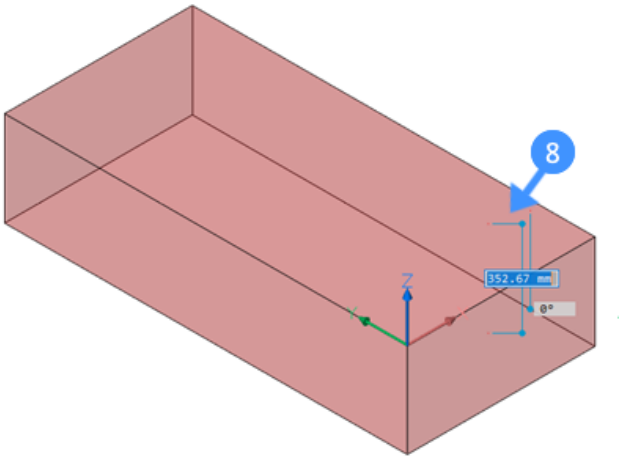
1 Use the outline of the solid which you have previously created (e.g. a rectangle).

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.

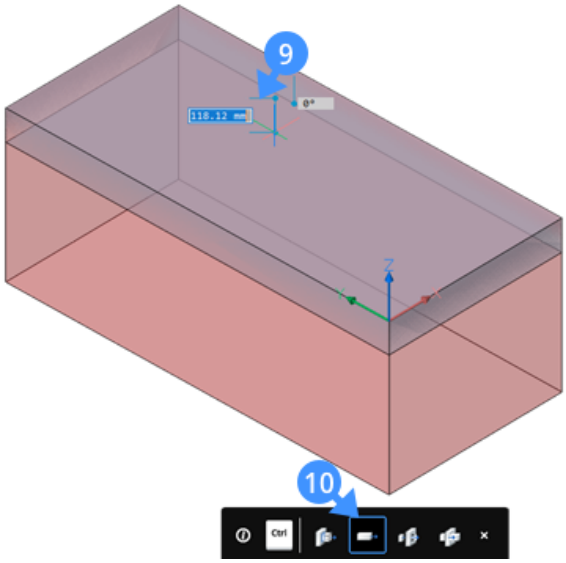
Step 3: Creating the fixed frame

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.

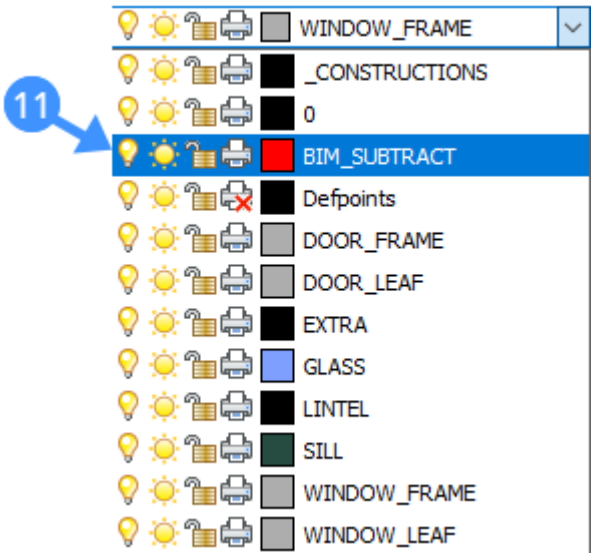
3 Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool 'Create' (10) option by pressing the CTRL key once.



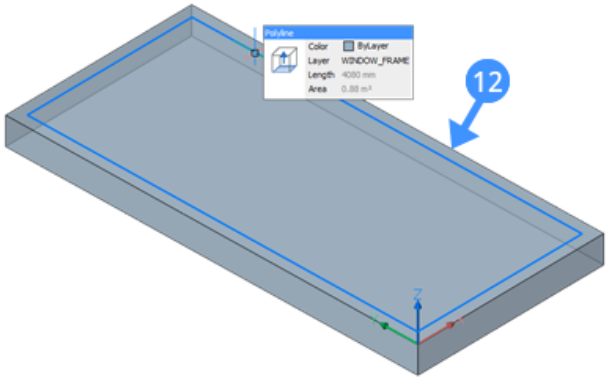
4 Type a value in the dynamic dimension field to specify the height of the fixed frame.

Uwaga: 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.

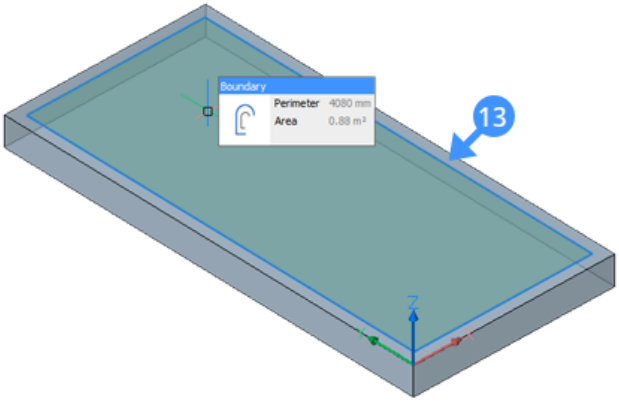


6 Use the **Offset** tool to create a parallel offset of the outline of the fixed frame (12).

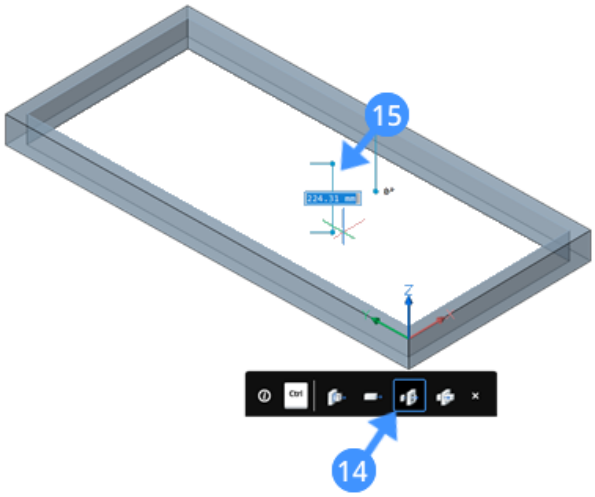


Uwaga: The distance of the offset equals the width of the frame.

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.

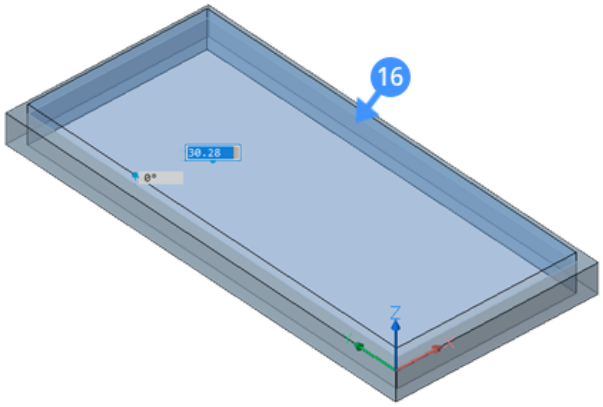
- Type a value in the dynamic dimension fields, then press Enter.

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.

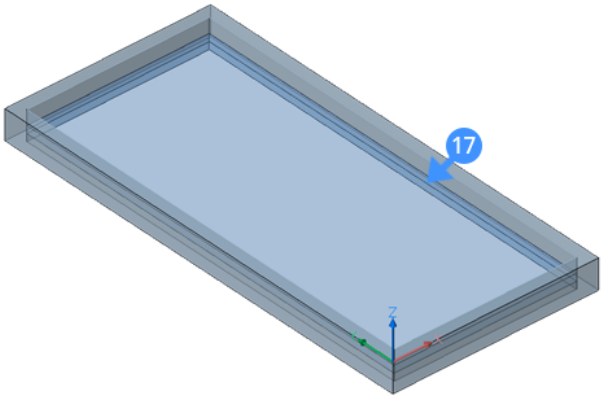
Uwaga: The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMExtrude to create a window glass pane.



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.

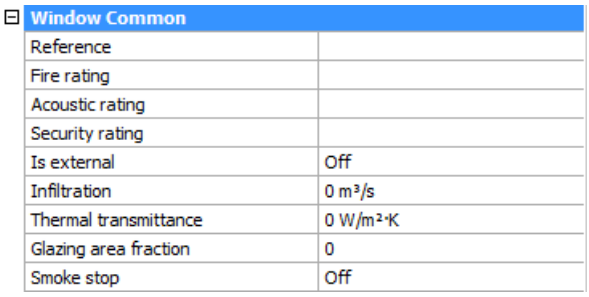
6 To do so, follow these steps:

- 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:

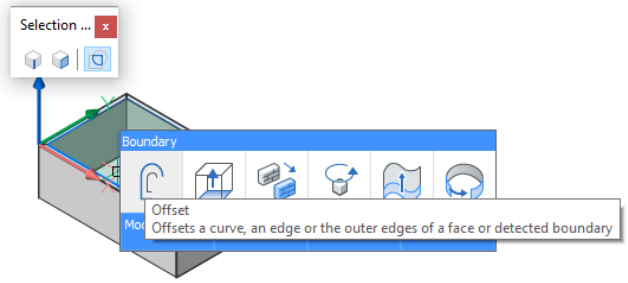
- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

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.

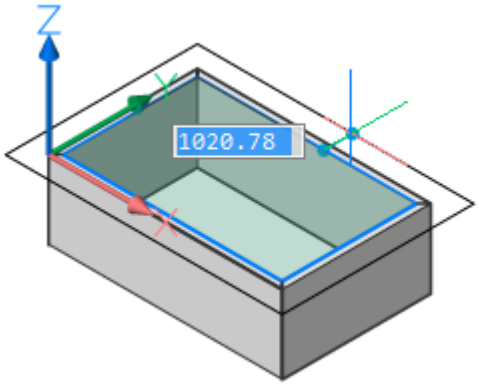
3 Set the **Enable boundary detection** ON in the **Selection Modes** toolbar or include the **Select detected boundaries** into SELECTIONMODES system variable.



4 Move the cursor to avoid area inside the exterior walls.

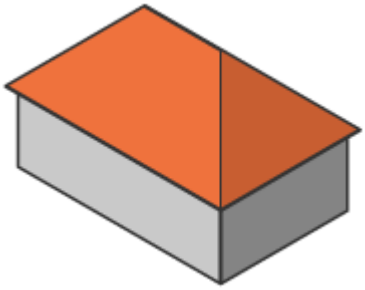
5 The inside boundary highlights.

6 Choose **Offset** in the **Draw** tab of the Quad.



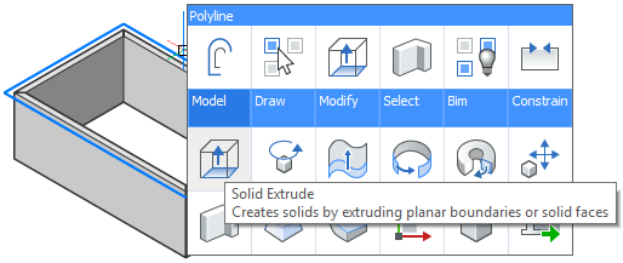
7 Do one of the following:
   - Type a value in the dynamic distance field.
   - Click to accept the value.

20.3.6.3 Creating a hip roof

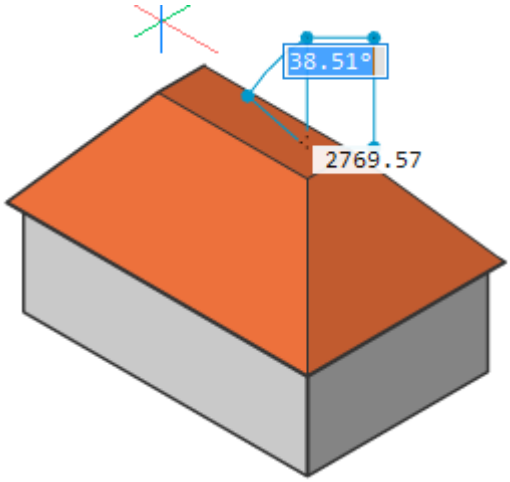


1 Create the roof footprint.

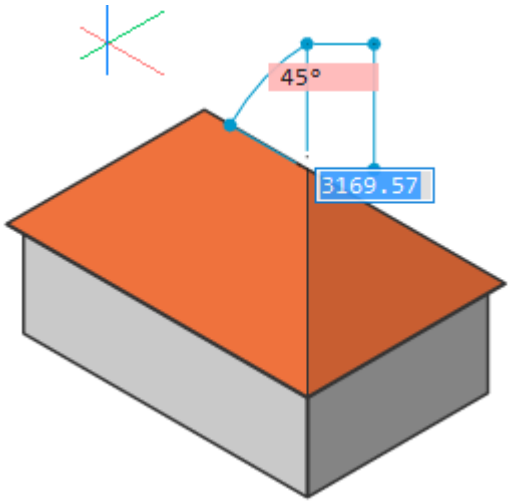
2 Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad.



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:

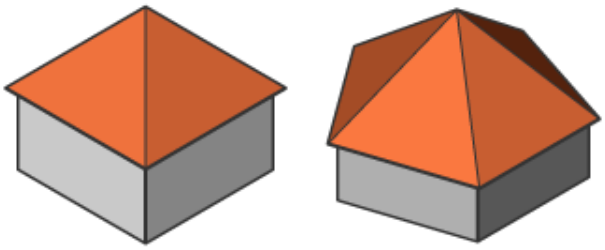
- Hit the Tab key and type another value in the **Taper Angle** field.

  The **Taper Angle field** is the complement of the inclination angle of the roof.

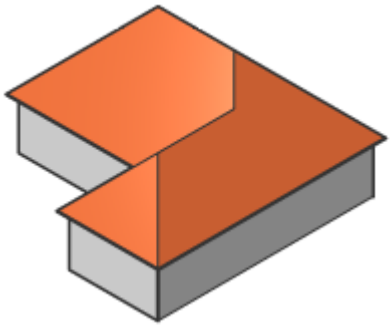
- Press Enter to create the roof.

6 Depending on the shape of the roof outline, the result is:

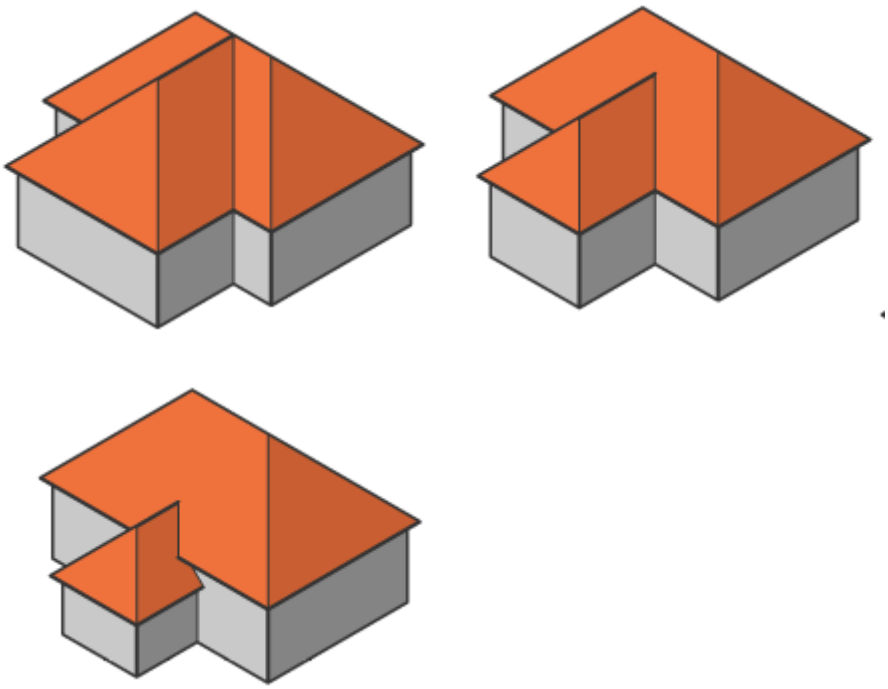
- Pyramid roof:



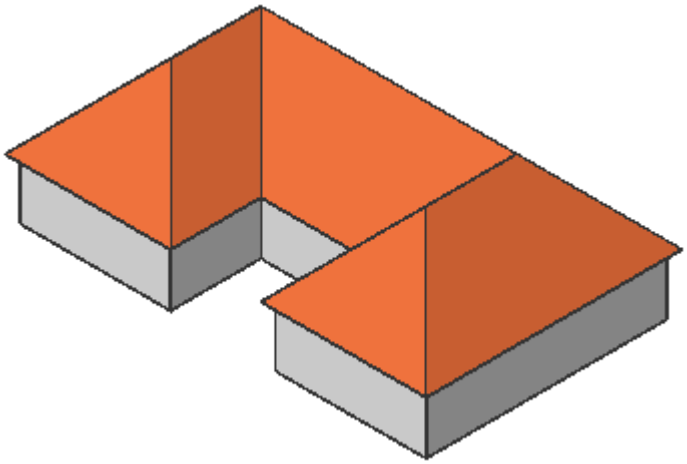
- L-shaped hip roof:

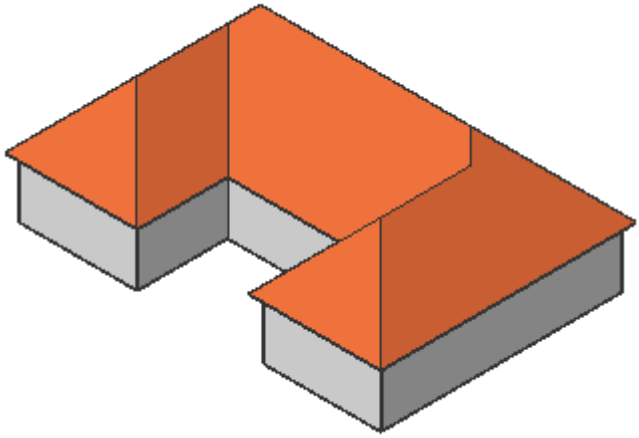


- T-shaped hip roof:

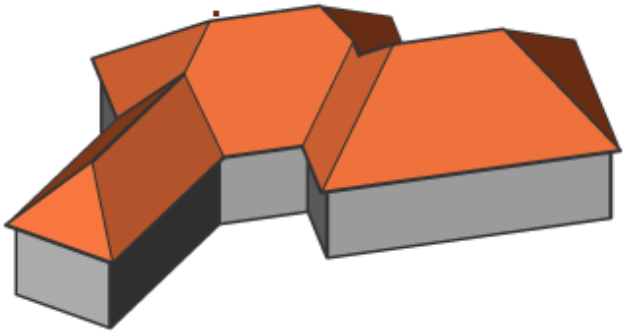


- U-shaped hip roof:





- 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.

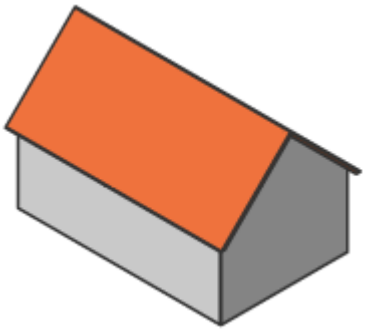
b You are prompted: Select 3D solid: Select the roof solid.

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.

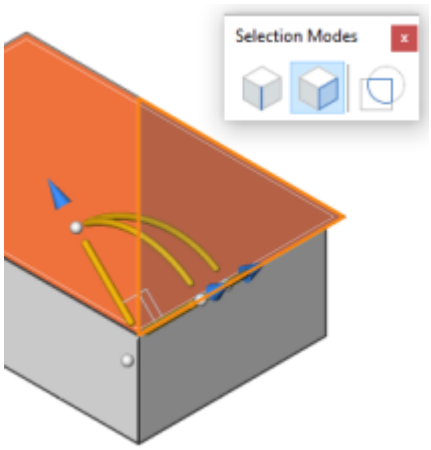
20.3.6.4 Creating a gable roof



1 Create a hip roof.

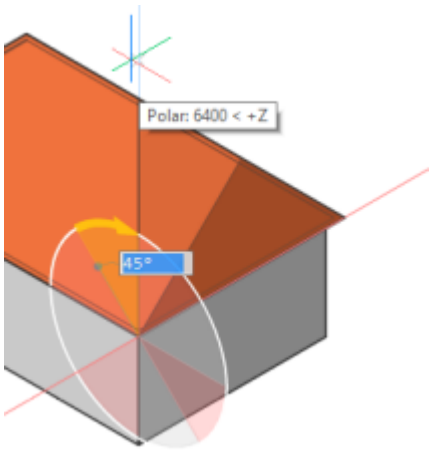
2 Make both end faces vertical:

- 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).



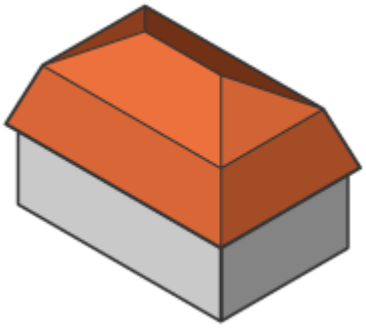
- Place the cursor over the vertical rotation arc. When the arc highlights, move the cursor.
- The selected face rotates dynamically.

  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.

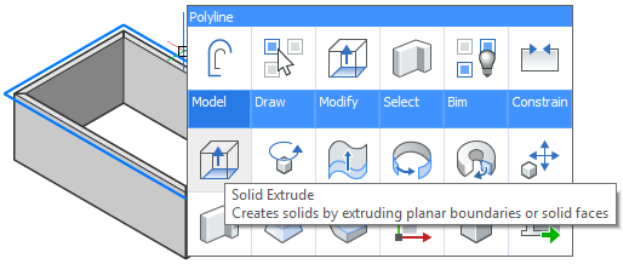
20.3.6.5 Creating a mansard roof



A mansard roof is composed of two solids, which we will create as two separate extrusions.

1 Create a roof footprint.

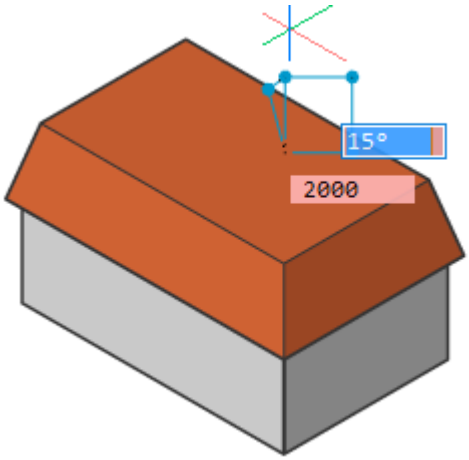
2 Move the cursor over the roof outline and choose to **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the Tab key.

4 Type a value in the **Taper Angle** field then hit the Tab key.

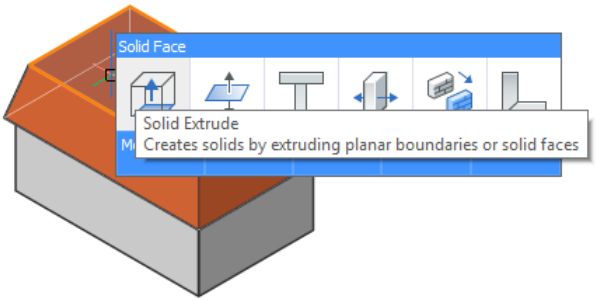
The **Taper Angle** field is the complement of the inclination angle of the roof.



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.

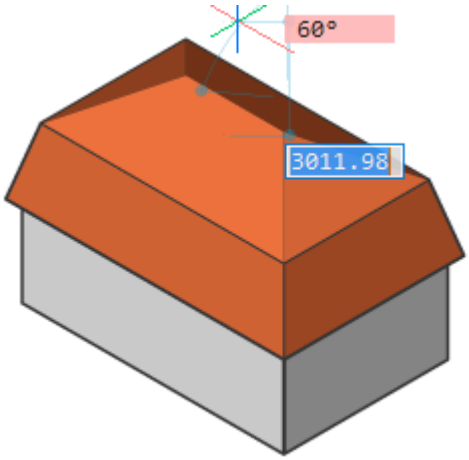
6 Optionally hit the Tab key to go back and forth between the entry fields and press Enter when you are satisfied with the result.

7 Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad.



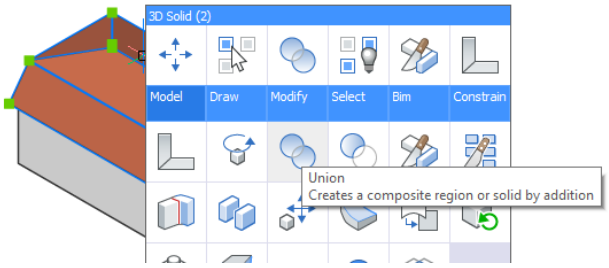
8 Move the cursor vertically, then hit the Tab key.

9 Type a value in the **Taper Angle** field then hit the Tab key.



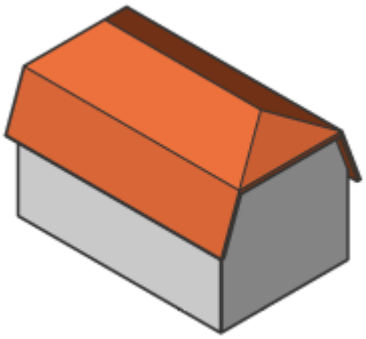
10 Do one of the following:
- Hit the Tab key and type another value in the **Taper Angle** field.
- Press Enter to create the first solid.

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.

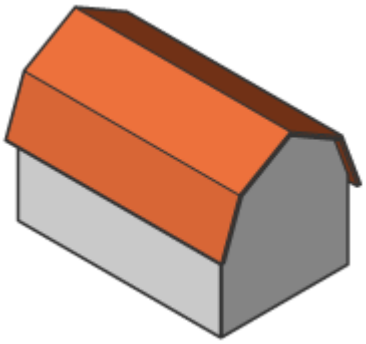
2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

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.

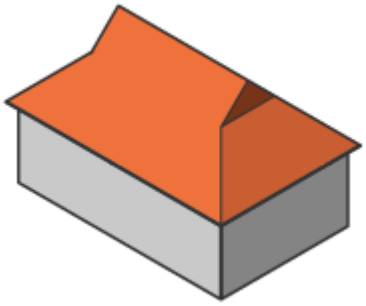
3 Execute steps 7 to 10 of Creating a mansard roof procedure to create the second solid.

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

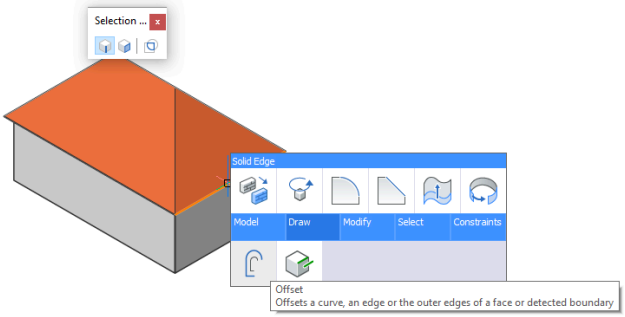


A Dutch gable roof is based on a hip roof.

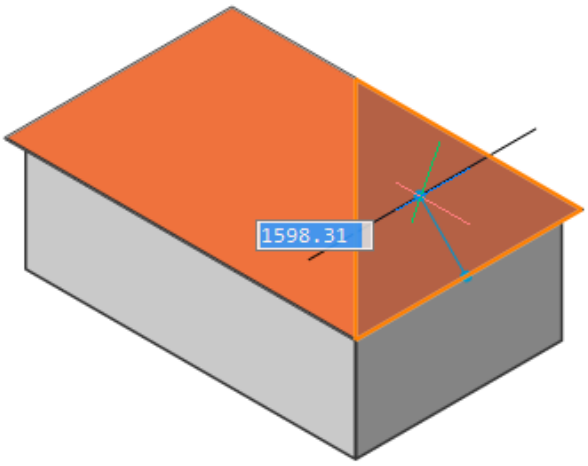
1 Create a hip roof.

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.

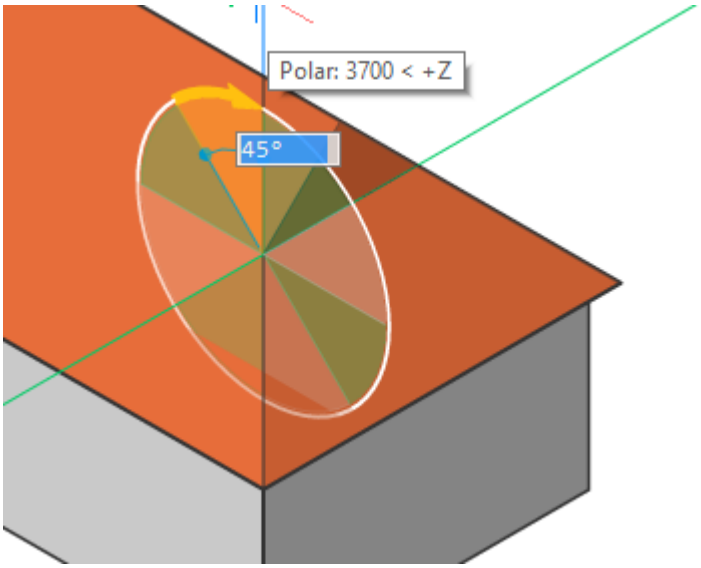
6 You are prompted: Select 3D solid, surface or region: Select the roof solid.

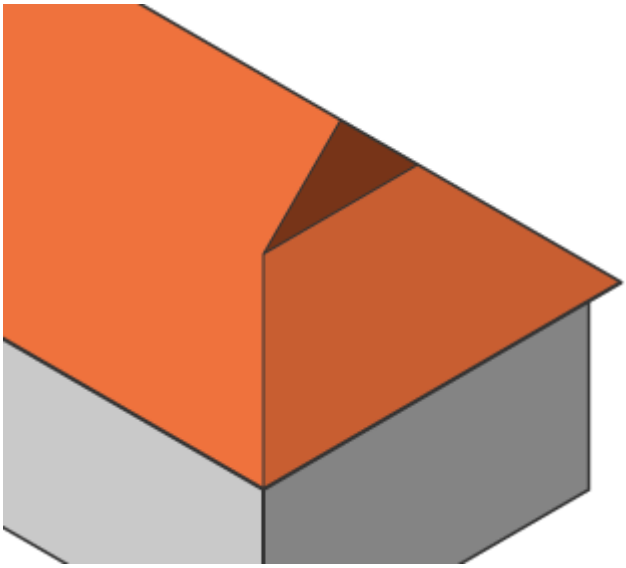
7 You are prompted: Select an entity to imprint: Select the offset roof edge.

8 You are prompted: Delete the source object[Yes/No] <No>: Choose Yes.

9 A triangular face is created.

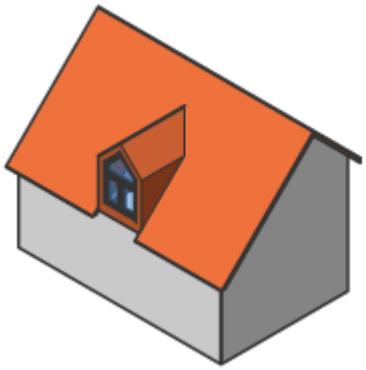
10 Execute step 2 of Creating a gable roof procedure to set the triangular face vertical.





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.

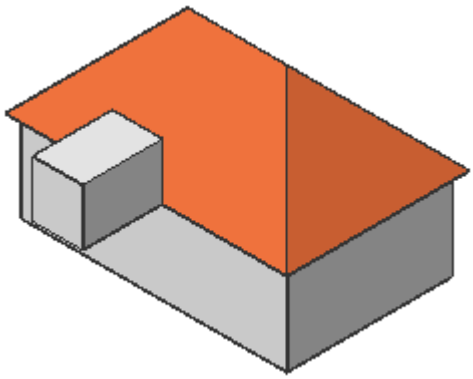
1 Right-click, then choose **Box** in the **Model** tab of the Quad.

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.

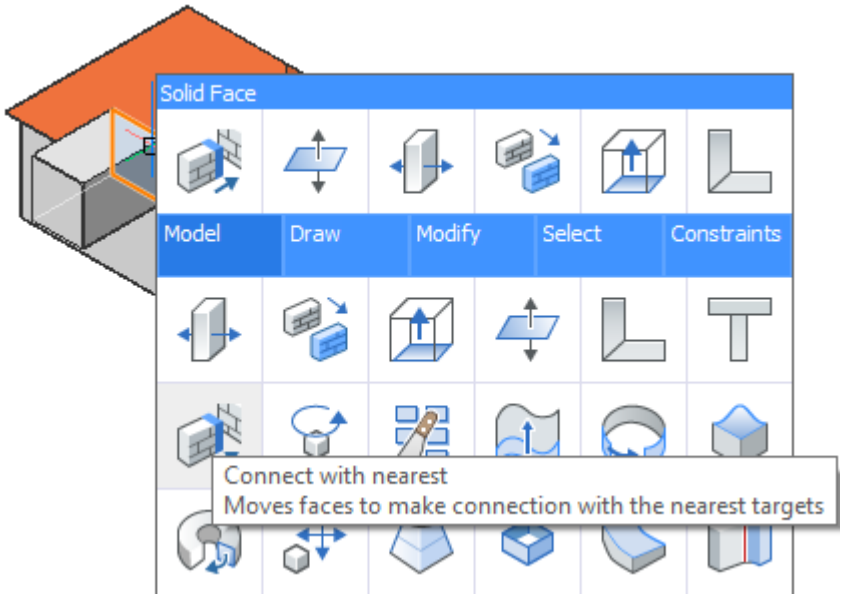
3 You are prompted: Set opposite corner or [Cube/Length of side]:

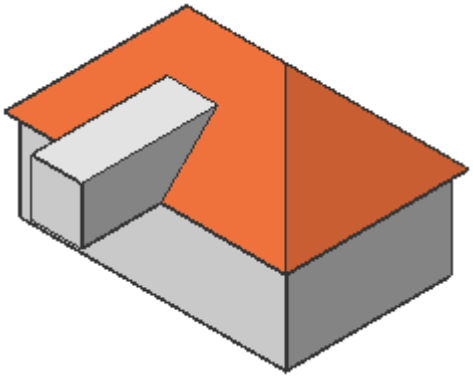
4 Click a point, then define the height of the cube: either click a point or type value.



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.



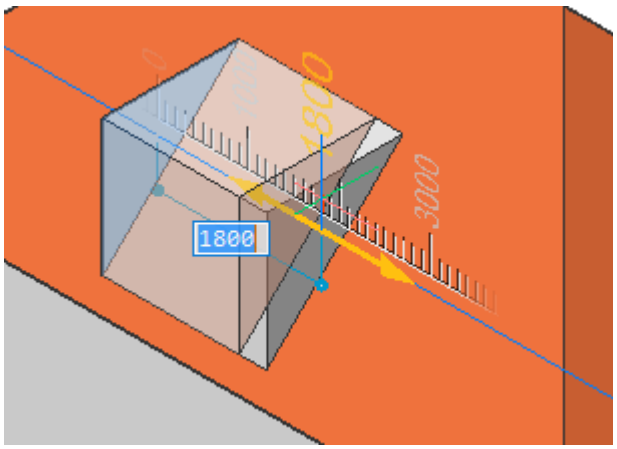


7 Select the front face of the box, choose **Push/Pull** in the **Model** tab in the Quad and move the face to the roof plane.

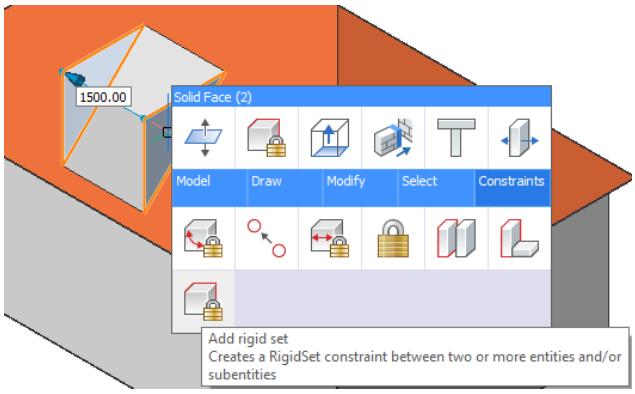
8 Use PUSHPULL command on the front, top and side face of the dormer solid to define the dimensions

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.



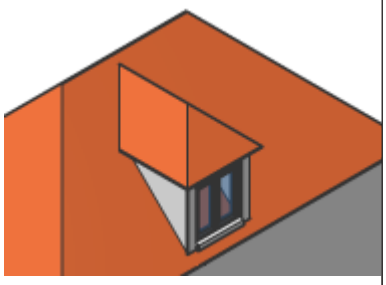
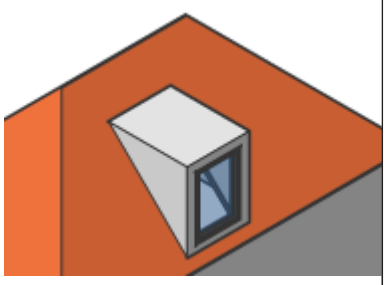
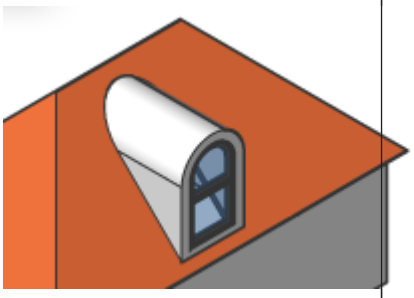
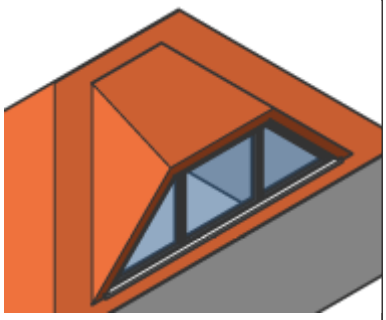
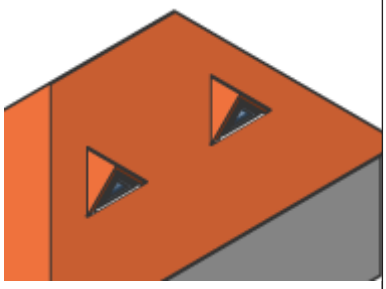
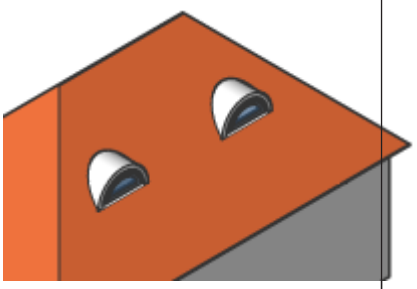
- Drag one of the side faces to position the dormer. Do one of the following:
  - Type a distance in the dynamic entry field.
  - Click to accept the value.
  - Move the cursor over a face that is parallel to the face being dragged and hit the Tab key. The distance field now shows the distance to this 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

Uwaga:

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.

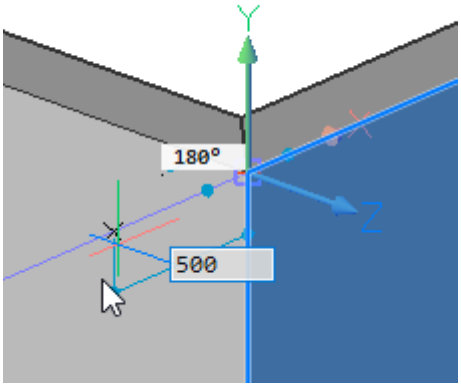
3 When the face highlights, hit the Shift key to lock the **dynamic UCS**.

4 Do one of the following:

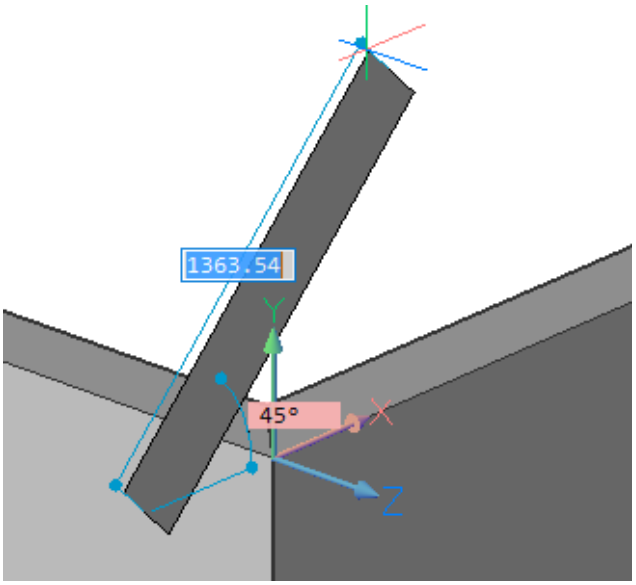
5 To create the roof at the top edge of the wall, snap to the exterior top corner of the wall.

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.



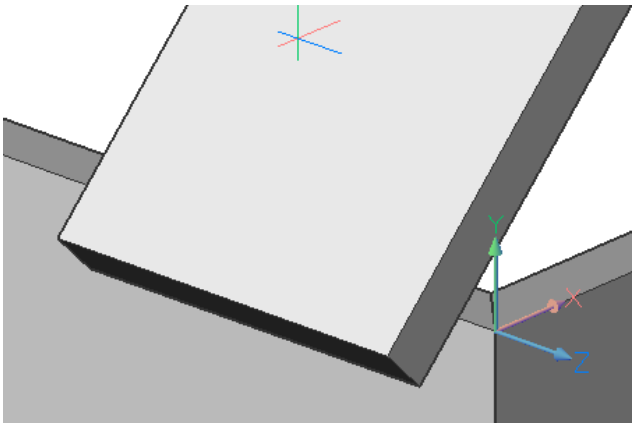
6 Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.



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.

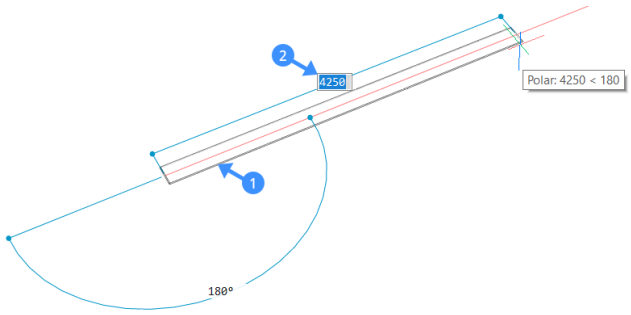
For more information about this command, visit the Command Reference article POLYSOLID.

20.3.8.3 Procedure: Create a simple wall

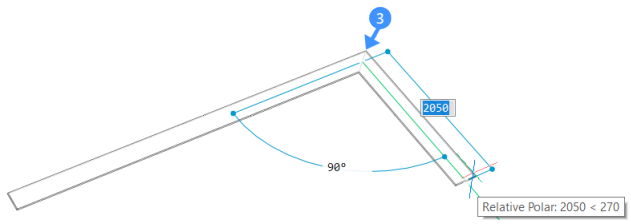
1 Launch the **Polysolid** tool.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

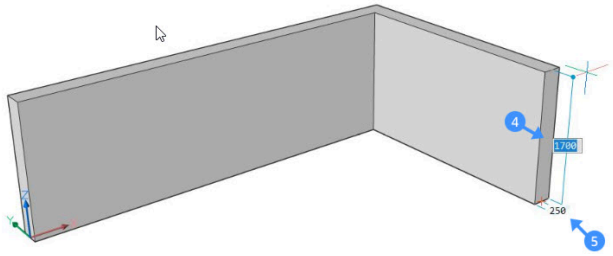
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



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.



Uwaga: The width and height of the previous wall will be the new default values of the next Polysolid.

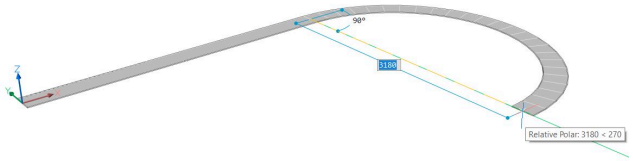
20.3.8.4 Procedure: creating a curved wall

1 Launch the **Polysolid** tool.

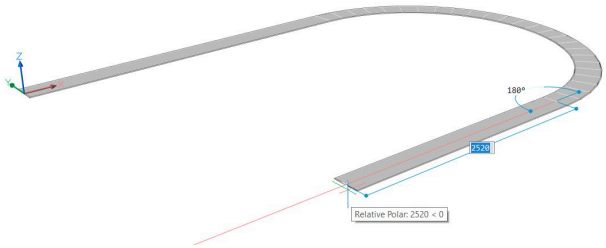
2 Select a start point and move the cursor to set the length or click to set the endpoint.

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.

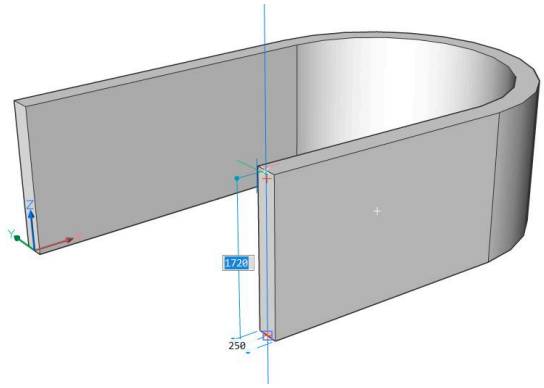


5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.



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.

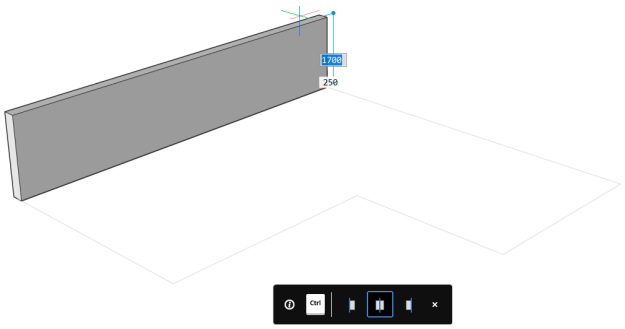
2 Choose a 2D linear entity which will be the Polysolid base.

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.

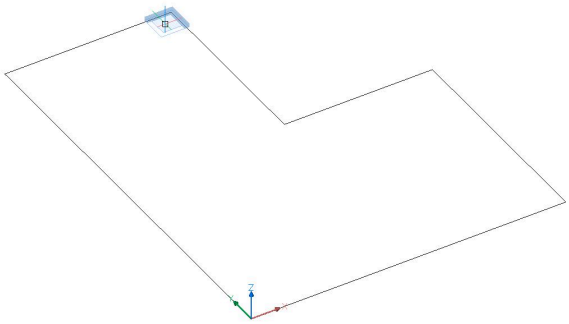
6 The Hotkey Assistant appears and displays the possible justification options. Press CTRL to cycle through the wall justification options.



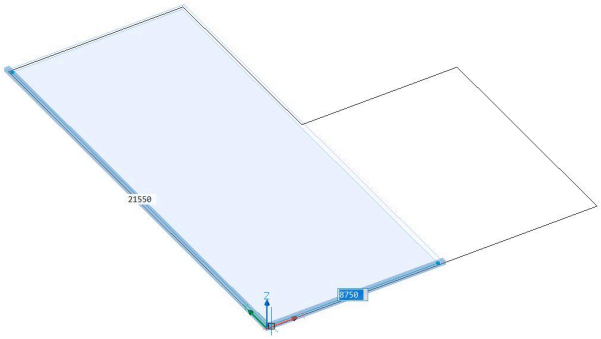
It is also possible to use an existing 2D plan to create walls using the **BimQuickDraw** tool:

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.



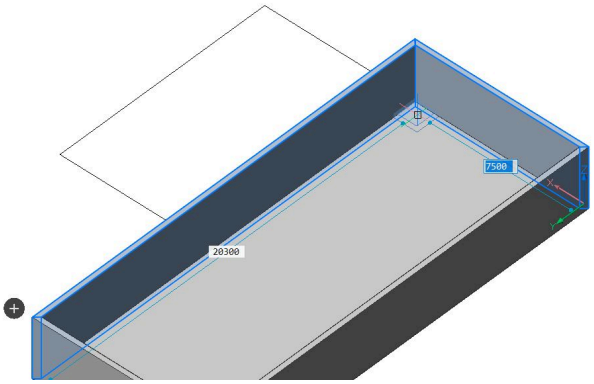
3 Hover your mouse to the opposite corner and left click again or type the values of the lengths of the sides in the respective fields and press Enter.



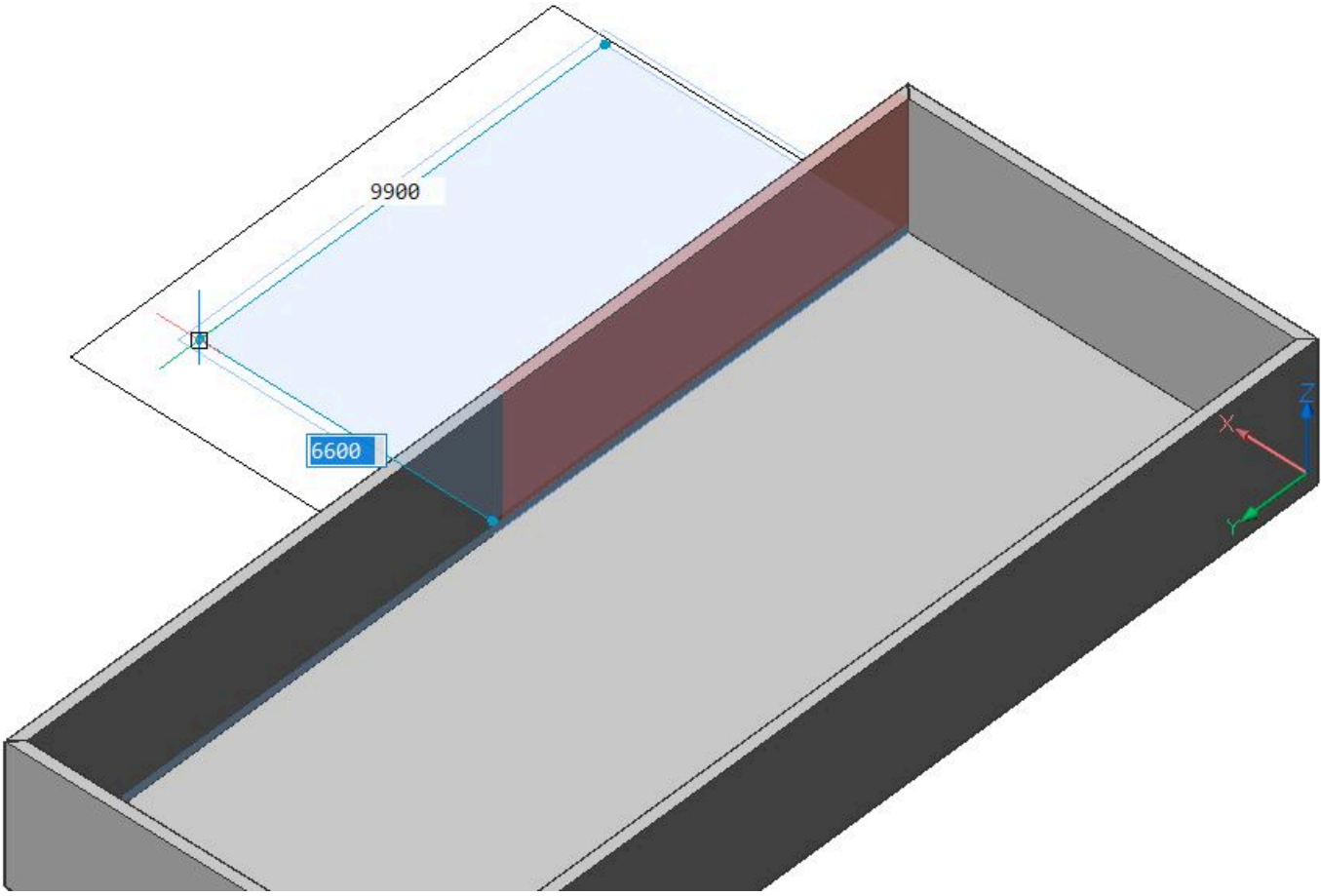
4 The **Quickdraw** tool always uses the default height and width settings.

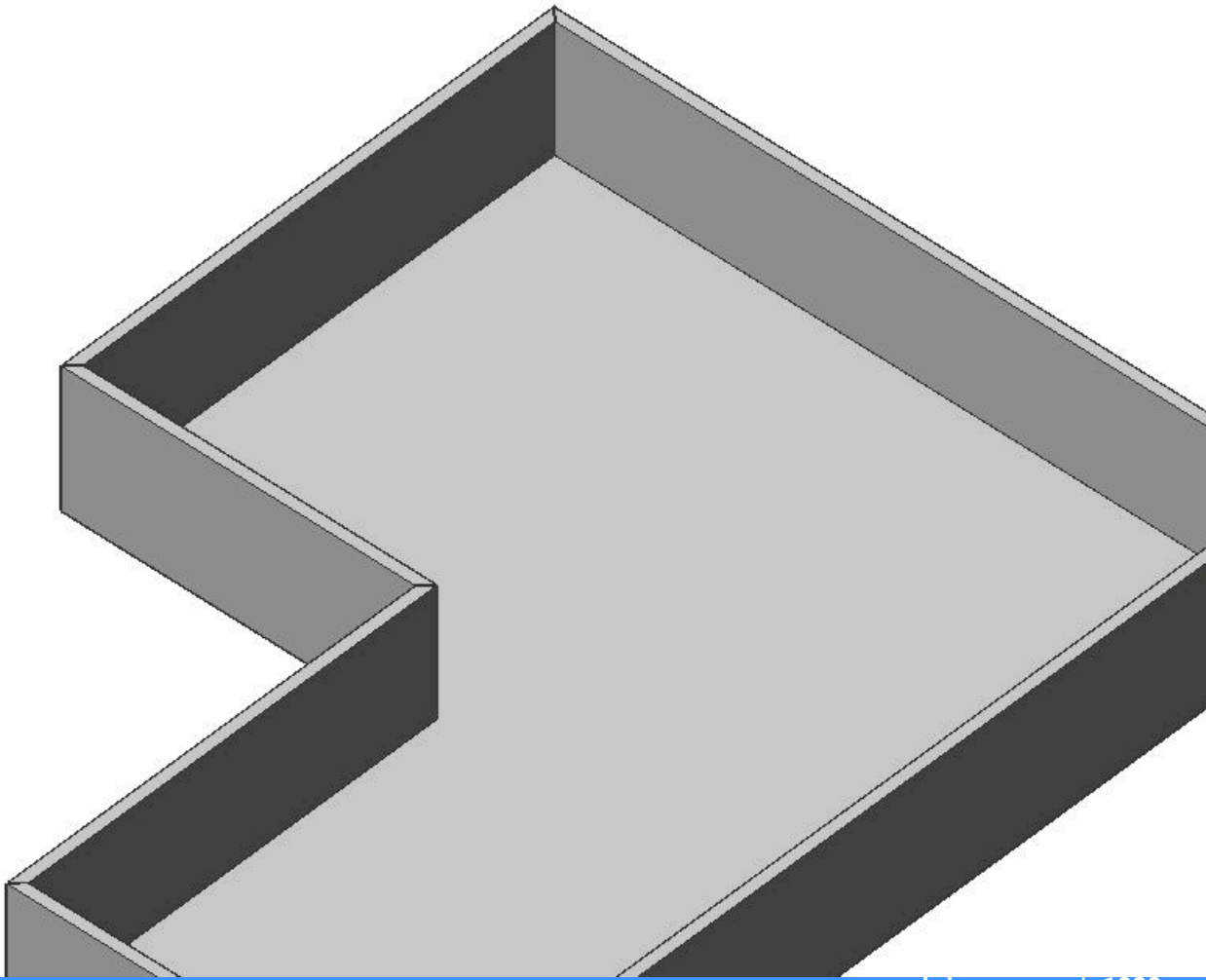
To change these values type S and press Enter to open the Quickdraw Settings. Change the values in this menu.

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.

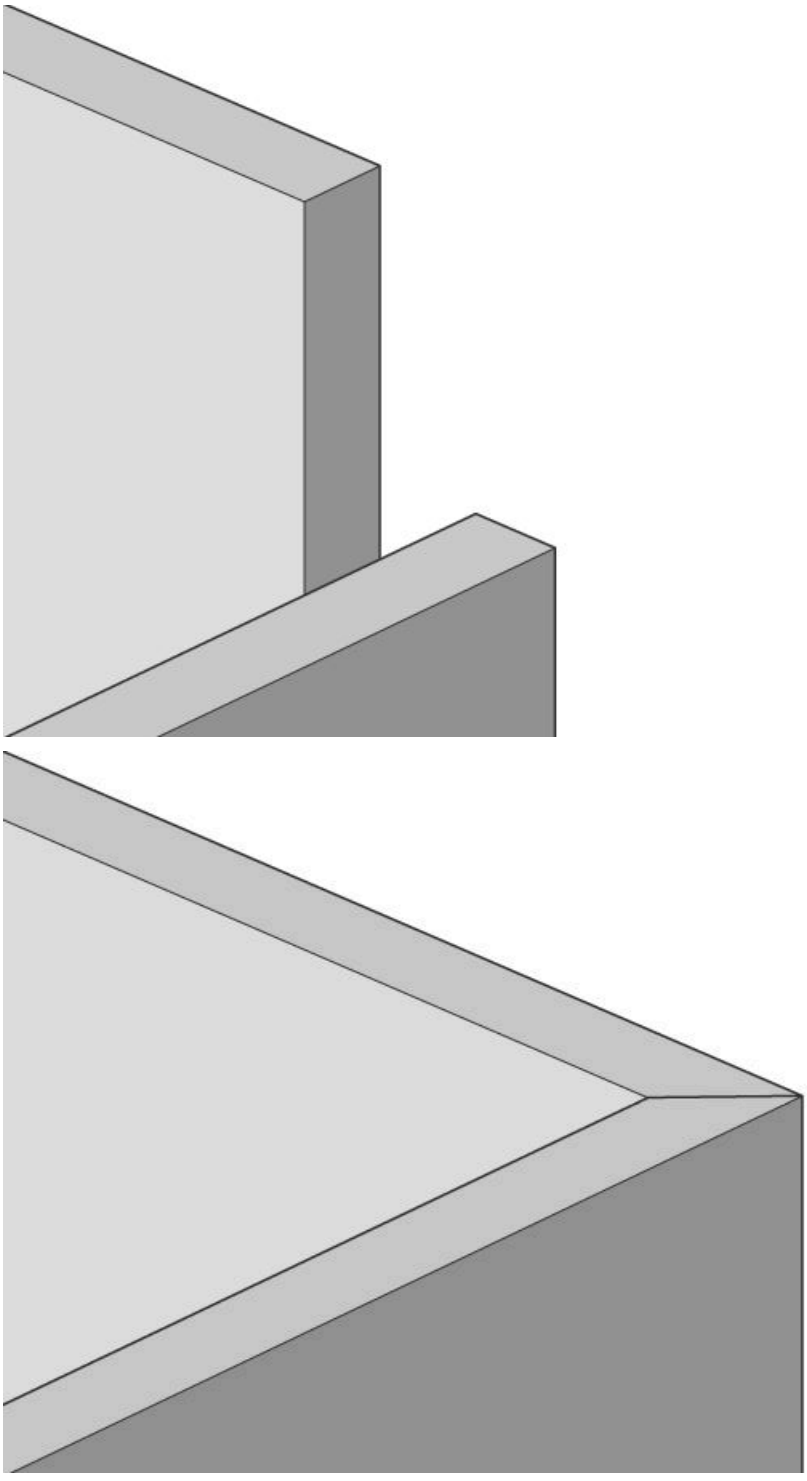


For more information about this command, visit the Command Reference article LCONNECT.

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’.

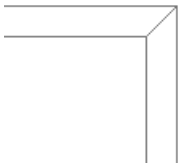
3 The selected connection type is indicated by a blue frame.



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.

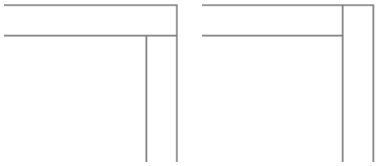
- **Bisector L-connection (miter)**

  When the **bisector L-connection** is selected, the walls are connected at an angle of 45°.



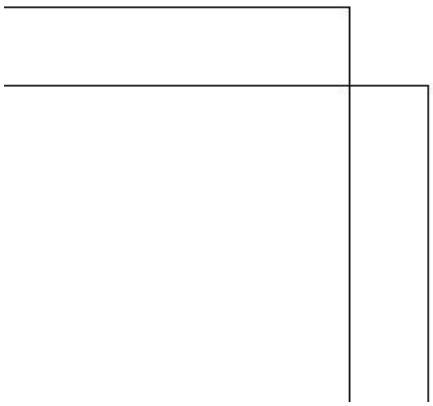
- **Parallel L-connection 1 Parallel L-connection 2**

  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.

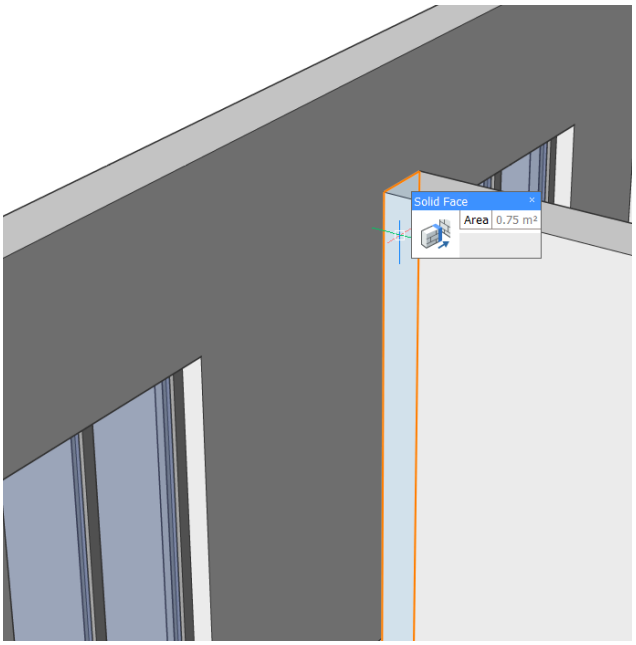
Uwaga: Before using the **Connect with Nearest** tool, make sure the **Select Faces** option is enabled in the selection mode settings.



For more information about this command, visit the Command Reference article TCONNECT.

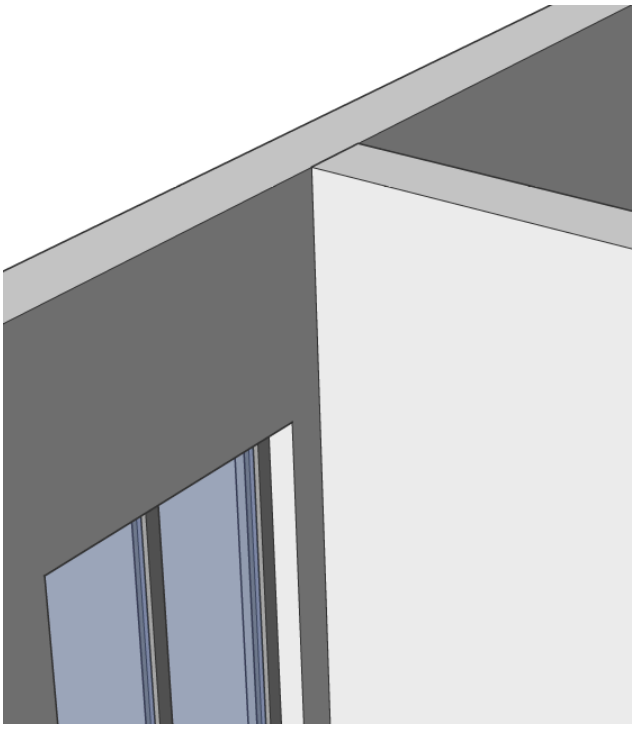
20.3.8.9 Procedure: creating T- connections between two walls

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.

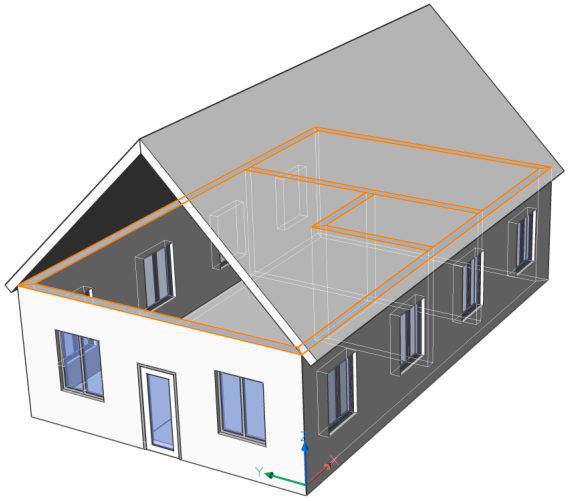


For more information about this command, visit the Command Reference article SELECTALIGNEDFACES.

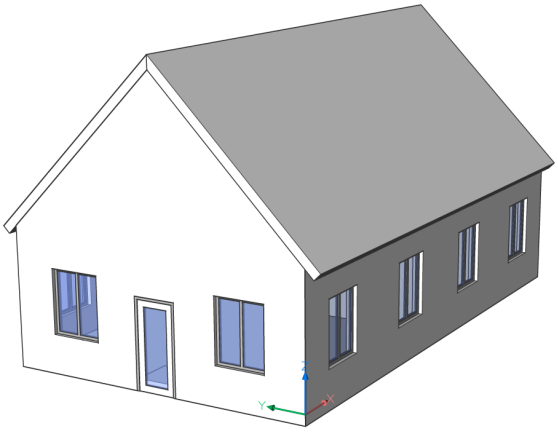
20.3.8.11 Procedure: connecting multiple faces of the wall to the roof slab

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

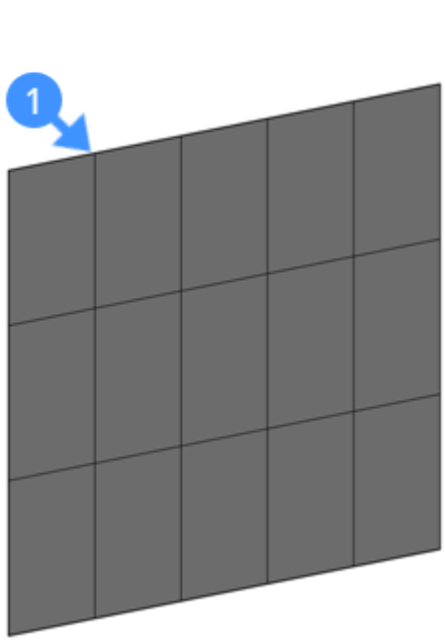
20.3.9.1 Commands

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.



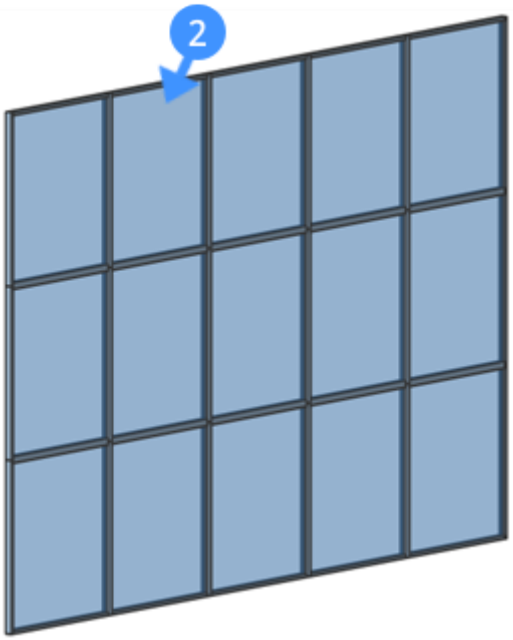


Figure 1, Curtainwall grid (1).

Figure 2, Curtainwall with glazing (2).

The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall.

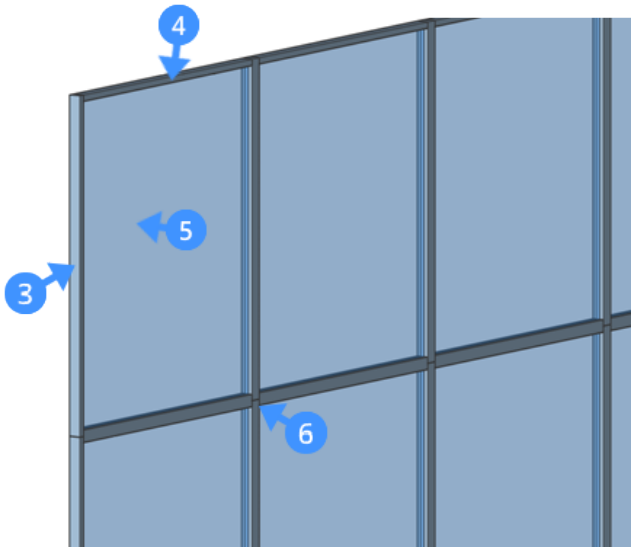
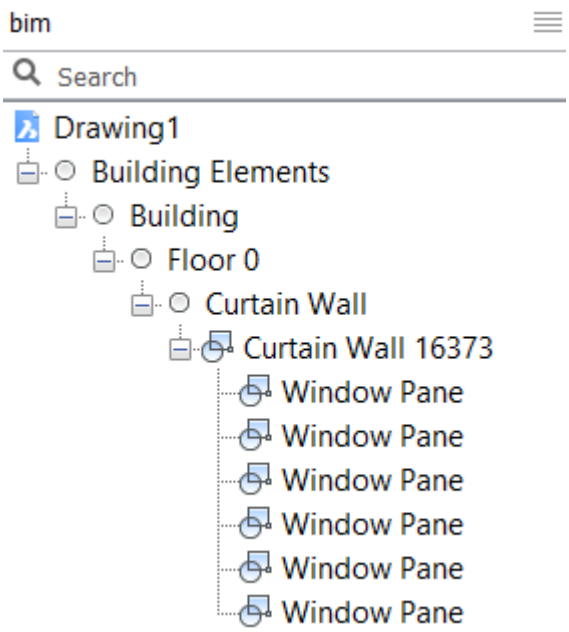


Figure 3, A curtain wall. The frame that coincides with the grid vertical (U) panels (3), the frame that coincides with the grid horizontal (V) panels (4), and glass in-fill (5) are shown. By default, the straight connection (6) is made between the beams.

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.

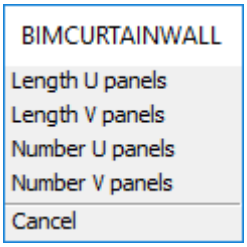
1 You are prompted: Select a face:

2 Select a face in the drawing area to create a curtain wall.

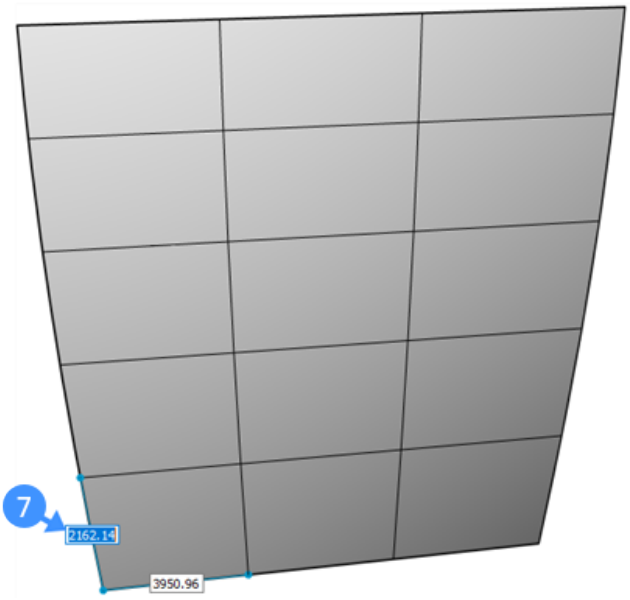
3 The curtain wall grid is automatically generated on this face.

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.

Uwaga: 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.

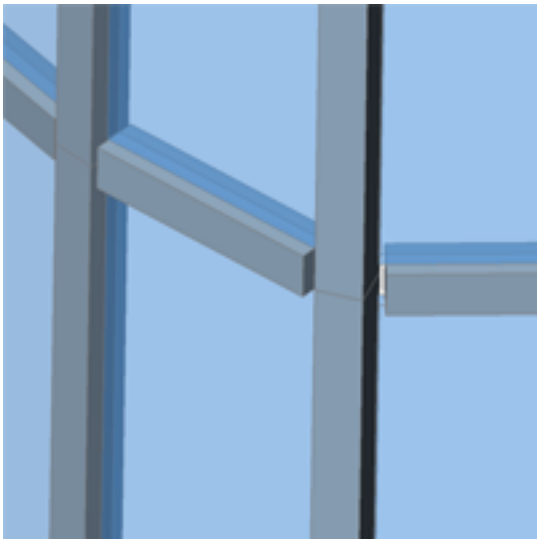
Default frame depth = 60.000000

(**Optional**) To change the glass thickness, select the **Glass thickness** option in the prompt menu or type "G" in the Command line.

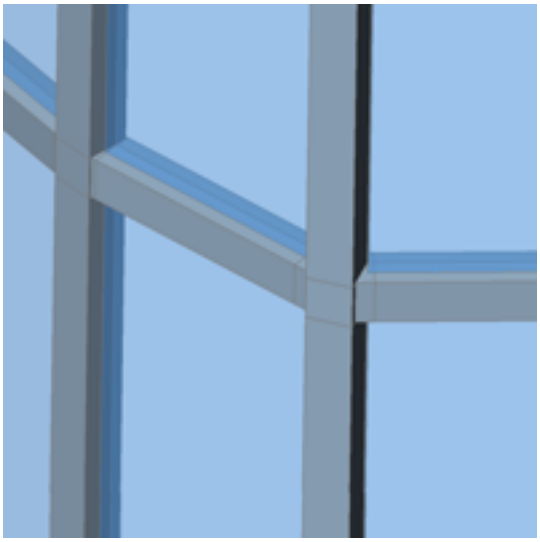
Default glass thickness = 20.000000

(**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.

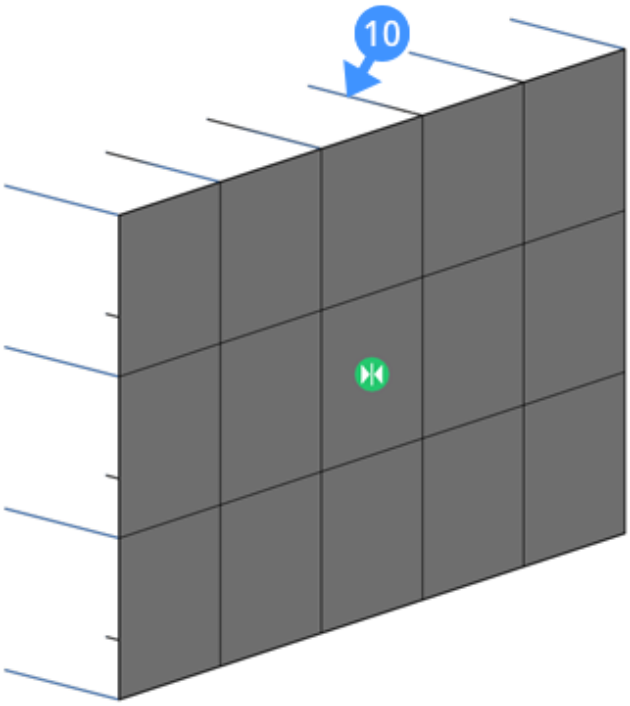






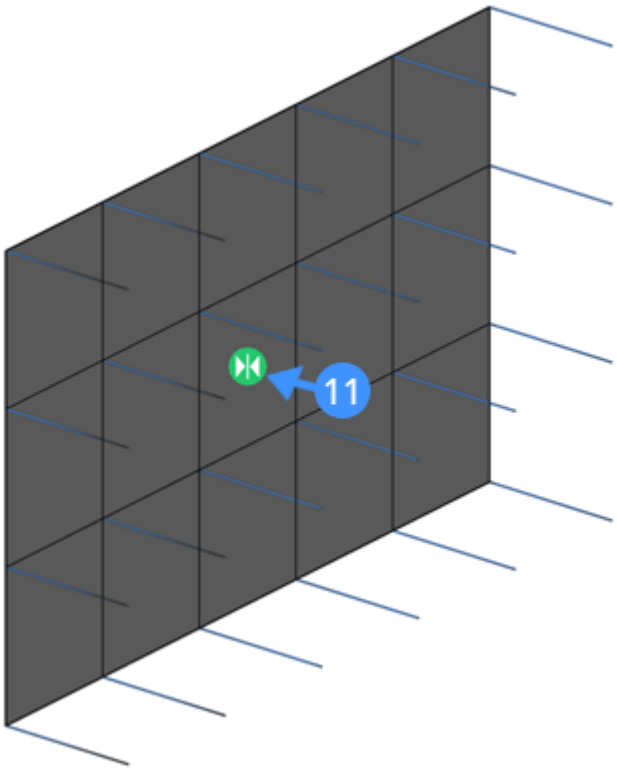
There are three types of connection. From left to right: Straight - Smooth - Nodes.

9 Blue lines (10) appear on the face to show the side a curtain wall will be created.



Uwaga: 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.

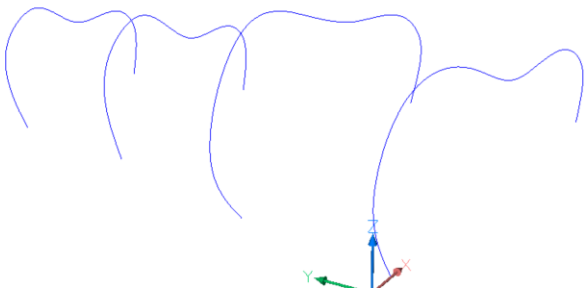


11 Press Enter to accept.

Uwaga: 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

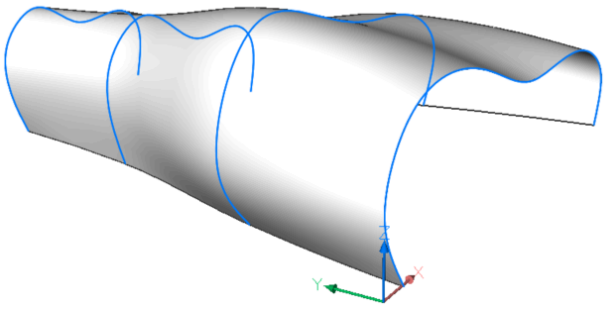
1 Start with 4 splines.



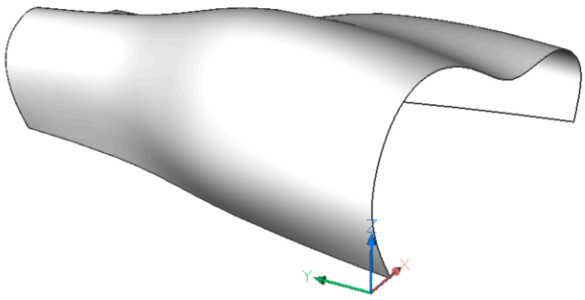
2 Launch the LOFT command in the Command line.

3 You are prompted: Select cross-sections in lofting order [MOde]:

Select the 4 splines in lofting order.



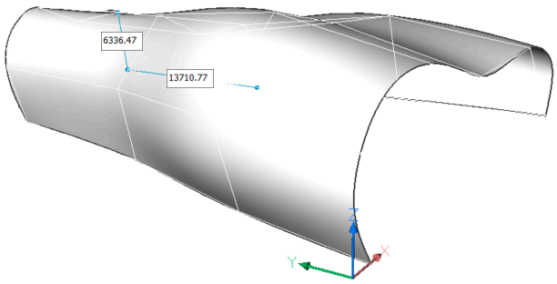
4 Press Enter twice to end the command.



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".

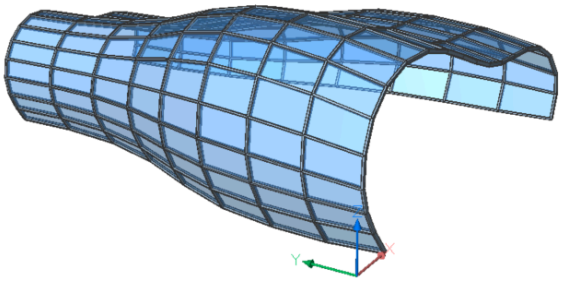
Press Enter to accept it.

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

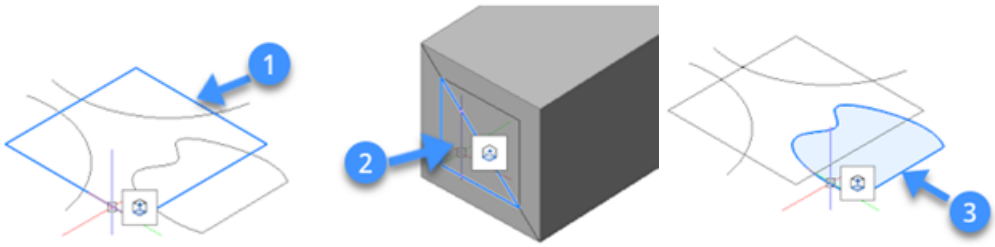
The **Extrude** tool allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The **Extrude** tool enables you to cut the solid model partially by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model.



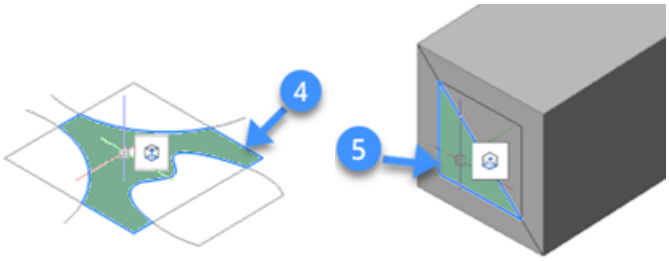
For more information about this command, visit the Command Reference article DMEXTRUDE.

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.

Uwaga: 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.

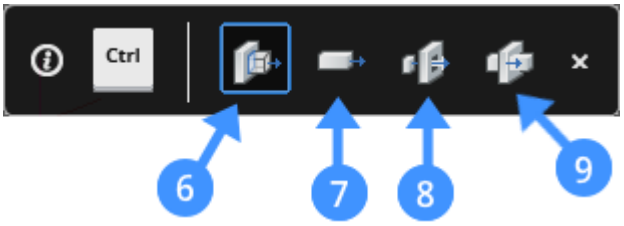
1 Hover the cursor over an entity or sub-entity in the drawing area.

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>.

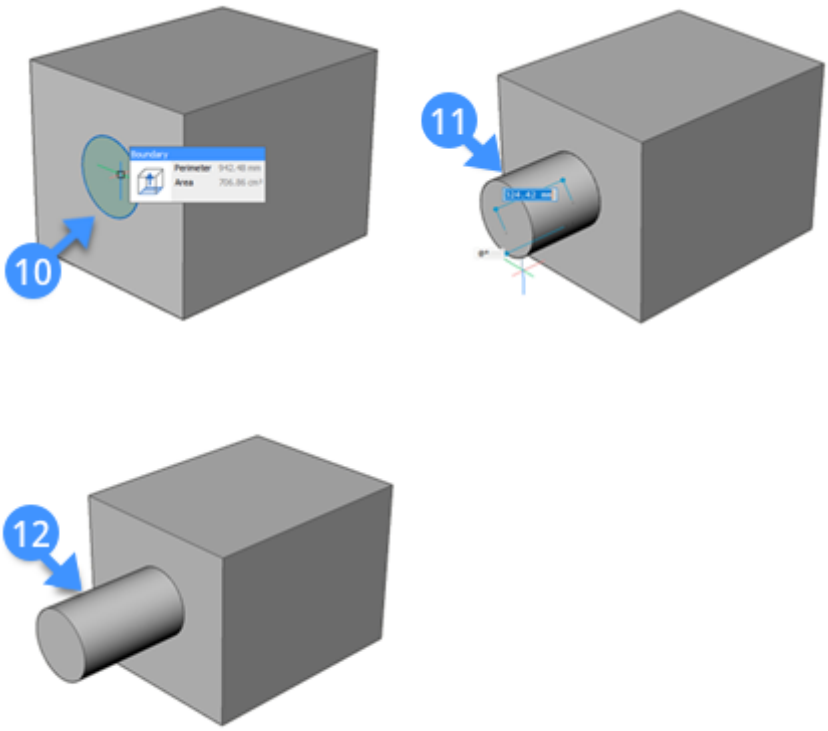
3 The “DmExtrude Hotkey Assistant” appears at the bottom of the screen with a set of extrusion options.

Uwaga: The default mode is indicated by a blue frame.

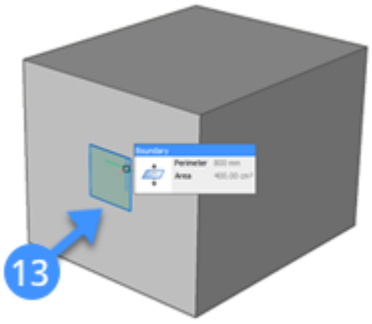


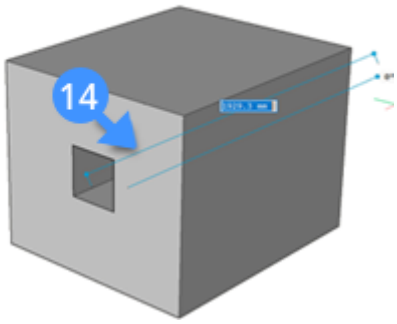
- **Auto (6)**: The result of the auto option depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable value. Note: By default, the value of the DMEXTRUDEMODE is set to “3”. The following illustration shows the extruded entity from the main solid while the default value is set to “3”. When you highlight an entity (10) and extrude it

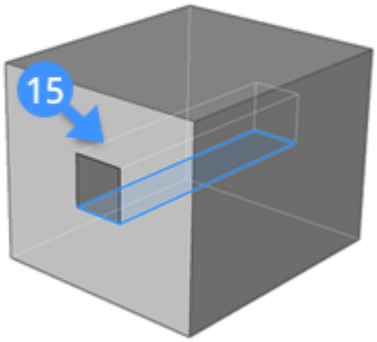
to the outward (11), a new volume is added (12) to the main solid model.



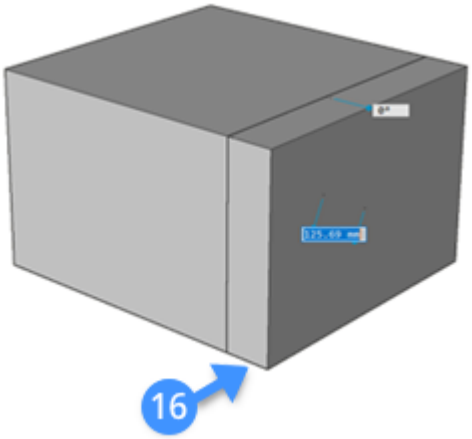
The following illustration shows the subtracted entity from the main solid. When you highlight an entity (13) and extrude it to the inward (14), a volume is subtracted (15) from the main solid.





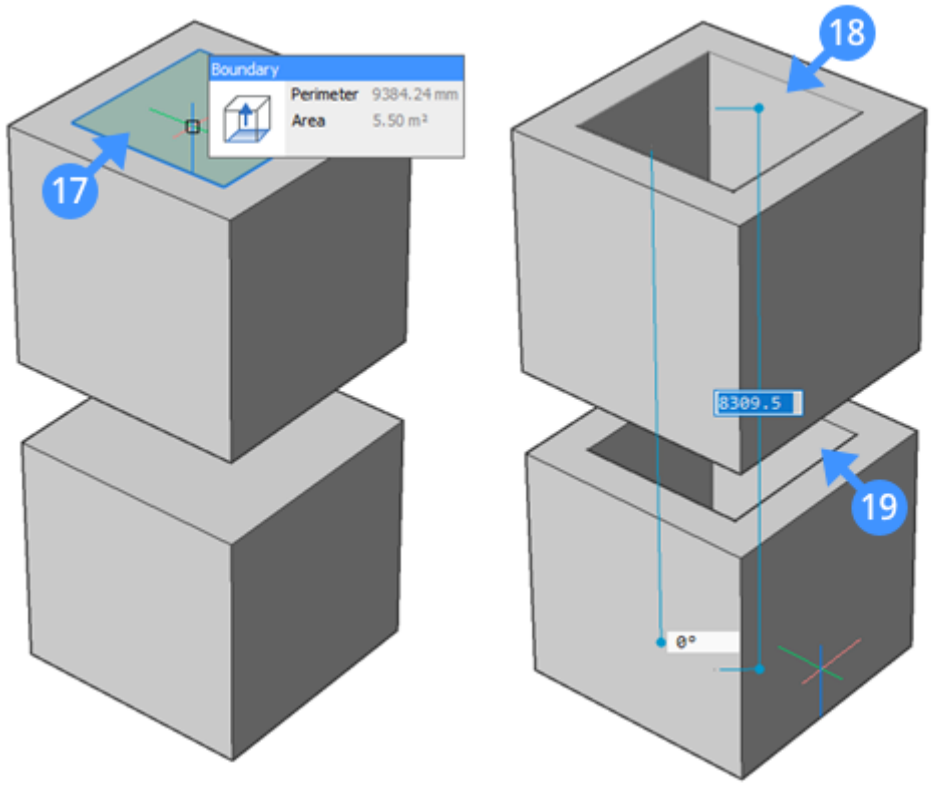


- **Create (7)**: Regardless of the extrusion direction, a new volume is created (16) from the existing solid.



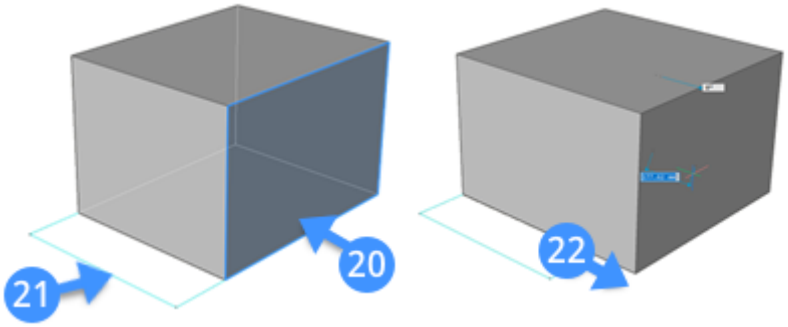
- **Subtract (8):** The 3D solid is subtracted from each interfering existing solid.

  The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17).

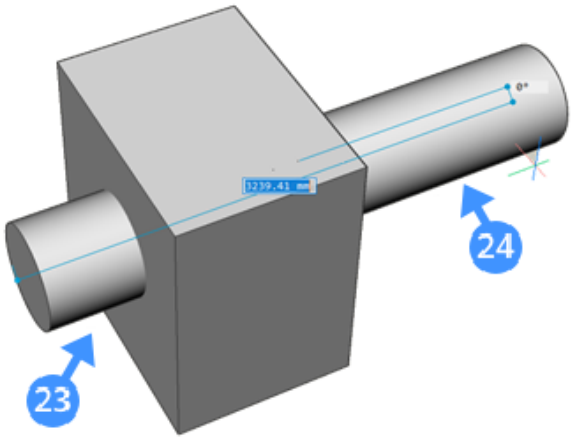


- **Unite (9):** The new volume is unified with each interfering existing solid.

  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

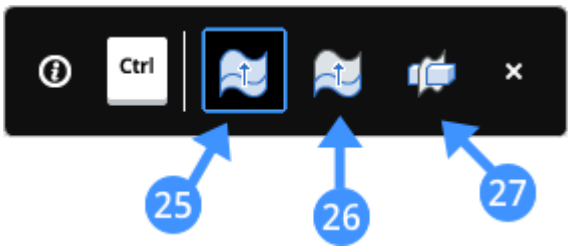
The following are general steps to create 3D surface(s) using the **Extrude** tool in the Surface mode.

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.



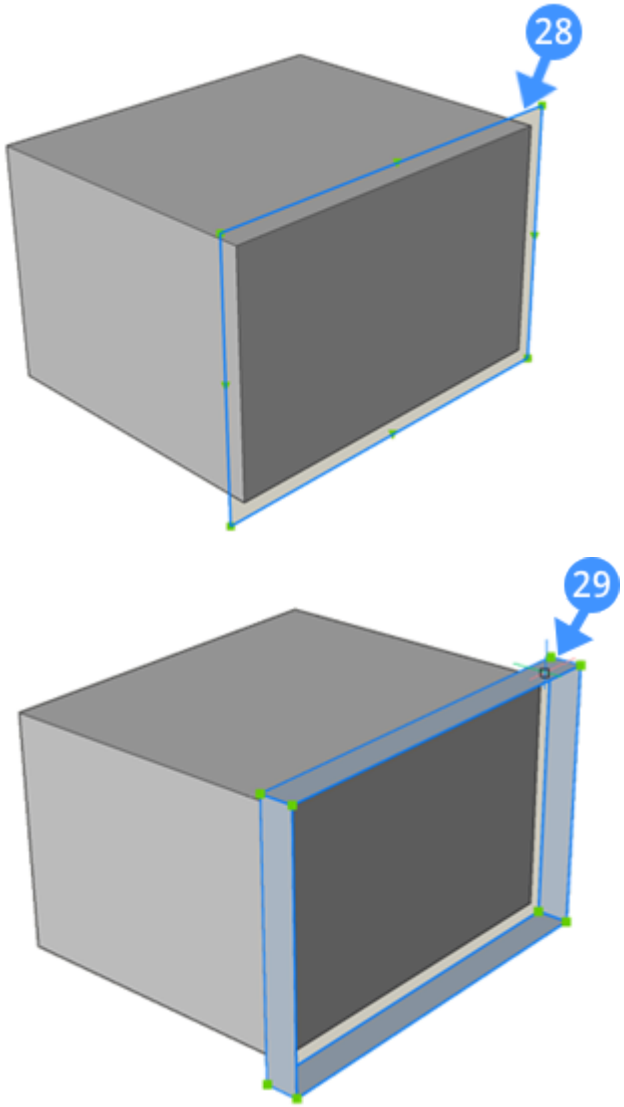
Uwaga: The default mode is indicated by a blue frame.

- **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.

Uwaga: **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

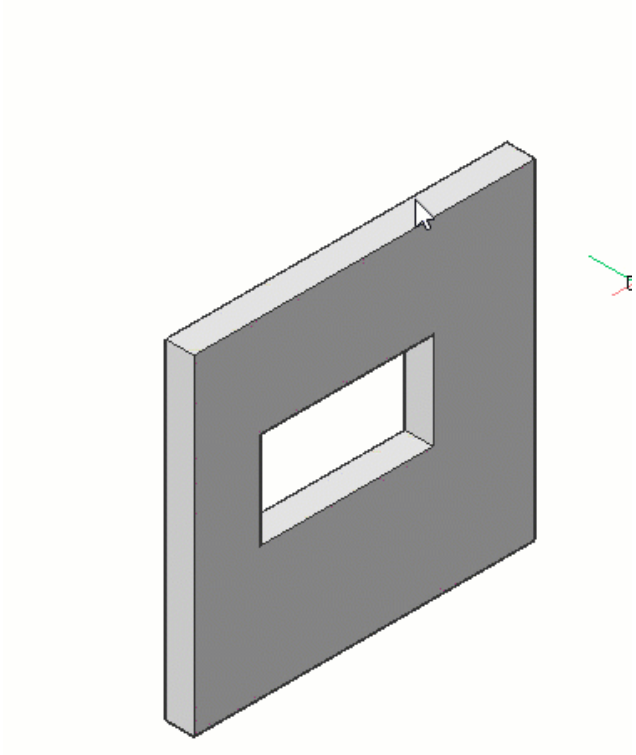
The following are general steps to add or remove volume from a solid(s) using the **Push/Pull** tool.

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

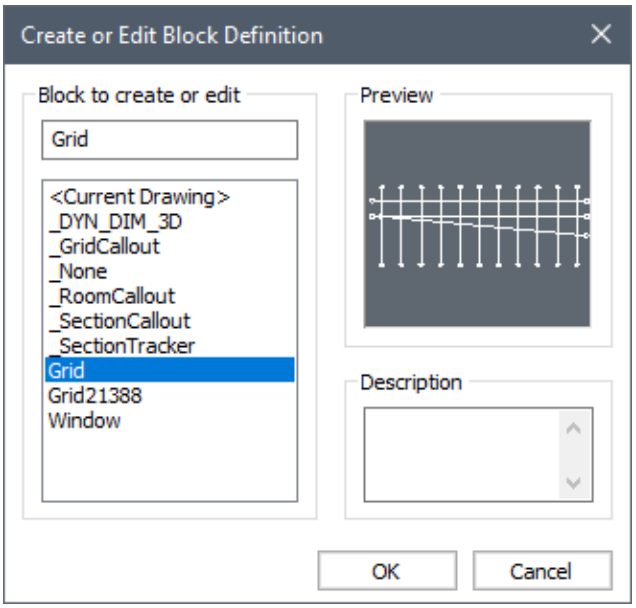
20.3.12.2 About

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

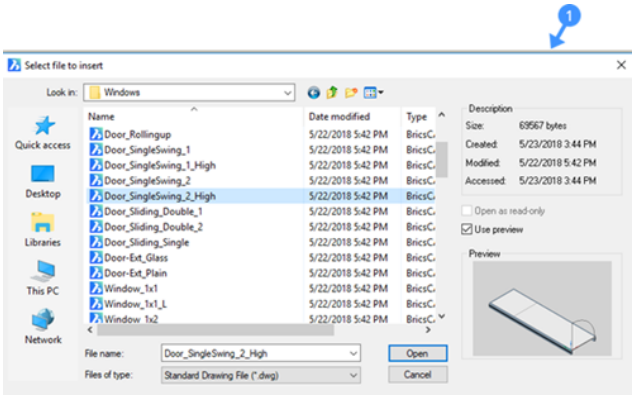
Uwaga: Make sure **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

Using the BIMINSERT command

1 Select **Insert BIM Component**from the **Model** tab of the No Selection Quad.

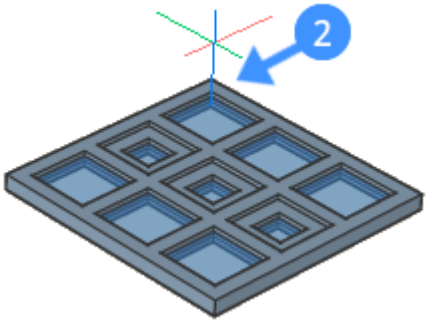
The file explorer opens, displaying the content of the most recently used folder to insert a BIM component. The default folders for BIM components sit in the following subfolder of the BricsCAD installation folder: ...\UserDataCache\Support\en_US\Bim\Components.

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).



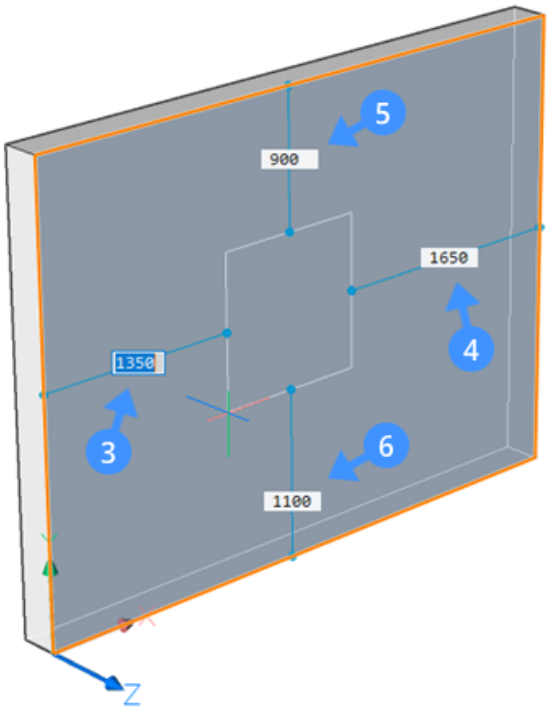
The parameters of the insert display in the **Properties** panel.

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>:

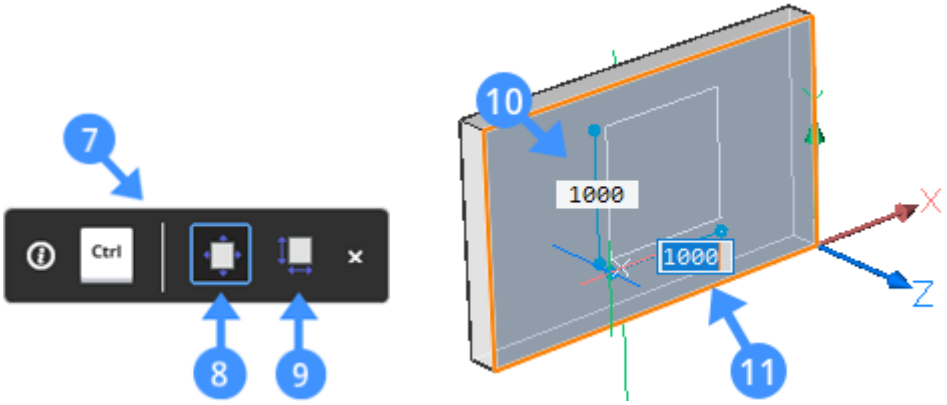
4 Optionally, edit the parameters in the **Properties** panel or choose an option (see Command Options of the BIMINSERT command).

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.



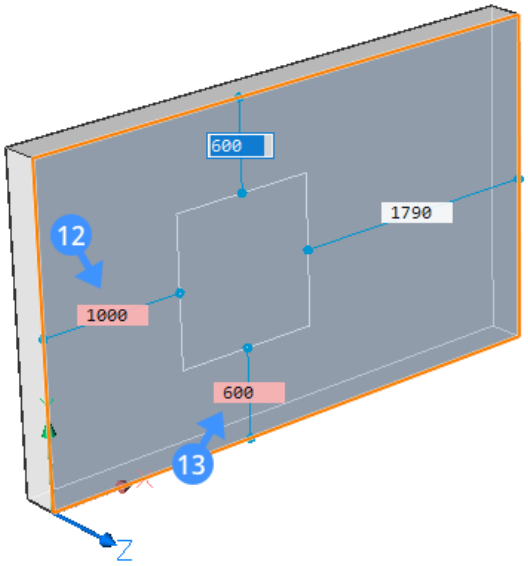
* Click the HKA field in the Status bar to toggle the display of the Hot Key Assistant.

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:

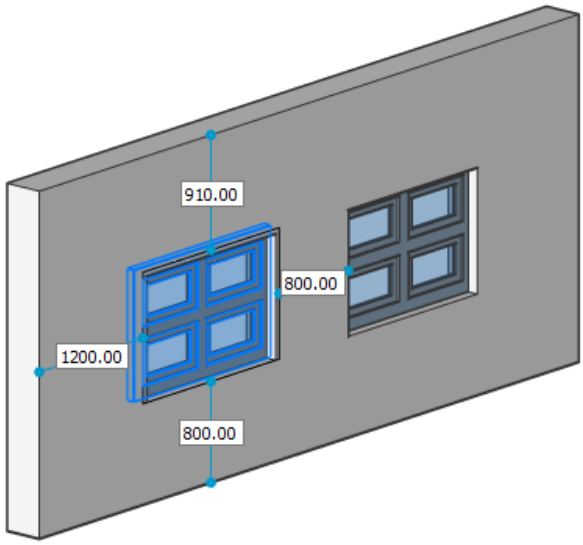
- Using Distance fields.

- 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:

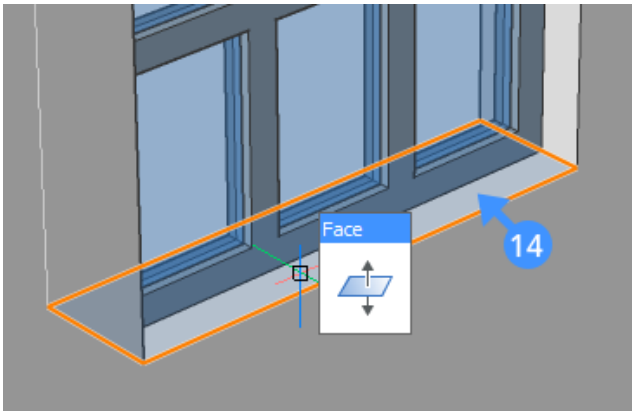


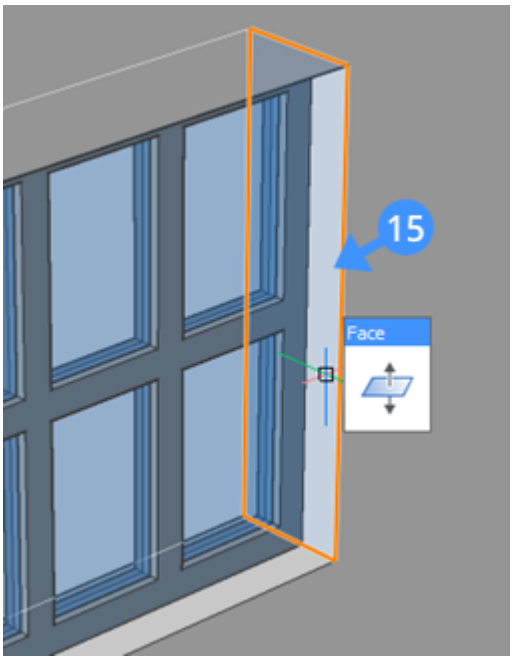
2 Double click a distance field, then type a value and press Enter.

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.

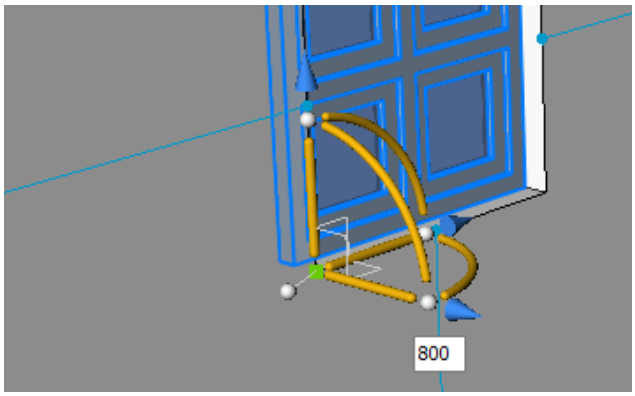
3 Do one of the following to complete the moving insert process.

- Specify a point.
- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

Procedure: using the Manipulator

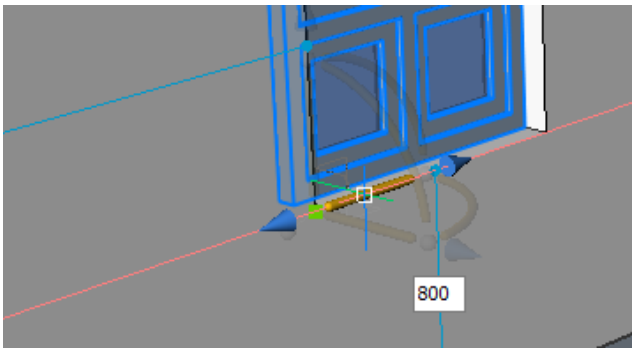
Select the insert by holding the left mouse button down a bit longer. Set the MANIPULATOR system variable to 2.

The Manipulator displays.



1 Hover over the vertical or horizontal axis trigger zone of the Manipulator.

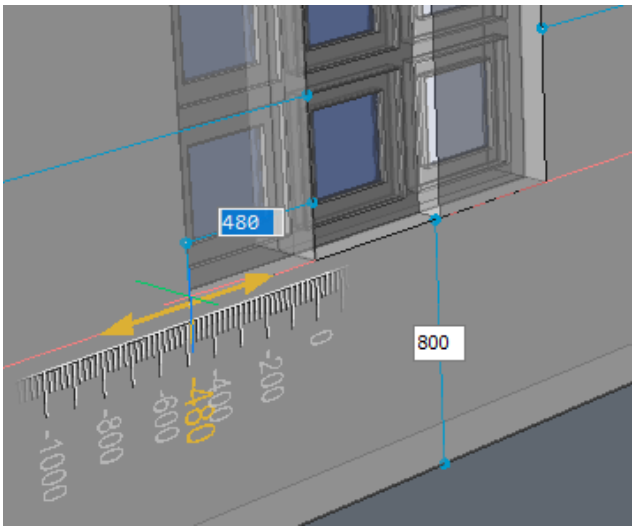
1 The axis highlights and the rest of the Manipulator fades.



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.

Uwaga: 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

Uwaga: Make sure that **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

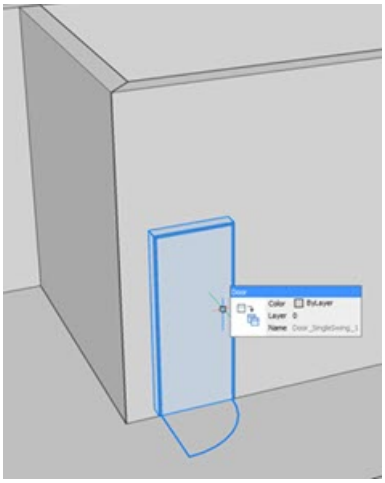
1 Select an insert using the **BIM Insert** tool.

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>:

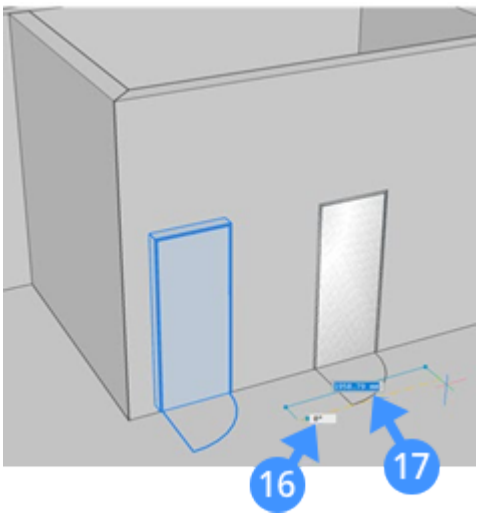
5 Pick a point in the drawing area.

6 You are prompted: Enter second point <Use base point as displacement>.

7 Type a displacement distance from the coordinates of the base point.

Uwaga: 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.

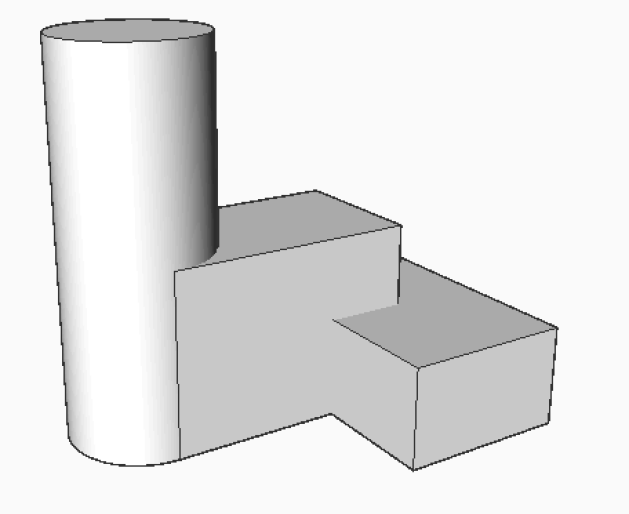


For more information about this command, please visit the Command Reference article BIMQUICKBUILDING.

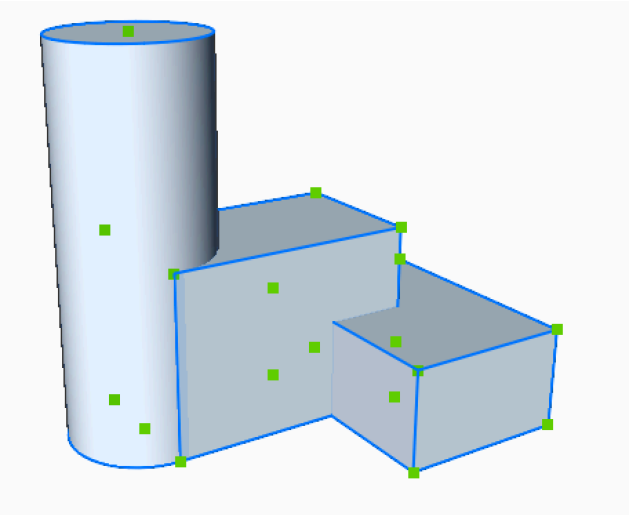
20.3.14.3 Procedure: top-down workflow

1 Draw one or more solids in your model space.

Uwaga: 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.

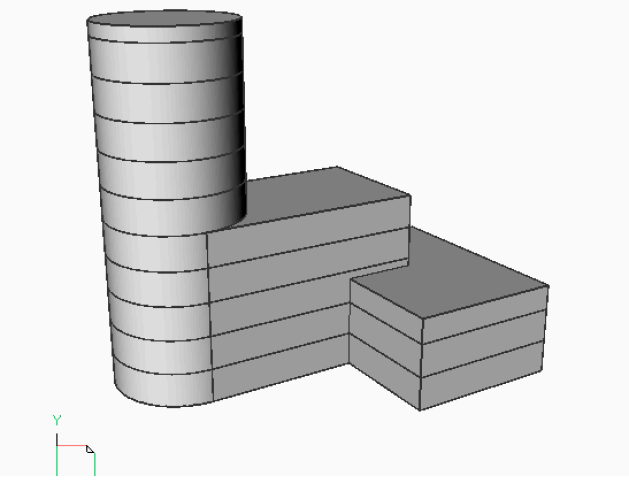


3 You are prompted: Do the selected solids represent [Buildings/Spaces]: Type B to continue with the top-down workflow and press Enter.

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.



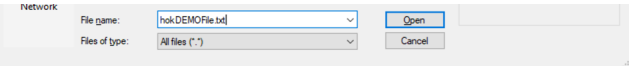
Uwaga: To generate different floor heights, type (2x4000,*3000,1x5000) which stands for two stories of 4000mm, the rest 3000mm and the top floor has a story height of 5000mm.

Or import the story height values from a .txt file.

Enter BIMQUICKBUILDING in the Command line,

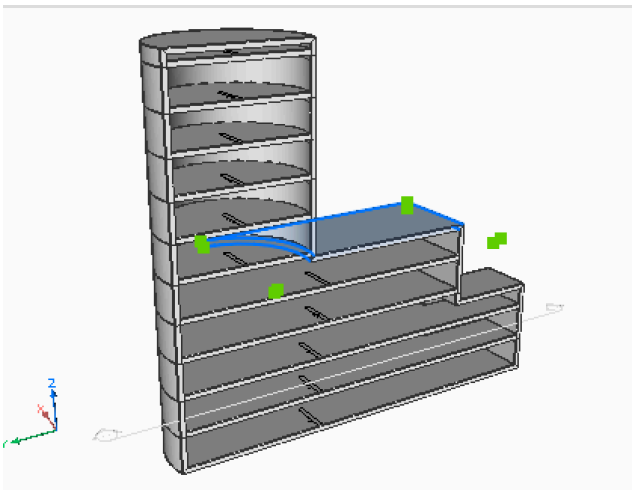
Type BUILDINGREQUIREMENTFILE and press Enter.

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.

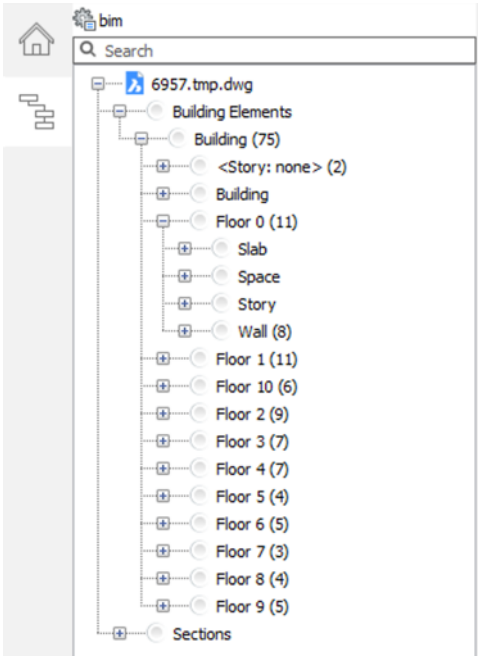


Select the roof and navigate to the **Properties** Panel - **BIM** tab. You will see that this entity has **Roof type** property, as well as some other properties such as spatial locations, quantities or a **GUID**.

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	

Uwaga: 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).

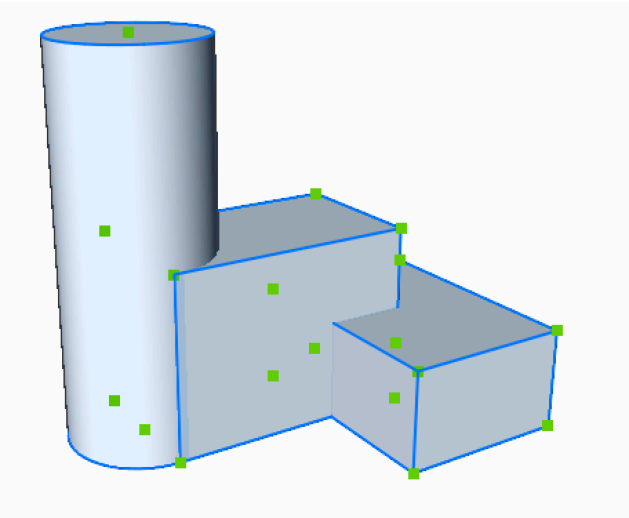


20.3.14.4 Procedure: bottom-up workflow

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.



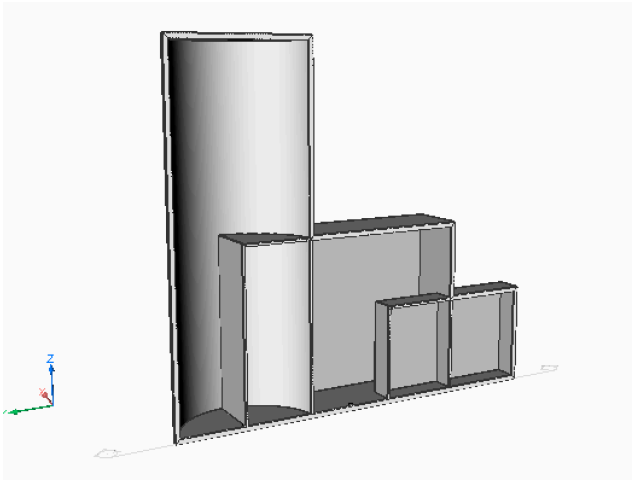
3 You are prompted: Do the selected solids represent [Buildings/Spaces]:

4 Type Spaces to create a building from the selected massing solids.

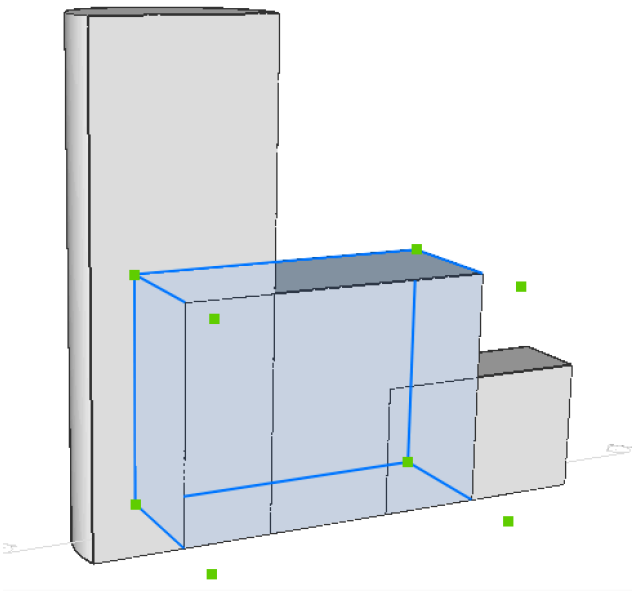
Uwaga: BricsCAD takes these defined solids and converts them to BIM Spaces with walls and slabs.

5 Press Enter to create the building.

6 (Optional) make a section to see the newly created spaces.



Uwaga: 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.

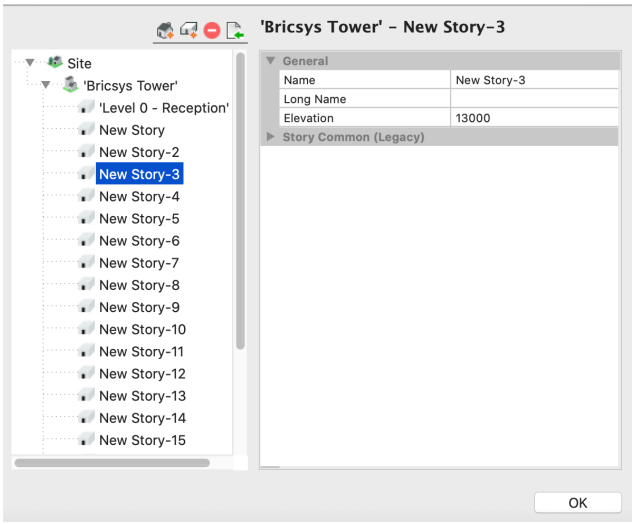


20.3.14.5 Procedure: SPATIALLOCATIONS as input

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.

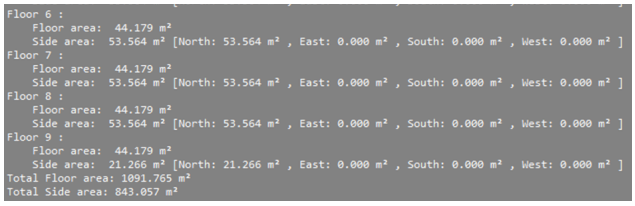
5 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatiallocations]:

6 Type SPATIALLOCATIONS to use the Spatial Structure as input.

Uwaga: 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.



Uwaga: 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.

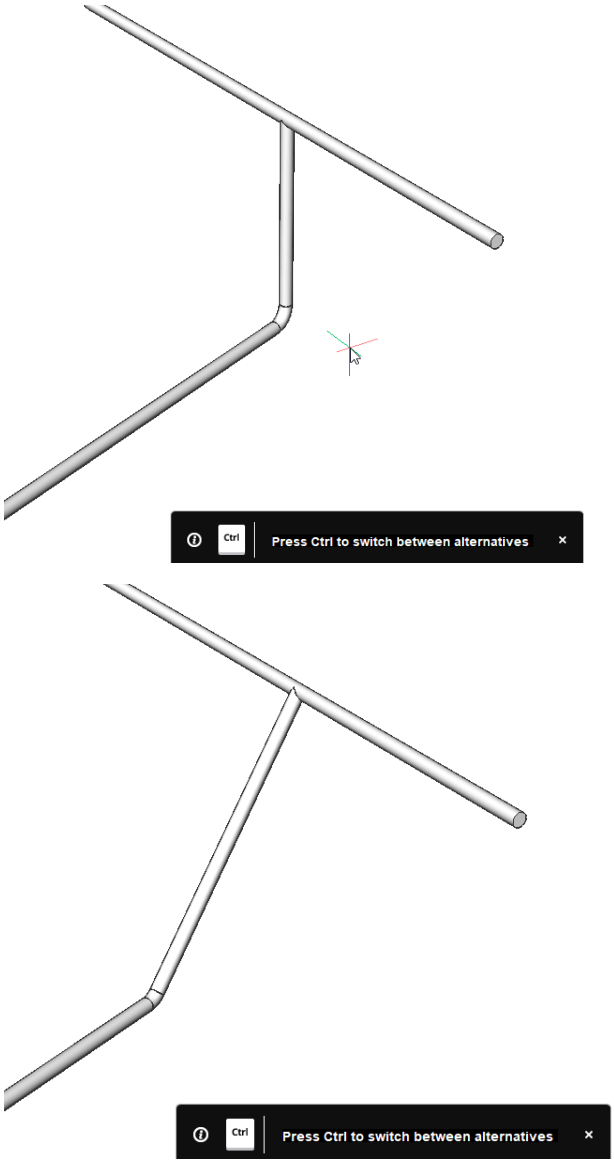


For more information about this command, visit the Command Reference article BIMFLOWCONNECT.

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.

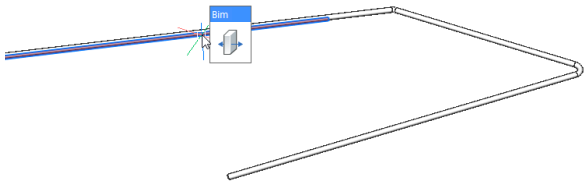
DRAG

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.

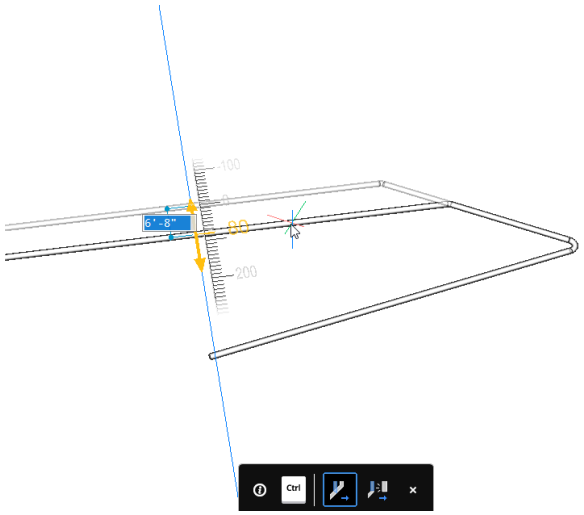
1 Hover over a segment, select a face perpendicular to the section of the profile.



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.

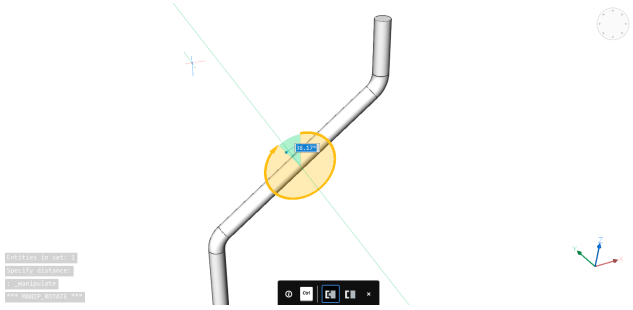
Uwaga: The software automatically regenerates the connection between the segments.



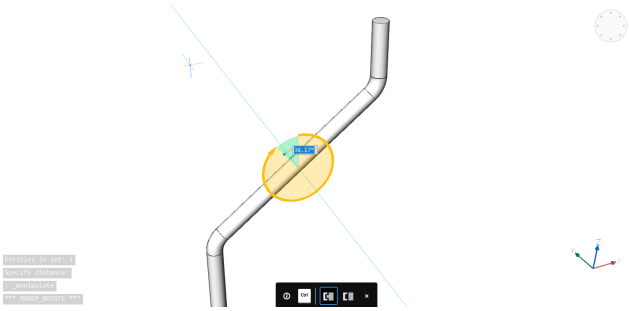
Manipulator

1 Hover over a segment, select a face perpendicular to the section of the profile.

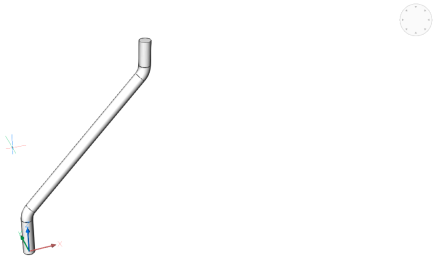
2 Select **Manipulate**.



3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the rotation.

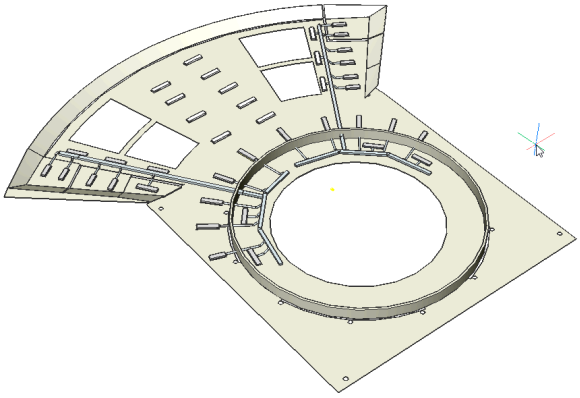


4 Click on the new position or enter the rotation angle and hit enter.



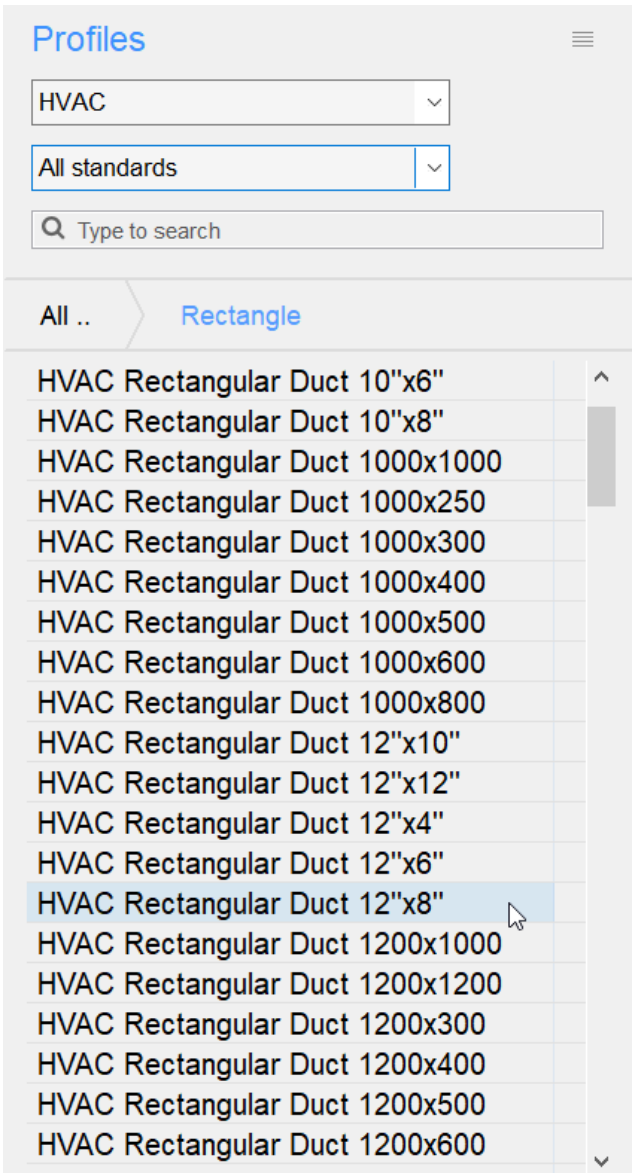
20.3.15.4 Procedure: using the MEP modeling tools to draw and connect profiles

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.



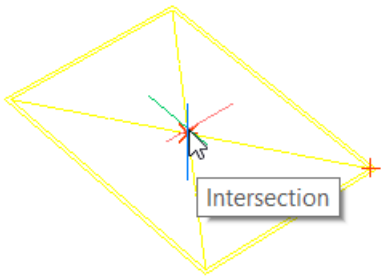
2 Open the BIM **Profiles** panel on the right-hand side of your screen (see Dockable Panels).

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.

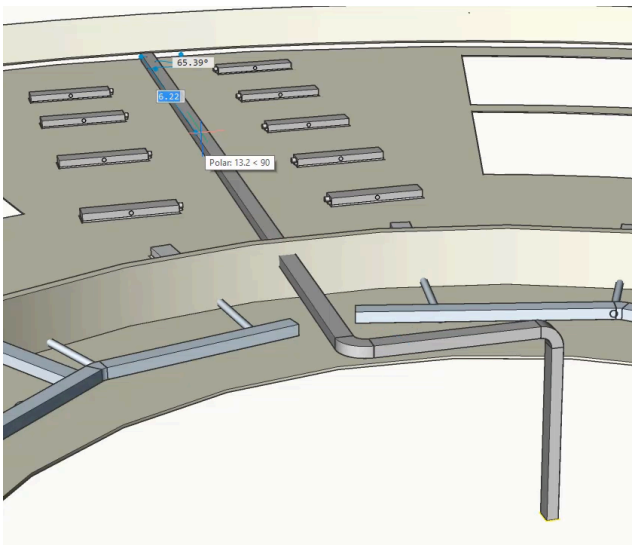
4 Snap to the geometric center of the yellow rectangle.



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.

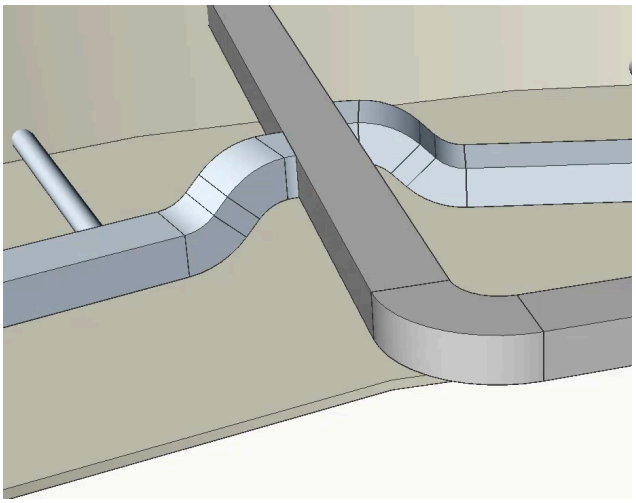
Go up 3.5, go 3.5 to the left and go 21 horizontally.



Uwaga: 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.

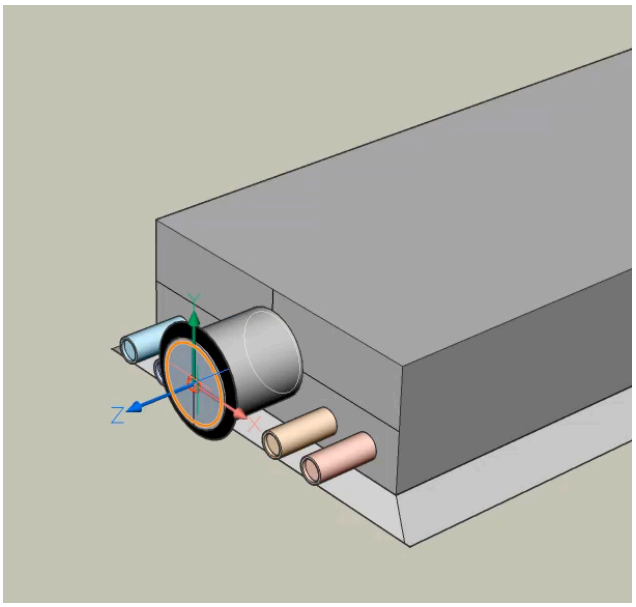
7 Select the ducts together with the main duct and launch the BIMFLOWCONNECT command. Your connection should look like the image below.



8 Create a duct from the flow terminal.

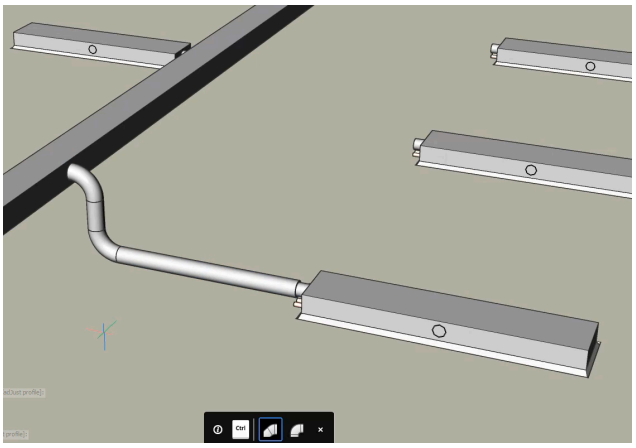
9 Launch the BIMLINEARSOLID command and select the flow connection point.

This connection point contains profile information, so the software automatically creates the predefined circular duct. Give the duct a length of 1.



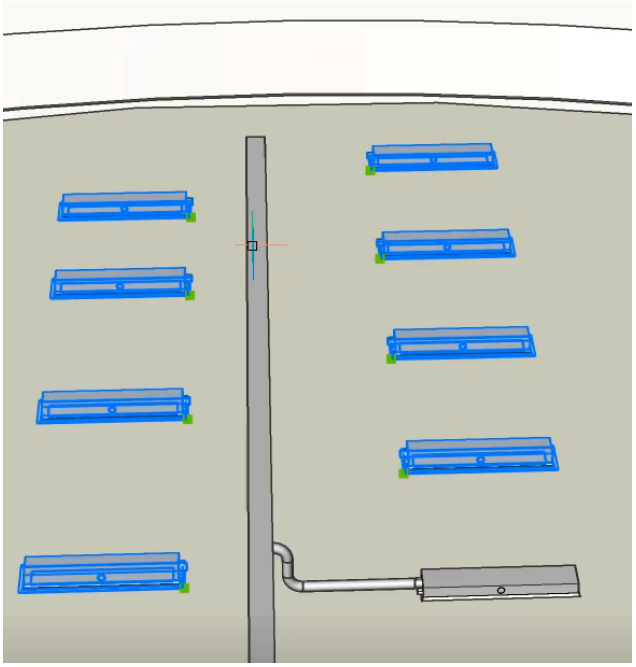
10 Connect the ducts from the flow terminal to the rectangular main duct.

11 Select the main duct and the duct from the flow terminal and launch the BIMFLOWCONNECT command.



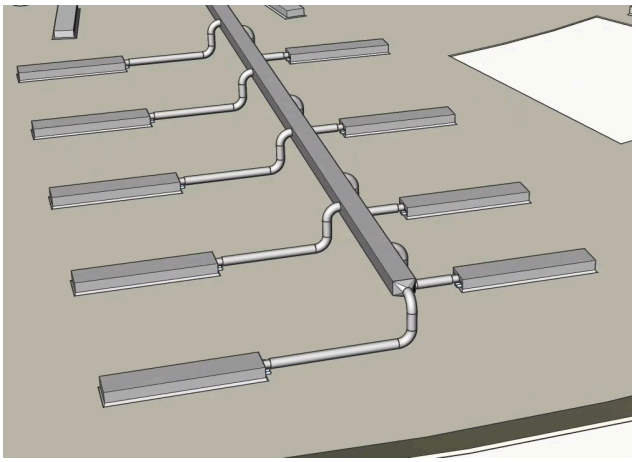
12 Connect all the other flow terminals to the rectangular duct at once.

13 Select the other flow terminals and the rectangular duct and launch the BIMFLOWCONNECT command.



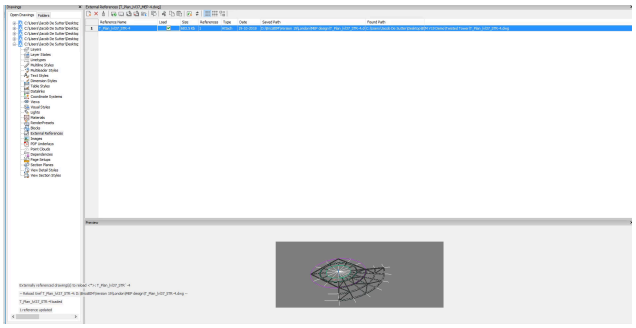
14 BricsCAD uses the information from the flow connection point to determine the type of duct used to connect with the main duct.

Uwaga: 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.



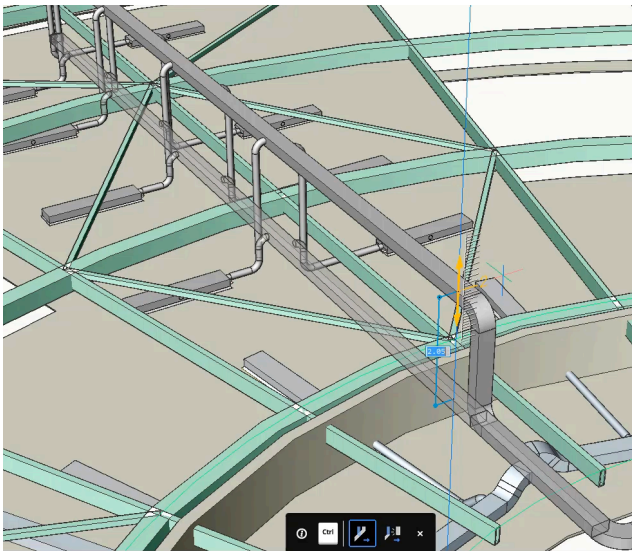
2 Use SLICE command and **Multislice** option to divide the rectangular duct.

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.

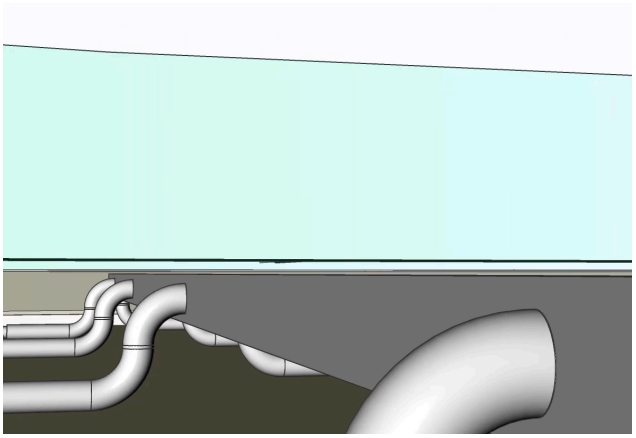
4 Select the upper face of the duct and launch the DRAG command.

Uwaga: 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.

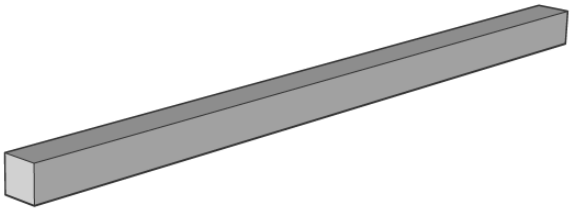
2 Open the BIM **Profiles** panel by clicking on it on the right-hand side of your screen.

Uwaga: 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:

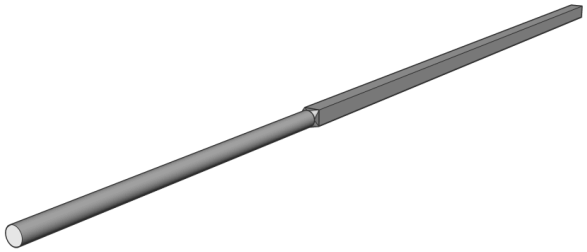


6 Search for the profile **HVAC Round Duct 8"** in the **Profiles** Panel and drag the profile in the model space.

7 Click on the end of the Rectangular profile, after hovering on its profile face.

Uwaga: To easily snap on the end of the rectangular profile, turn on the **Display Axes** setting or set the DISPLAYAXES system variable to 1.

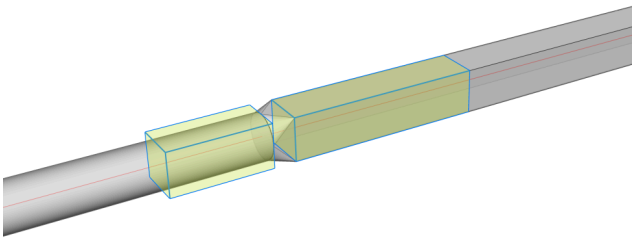
8 Make sure to draw the circular profile along the same axis as the rectangular one. After accepting the last point of the circular profile, it will be connected to the rectangular profile through an automatically generated reducer.



Uwaga: 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.

Uwaga: Make sure the **Display Sides and Ends** and **Display Axes** is enabled.

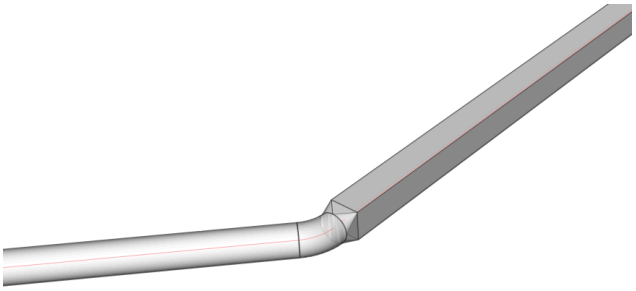
9 Select the connecting ends of both segments.



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

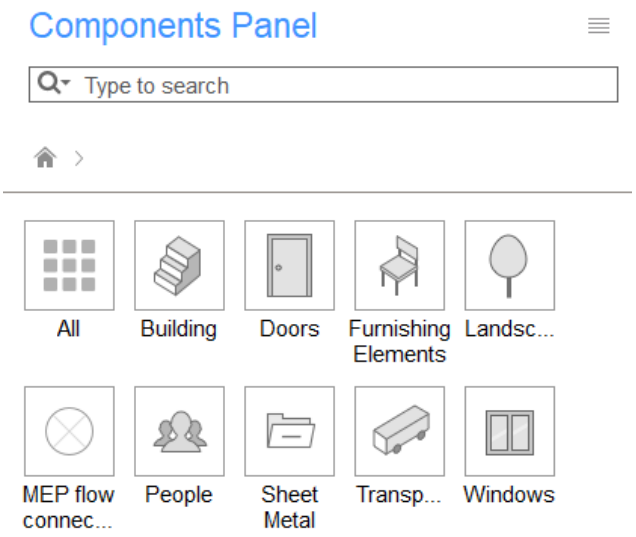
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

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.

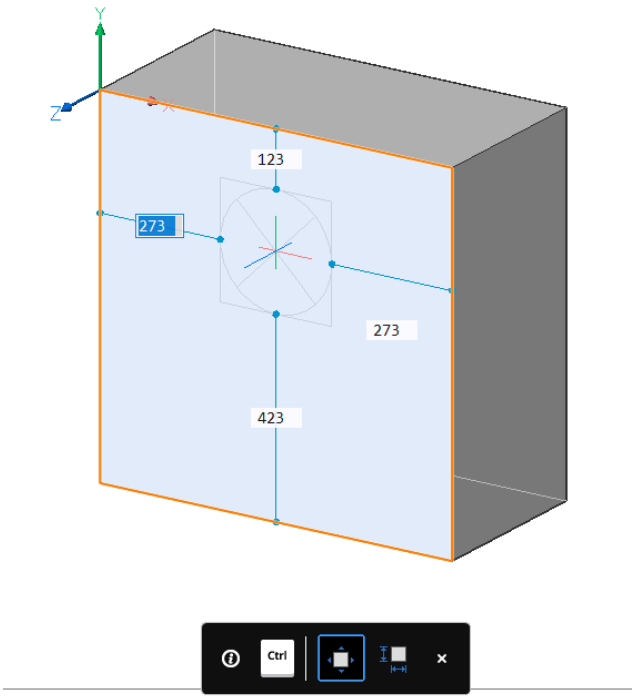
3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Open the **Library** panel.

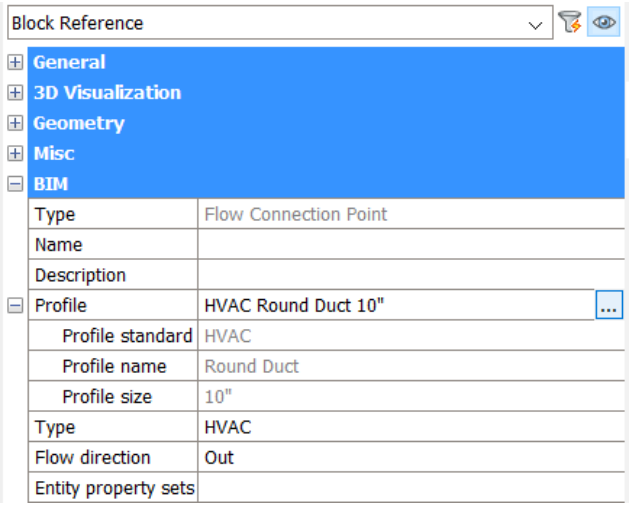
5 Go to the **MEP Flow Connection Points**.



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.

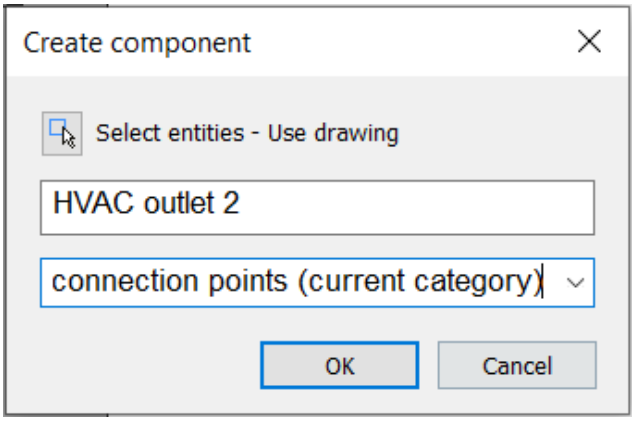
9 Set **Type** and **Standard** = **HVAC**.

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

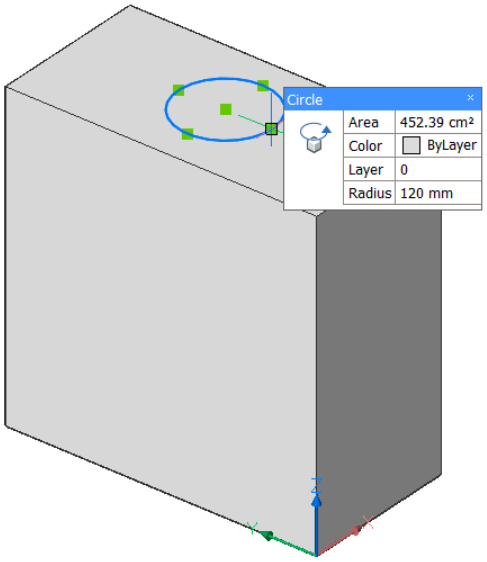
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

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.

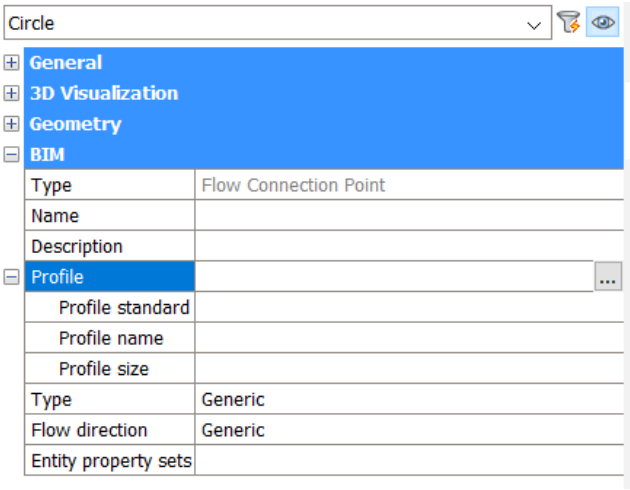


5 Select the circle and launch BIMCLASSIFY with the **Other** option.

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

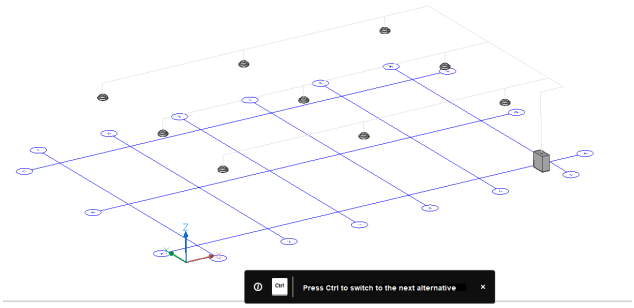
In this procedure, we will use flow connection points to create a ventilation system.

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.

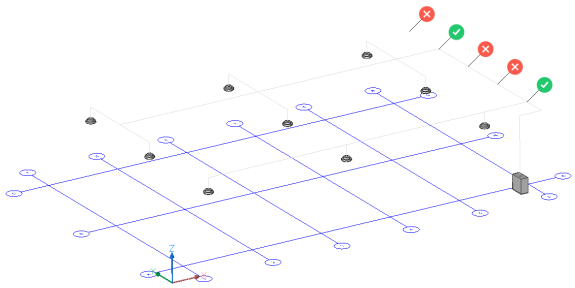
4 Press the CTRL-key to switch between the alternative setups and press Enter.



5 Click the **Tune** option or enter T in the Command line to modify the chosen setup.

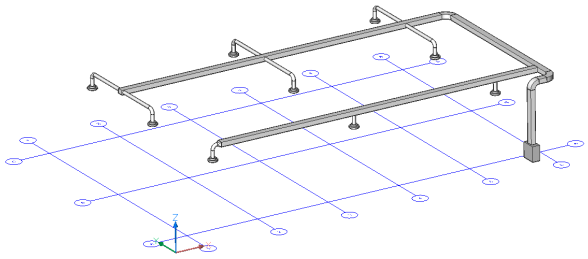
6 Select **Topology** in the toolbar or enter T and press Enter.

7 Select the check icons to modify the layout of the system.



8 Press Enter 3 times to exit the command.

9 ***Uwaga:*** 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.

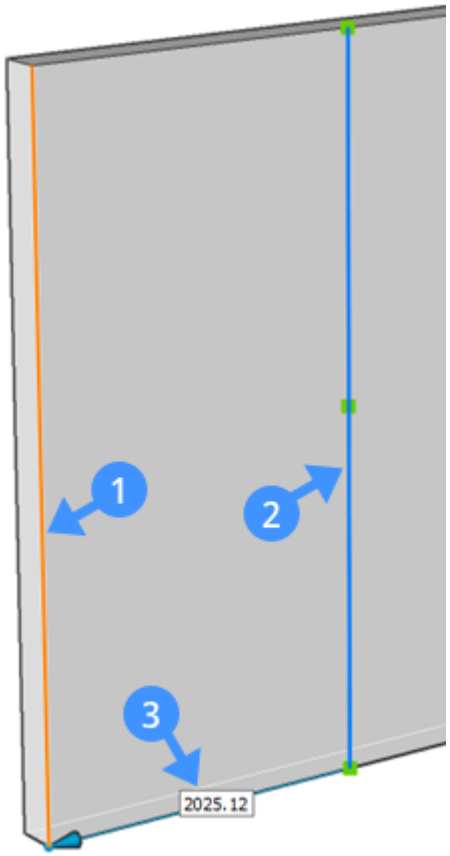
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by dragging the edge of the solid face.

Uwaga: 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.

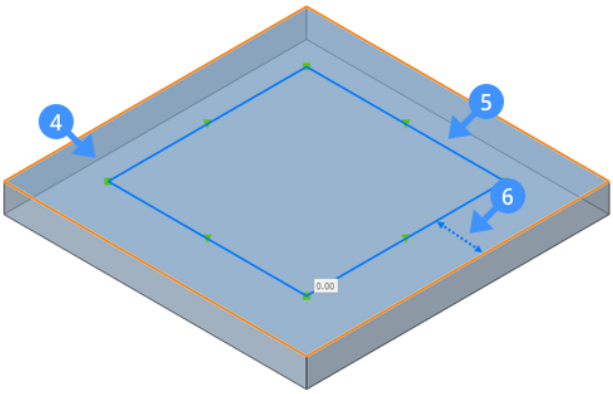
2 Choose the **Offset** tool from the Quad.

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.

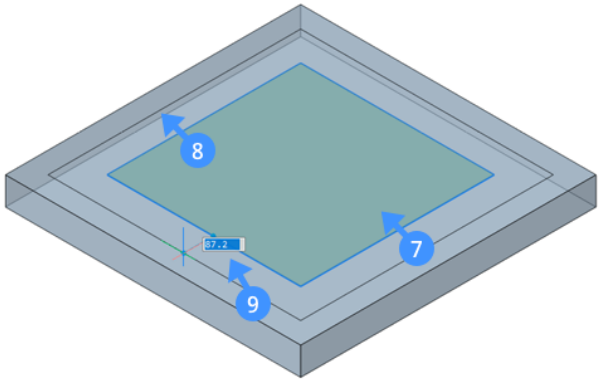
2 Choose the **Offset** tool from the Quad.

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

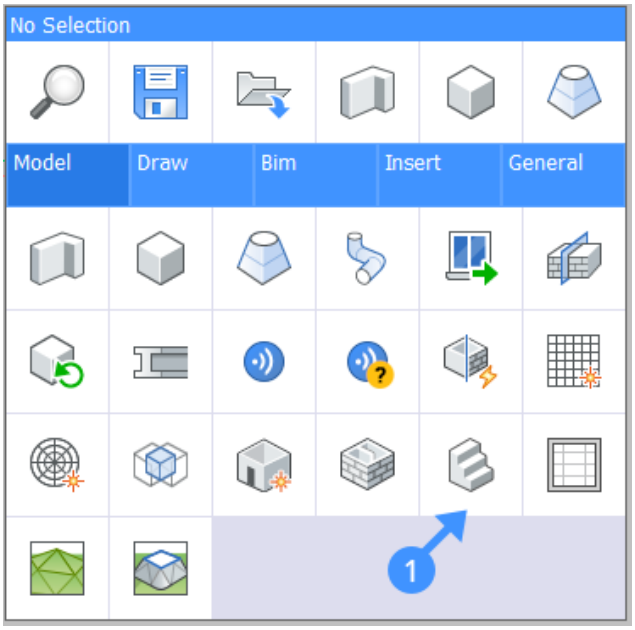
The **Stair** tool is a powerful tool that allows you to add a custom flight of stairs for your project in just two clicks. You can change the default stair settings as required. The **Stair** tool will automatically try to fit these requirements. It is possible to create single-flight straight, double-flight L-shaped, and double or triple-flight U-shaped stairs.



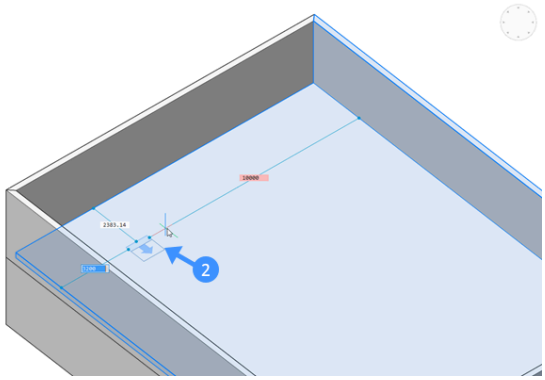
For more information about this command, visit the Command Reference article BIMSTAIR.

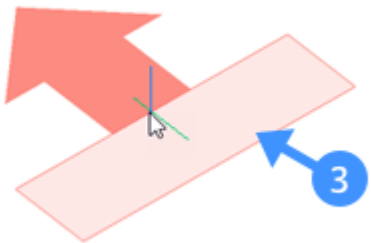
20.3.19.3 Create a straight flight of stairs

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).



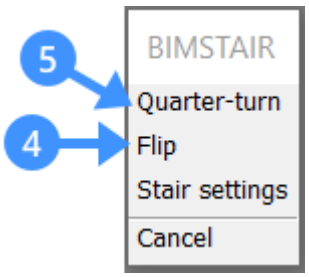


3 To flip the direction of the first step, click the **Flip** button (4) or type F.

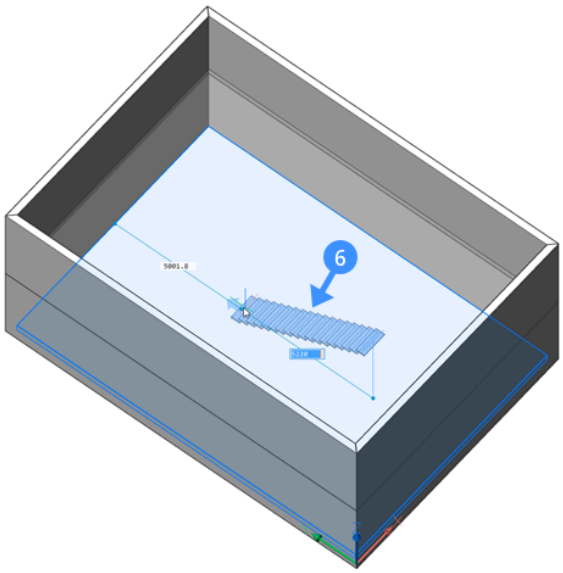
To turn the direction of the first step 90°, click the **Quarter-turn** button (5) or type Q.

To cycle through the different stair variants, the CTRL-key can be used when the Hotkey Assistant is active.

Uwaga: 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.

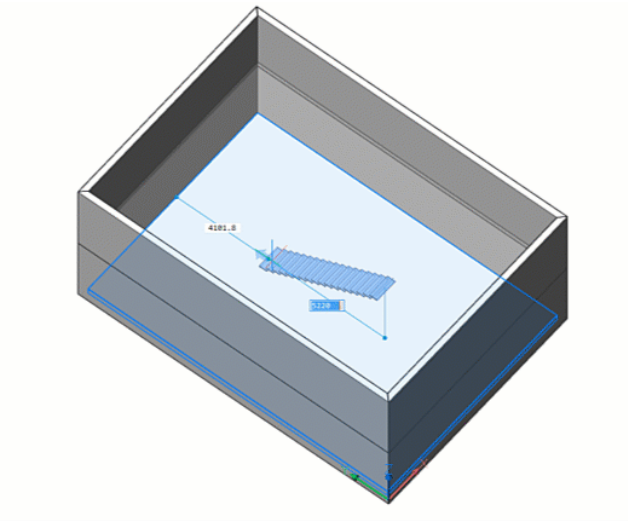


4 Once the cursor is in position, press Left-click. If the upper slab is blocking the view, the slab will become transparent.

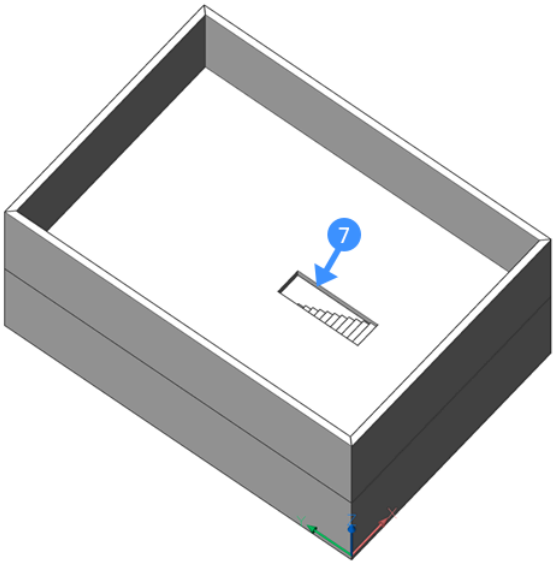


5 To create a straight flight of stairs, align the mouse perpendicular to the first step.

Uwaga: 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).

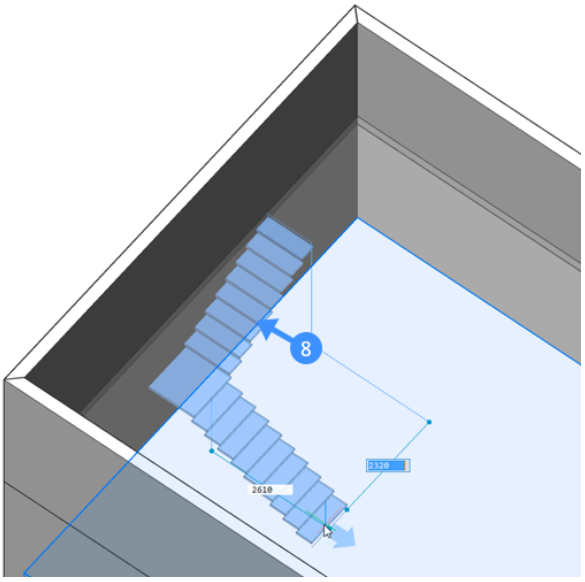


20.3.19.4 Create an L-shaped staircase

1 Select the **Stair** tool from the Quad.

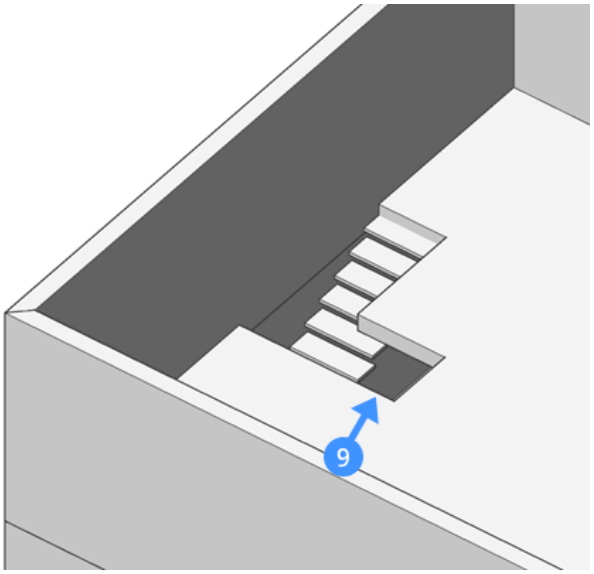
2 Hover over a slab. The Stair Cursor will appear. Left-click once the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to create an L-shape flight of stairs (8).



Uwaga: 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.

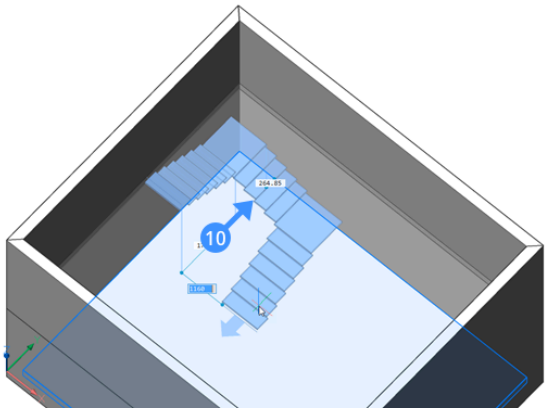
1 Select the **Stair** tool from the Quad.

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.



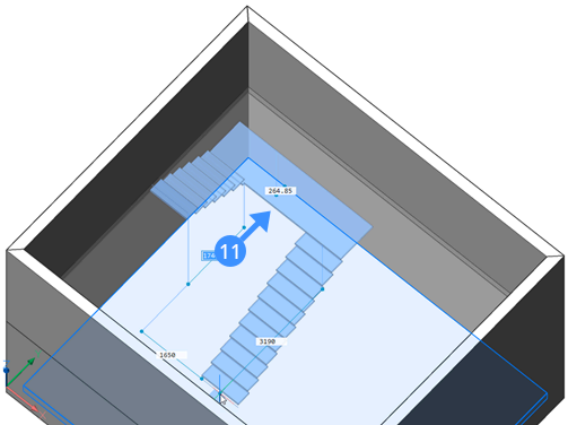
Uwaga: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (10). The **Stair** tool will automatically snap to the configurations that correspond to the current settings. When the optimal configuration is satisfied, this preview will display in a darker shade 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).

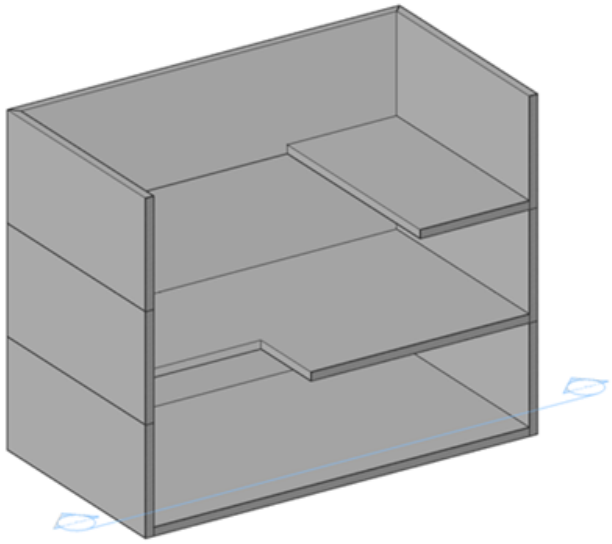
It is possible to switch between these two modes by typing U and hitting Enter.

Orpress CTRL to cycle between different stair variants.



20.3.19.6 Create a flight of stairs in an existing opening

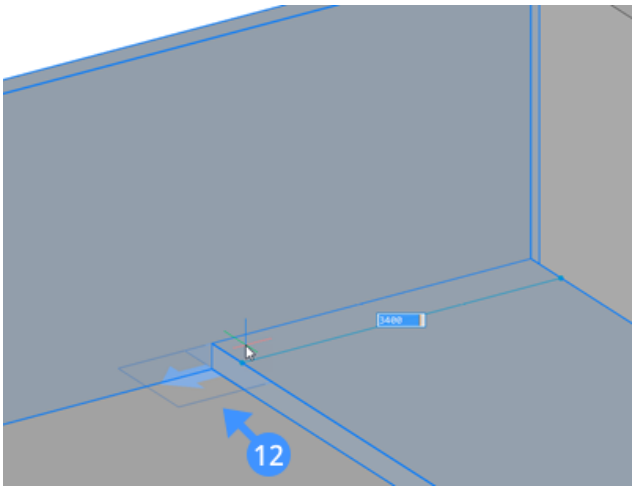
Sometimes it is necessary to create a flight of stairs starting from the edge of a slab, where an opening already exists.



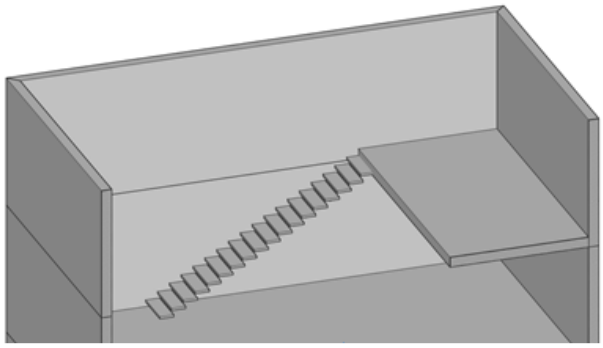
In the example above, different floors need to be connected with flights of stairs. However, an opening does not need to be created.

1 Select the **Stair** tool from the Quad.

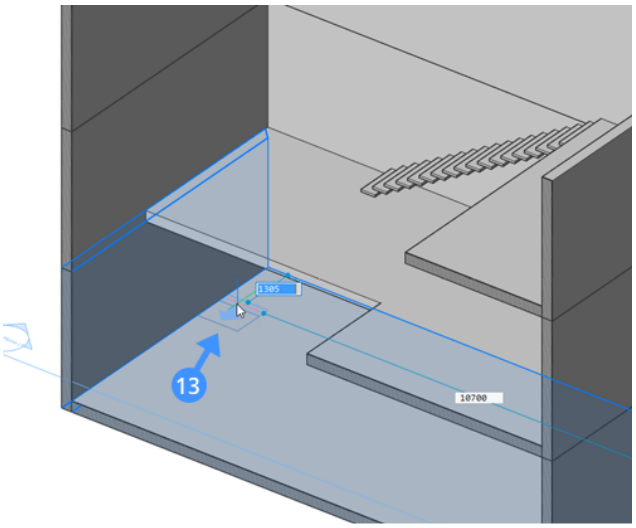
2 Hover the Stair Cursor near the edge of the slab (12). The cursor automatically snaps to the outer side of the edge.



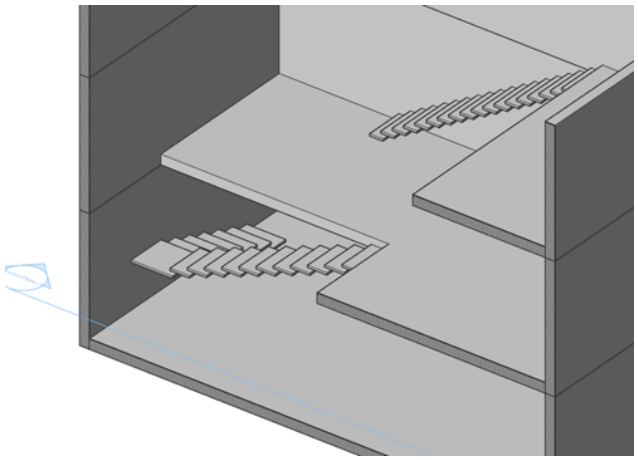
3 You can now create a flight of stairs as described above.



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'.

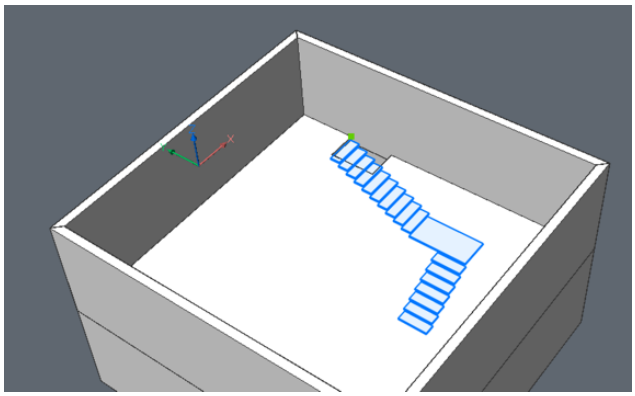
Edit a flight of stairs

You can adjust multiple parameters of a flight of stairs created with the **Stair** tool in the **Properties** panel. These parameters include:

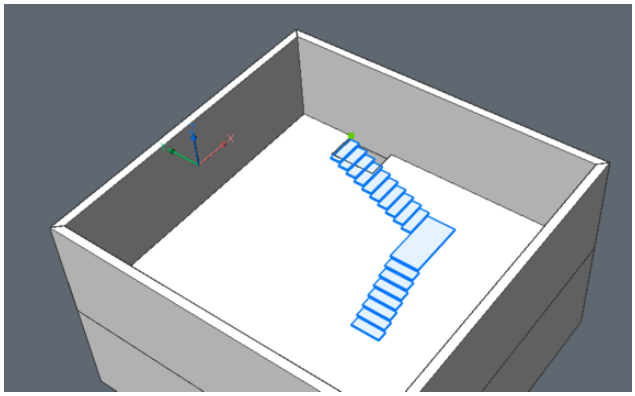
Parameters	
Height_Staircase	325 cm
Stair_Width	1000 mm
Ideal_Riser_Height	170 mm
Step_Thickness	50 mm
Nosing	50 mm
Length_StairFlight_0	145 cm
Length_StairFlight_1	174 cm
Length_StairFlight_2	145 cm

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

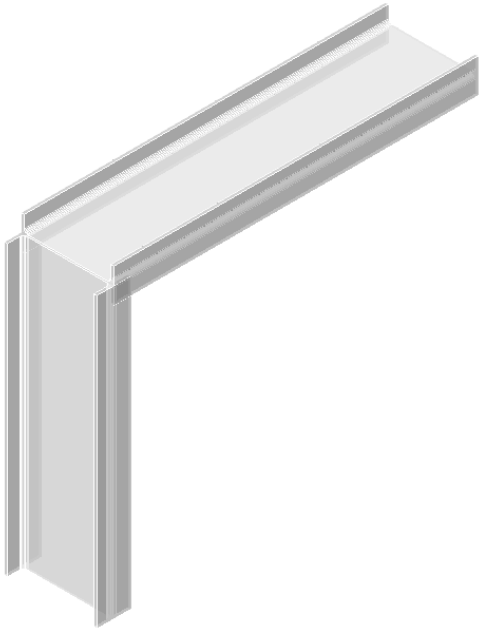
20.3.20.2 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the BIM **Profiles** panel to the model space.

Uwaga: 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.



Uwaga: 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.



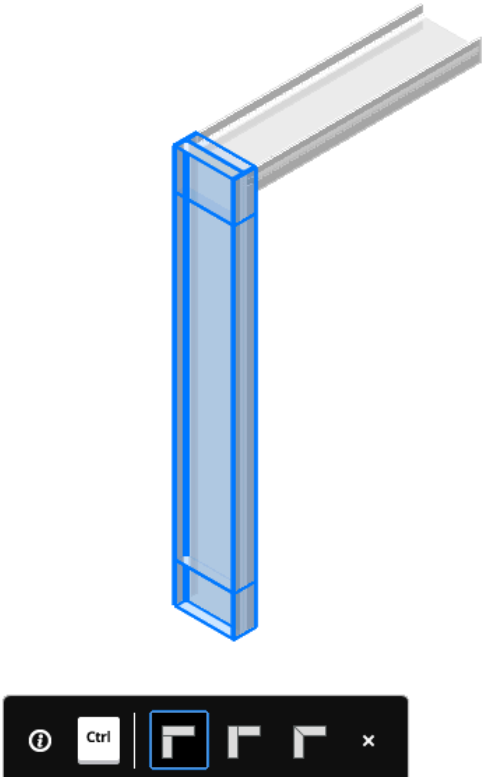
For more information about this command, visit the Command Reference article BIMSTRUCTURALCONNECT.

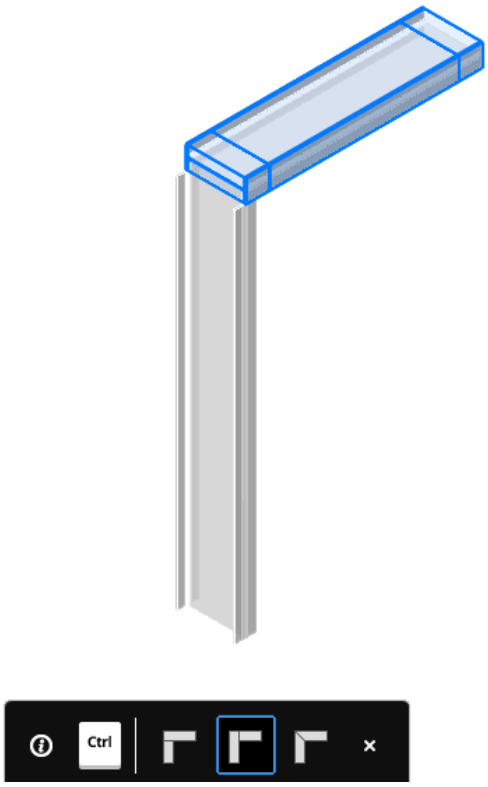
Making a structural connection to linear solids

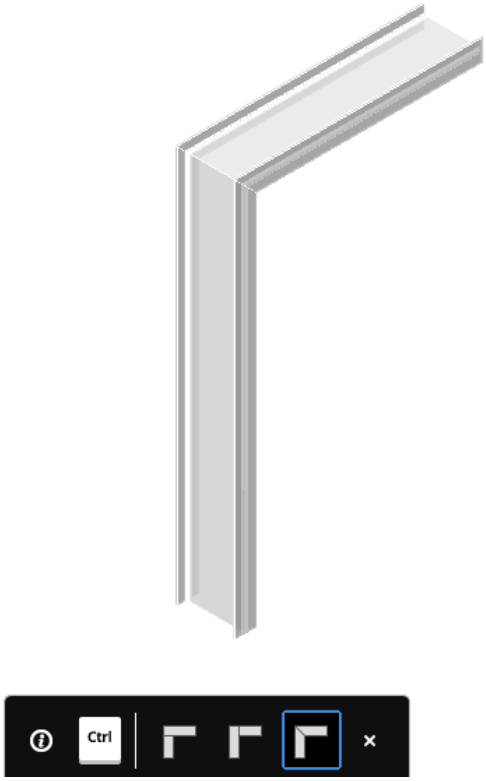
1 Select the profiles you want to connect with.

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.



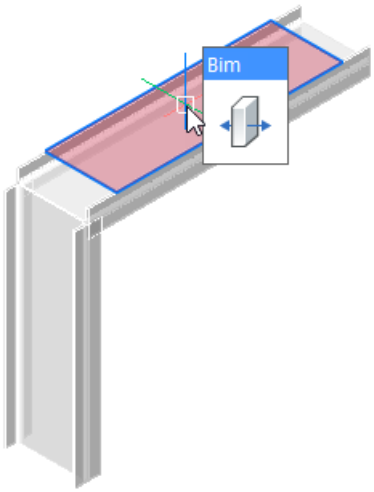
For more information about this command, visit the Command Reference article DRAG.

Altering the positioning of segments

Uwaga: This only works if the connections were created with the BIMSTRUCTURALCONNECT command.

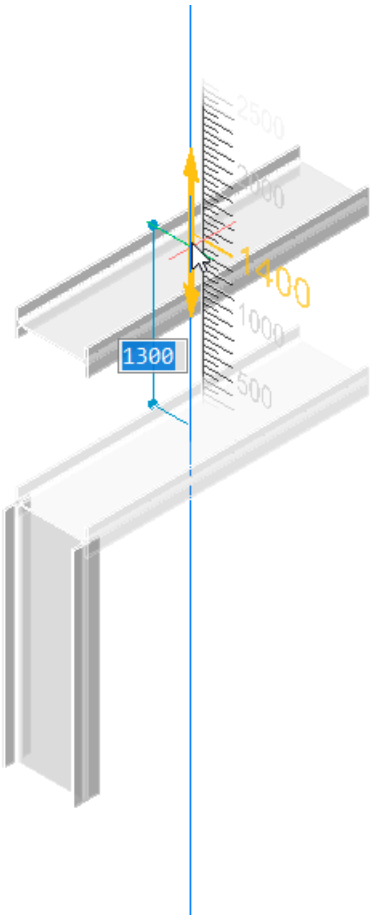
1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

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.

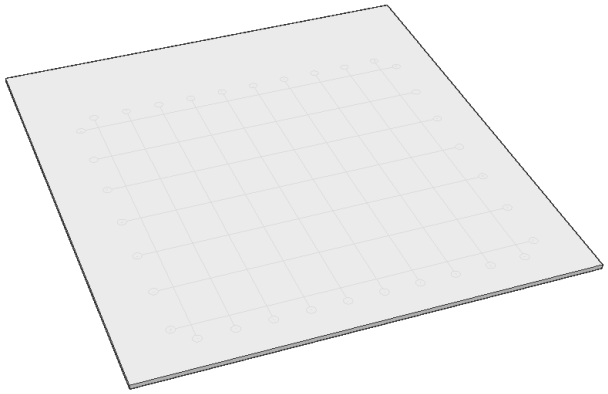
2 Launch the BIMGRID command.

3 Click a starting point and enter values into the dynamic dimension fields.

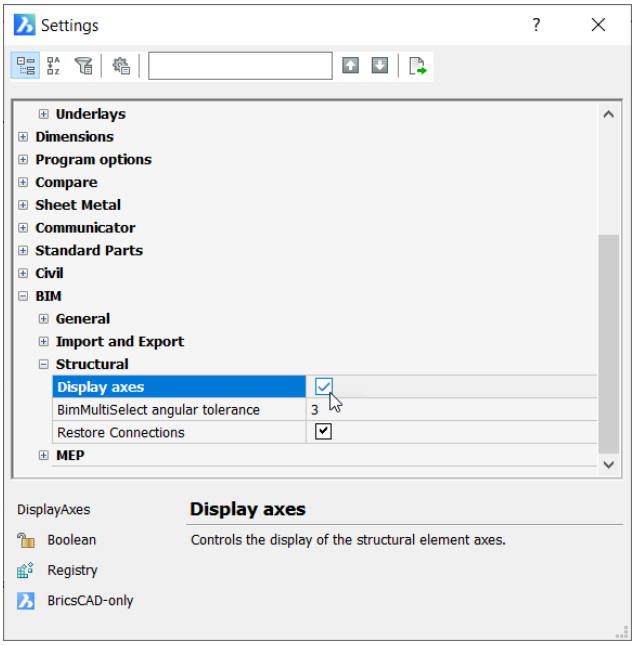
4 Draw a base slab underneath the grid.

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.

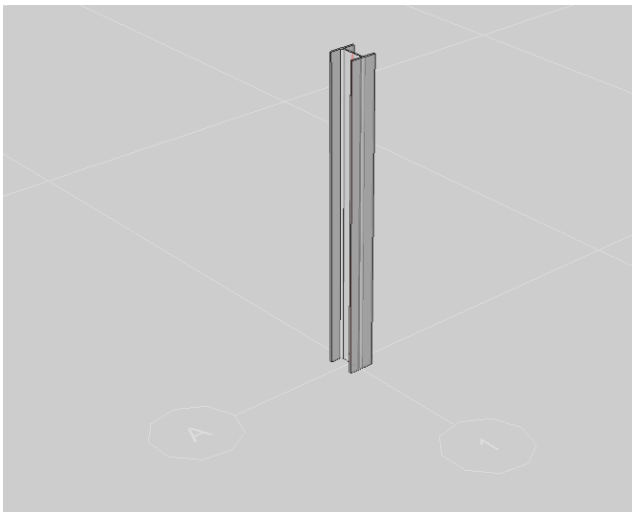


7 Go to the **Settings** dialog and under the **BIM** > **Structural** section. Ensure that **Display Axes** is checked.



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.

Uwaga: 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.

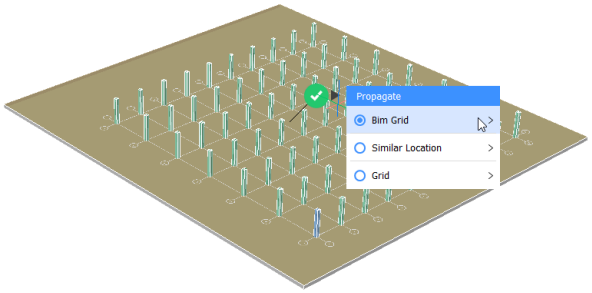
11 Select the column as your detail element and press Enter. BricsCAD will zoom into the 3D detail. Press C to copy the column as a solid, not as a block.

Uwaga: The default setting for BIMPROPAGATE is to copy the object as a block.

12 Hit Enter.

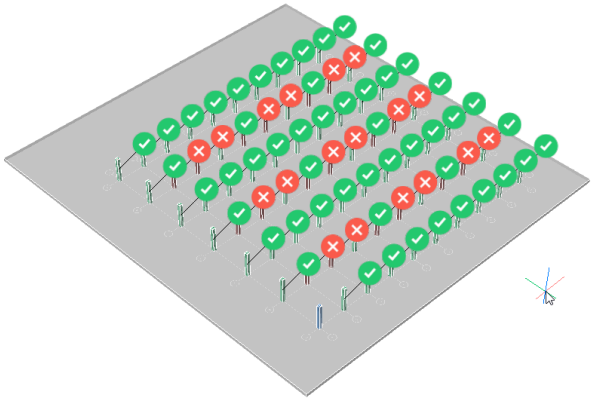
Uwaga: BricsCAD automatically copies the beam to every grid intersection. This is because you chose the floor slab with a BimGrid as your base solid.

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.

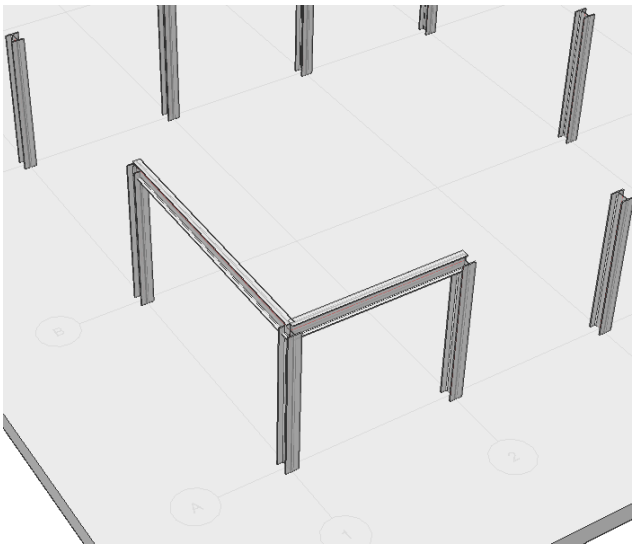
15 Press Enter to accept and end the command.



Create beams between the columns

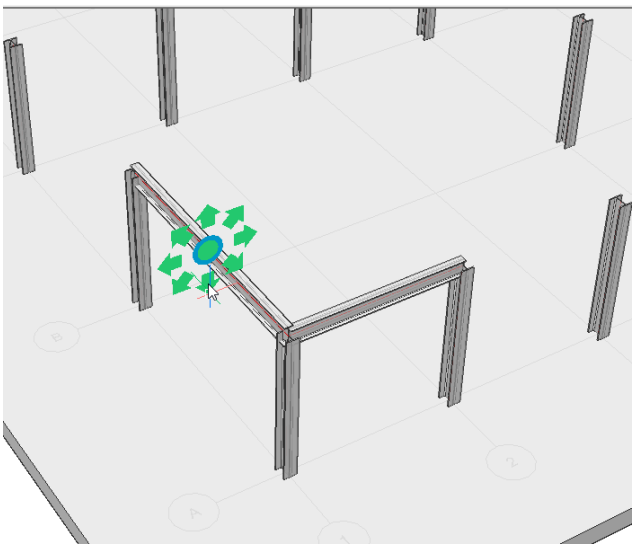
1 Open the **Profiles** panel and drag a structural steel profile into your model. To set the start and endpoints, click the top of the column and then the next. Do this in both directions.

Uwaga: 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.

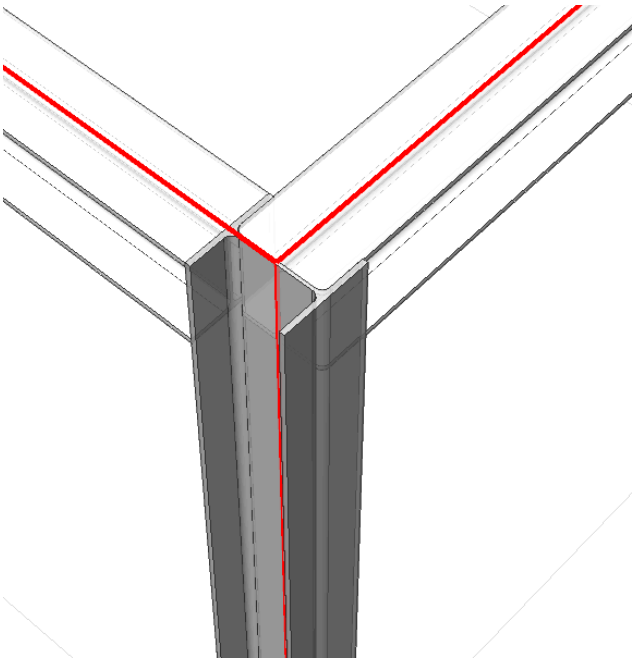


4 Click the down arrow. The beam moves down in relation to its axis.

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.

Uwaga: Suggestions are made at the bottom of every column because the situation is symmetrical. You can toggle the suggestions off.

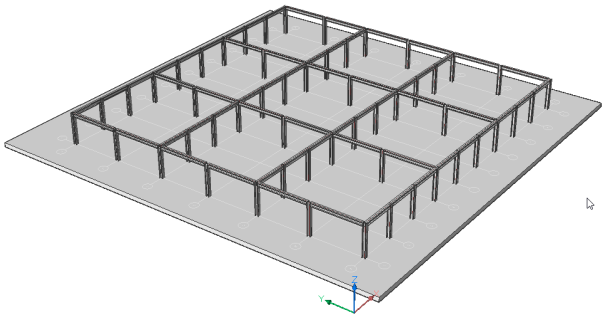
14 Repeat steps 9 - 13 to create beams on the X-direction.

Uwaga: You can use box selections to select all the lower beams at once.

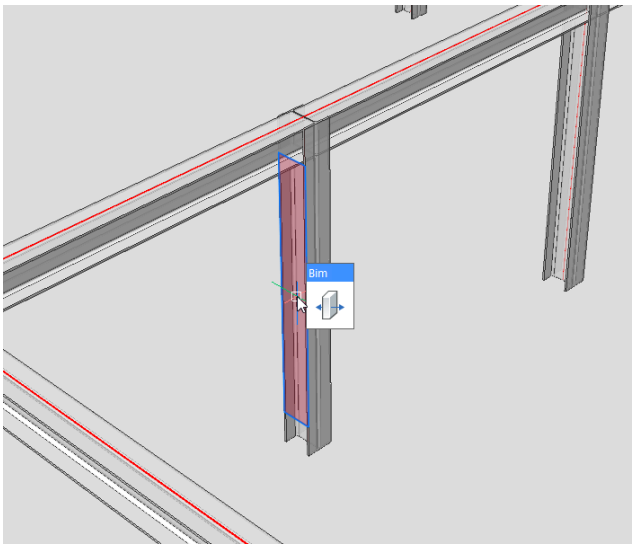
Move a Beam

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.



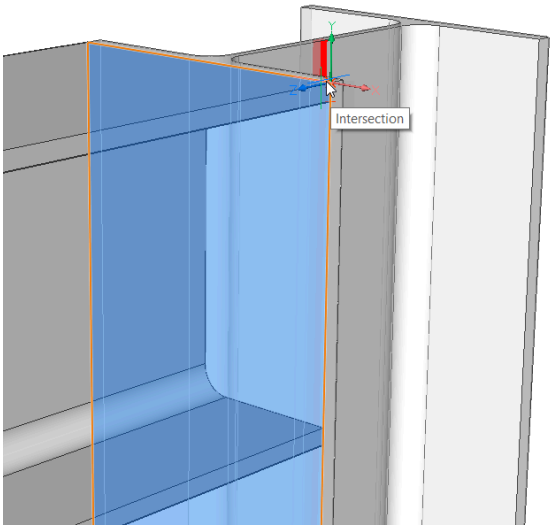
Uwaga: The connections with the adjoining beams are maintained.

20.3.20.6 Procedure: use the modeling tools to create a more detailed connection

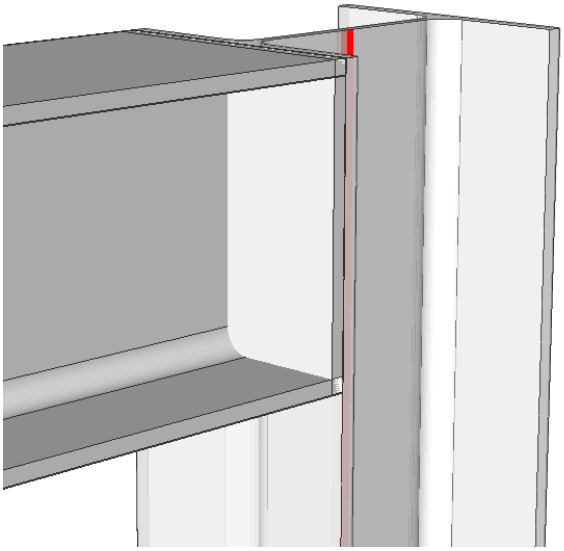
1 Set the value of DISPLAYSIDESANDENDS system variable to 0.

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.

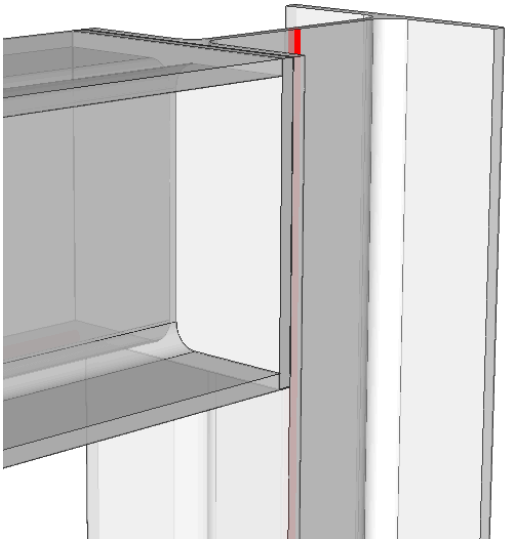


4 Now draw a plate on the flange of the column.



5 Select the beam and launch the SUBTRACT command.

6 Select the endplate and press Enter.

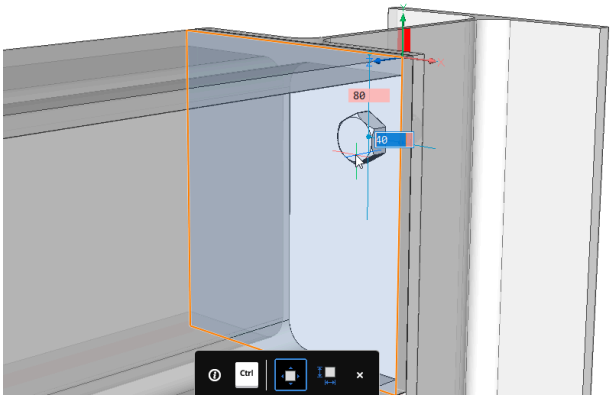


7 Select the face and launch the BIMIFY command from the Quad.

Uwaga: 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.

Uwaga: You can position it precisely by entering values in the dynamic dimension fields.

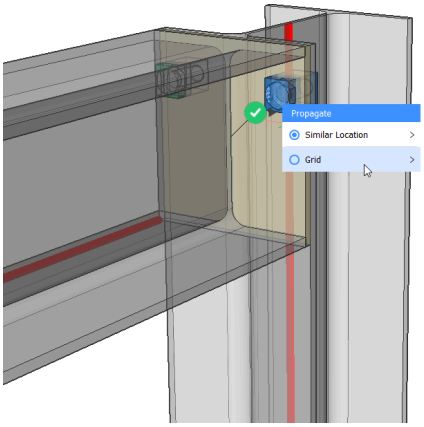


9 Select the plate and launch the BIMPROPAGATE command.

10 Select the bolt as the detail and press Enter.

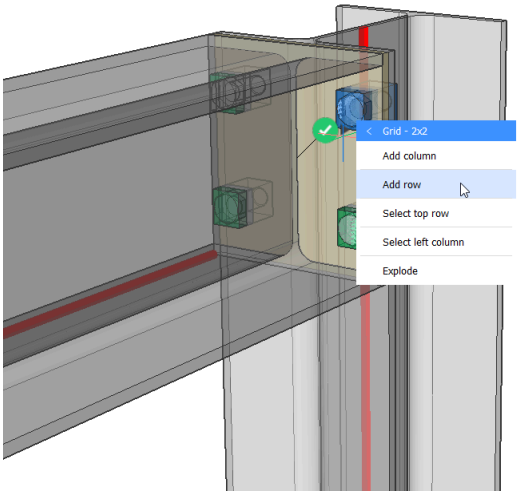
11 Press Enter again and click on the blue question mark. It will turn into a green checkmark.

12 Hover over the checkmark and select **Grid**.

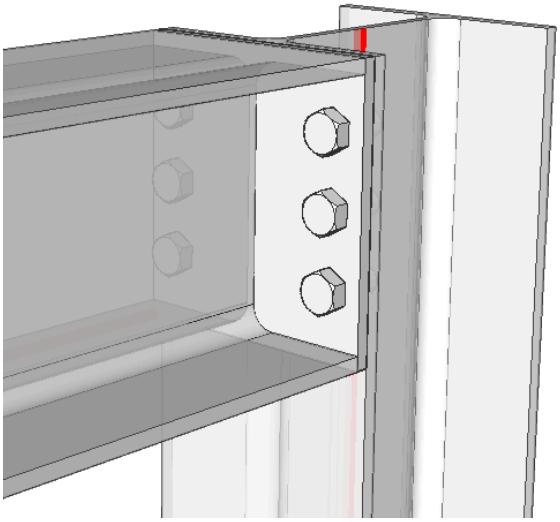


13 Add one row.

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.

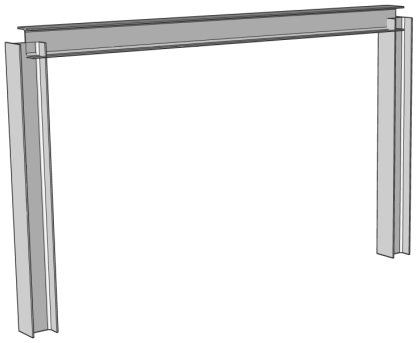
Uwaga: If the **Profiles** panel icon isn't visible yet, right-click on the ribbon and check **BIM Profiles** in the context menu under **Panels**.

3 Drag the profile you want to draw a structure with within the model space.

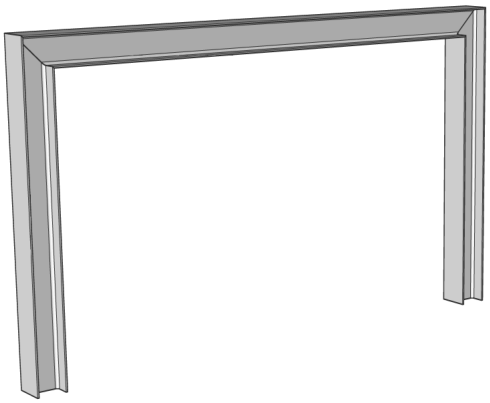
4 Draw a column with a height of 3000 mm. Don't exit the command and draw a connecting beam with

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.

Uwaga: 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.



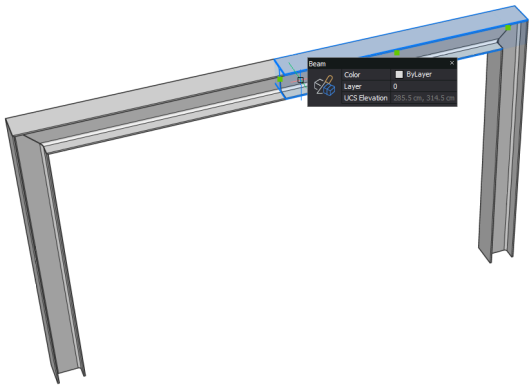
6 In the ribbon, make sure you enable the **Display sides and ends** and **Prioritize selection of Faces**. You can find them in the **Structural**/**MEP** – **View** tab or set the value of DISPLAYSIDESANDENDS system variable to 1 and SELECTIONMODES system variable to 2.

7 Check if the profiles are correctly classified as **Beam** and **Columns**.

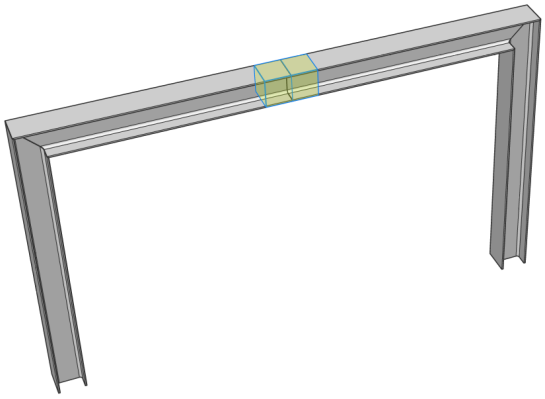
8 Launch the SLICE command and slice the beam in the middle.

9 You will be prompted: Slice along axis? Press Enter to accept the default option (Yes).

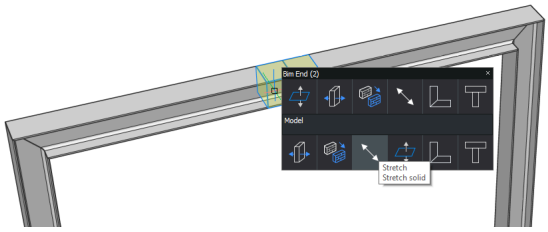
10 Select the midpoint of the beam and press Enter. Your beam is now sliced into two beams.



11 Select the 2 ends of the beams in the middle of the structure. They will be highlighted in yellow.

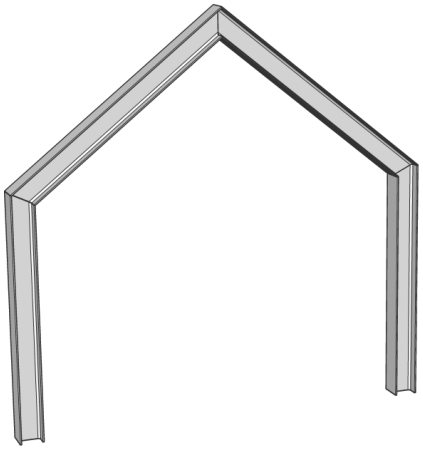


12 Quad select the BIMSTRETCH command in the **Model** tab.



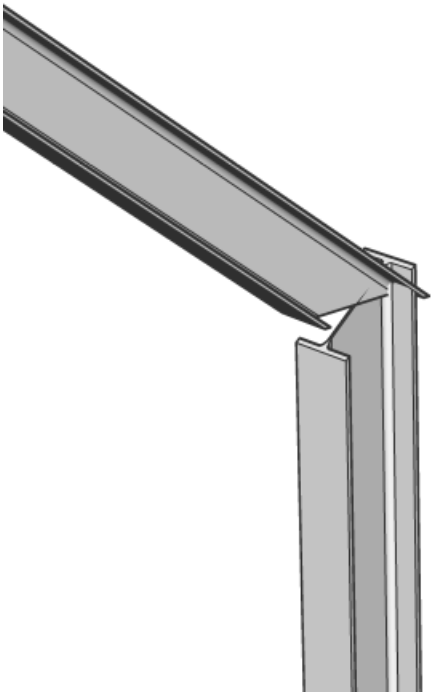
13 Hover orthogonally upwards, enter a height of 2000 mm and press Enter to accept.

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.

Uwaga: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.

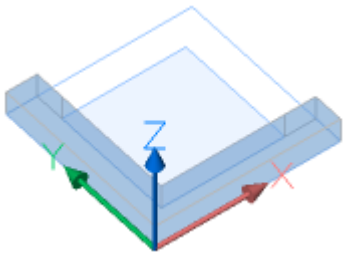


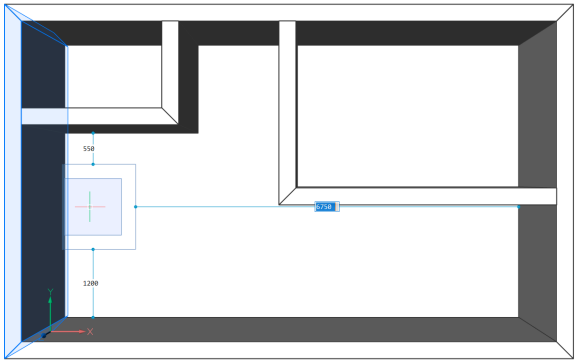
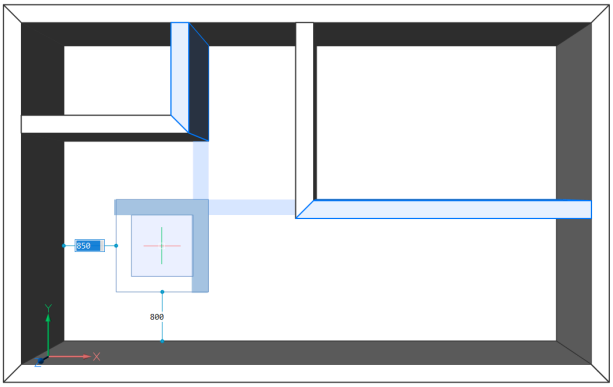
For more information about this command, visit the Command Reference article BIMQUICKDRAW.

20.3.21.3 The magnetic snapping feature

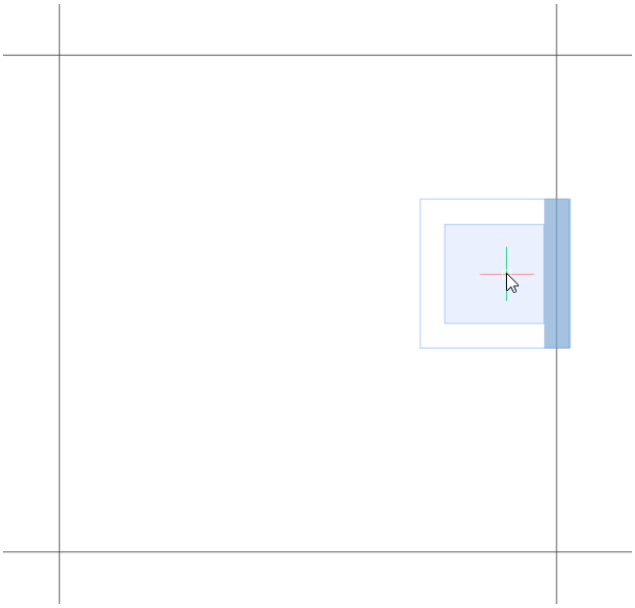
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object.

You have the following options:

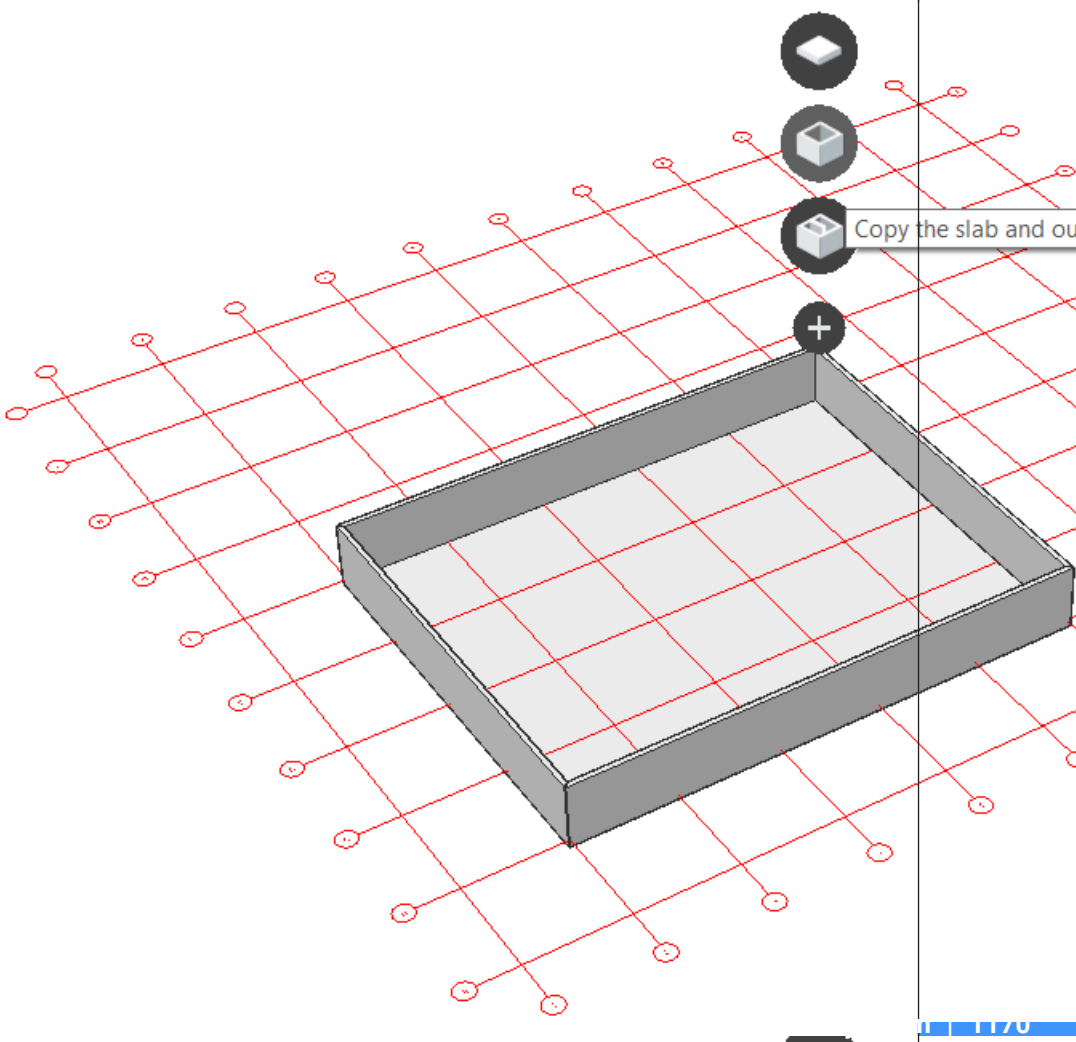
1. Align to the origin	

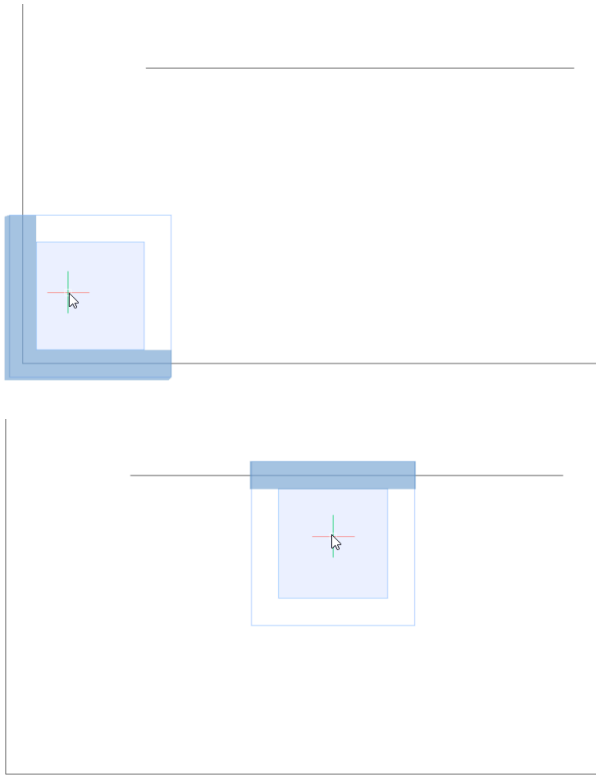
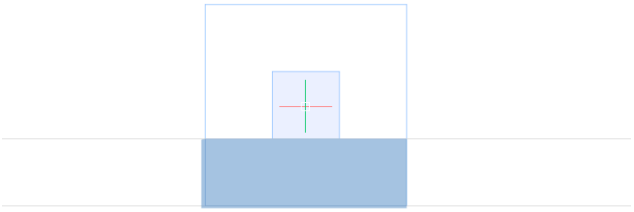
2. Align to existing walls	To the wall itself  To the extents of the wall 

3. Align to lines	To grid lines



Uwaga: 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

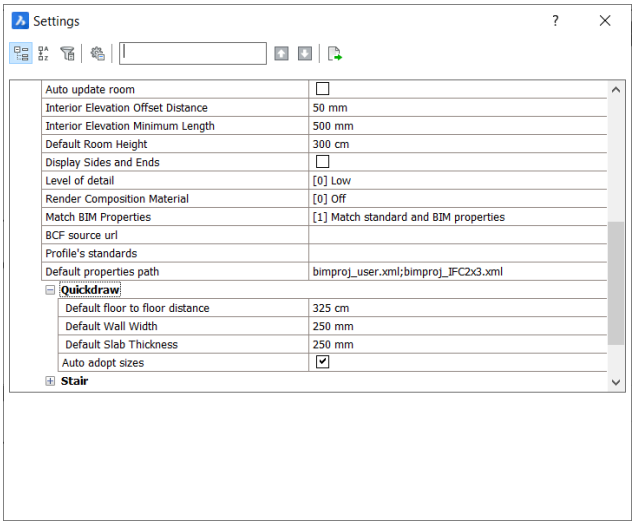
Command line.

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.

Uwaga: 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]:

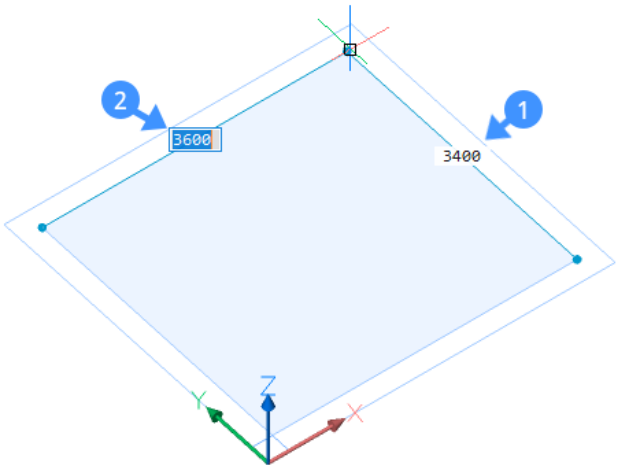
Uwaga: Use the **Quickdraw Settings** to change the wall width, wall height and slab thickness of the room before you click and then specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

Uwaga: To start from the origin, use the magnetic snapping feature of **Quickdraw**.

3 Drag the mouse cursor to define the footprint of the building.

Uwaga: 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.

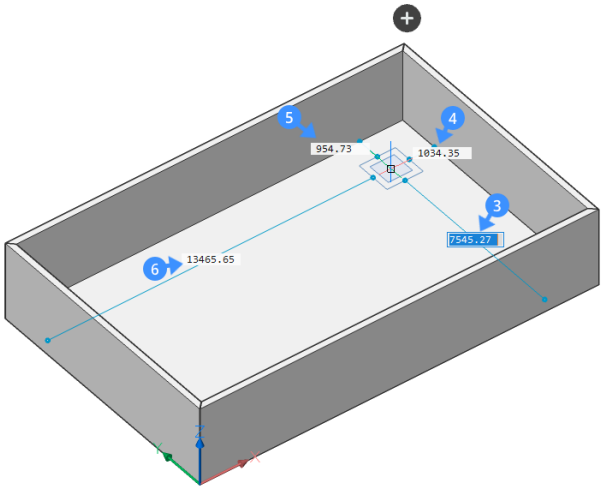


Specify the values of the room width and length in the Dynamic dimension field. Press the Tab key to switch between width and length and press Enter or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

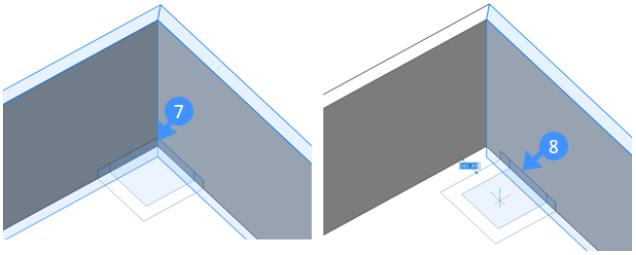
4 Place the Quickdraw cursor inside the existing building to create more rooms.

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.



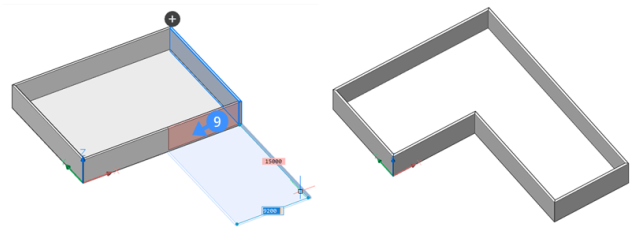
To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

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.

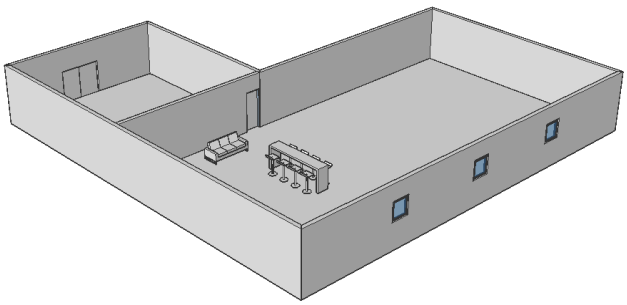
Uwaga: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

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.

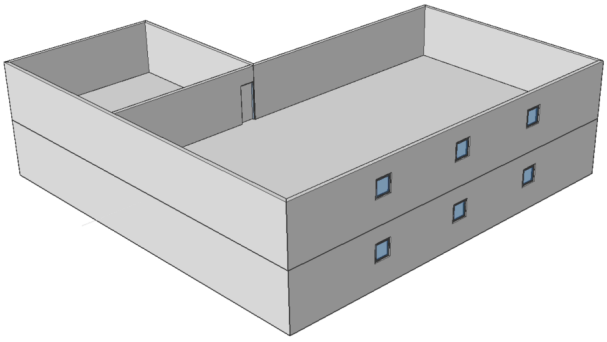
2 Click on the icon (⊕) to see the options:

- 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).

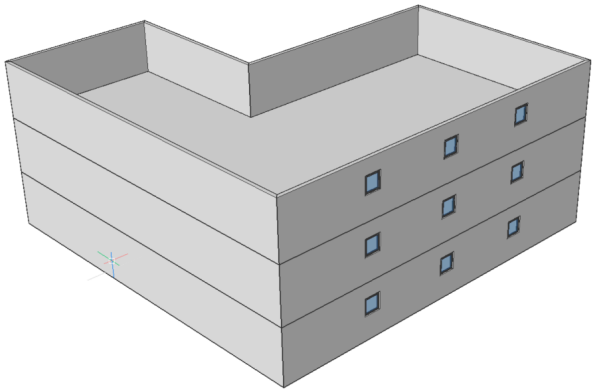


Uwaga: The interior wall and interior door and the windows in the exterior wall are copied. The door in the exterior wall on the ground floor is not copied.

Uwaga: 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.

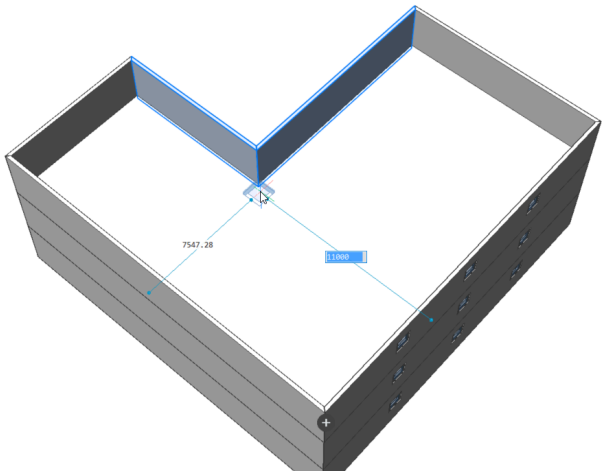
Uwaga: Walls inherit the spatial location of the slab on which they are created.

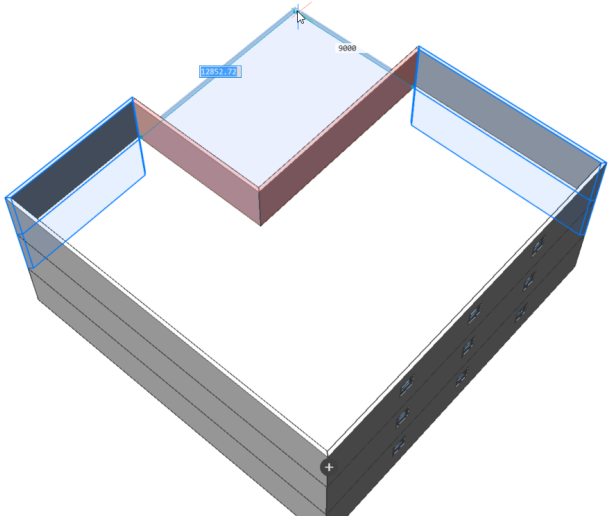
4 Repeat step 2. Choose the second option (Copy the slab and the outer walls) to create the top story.



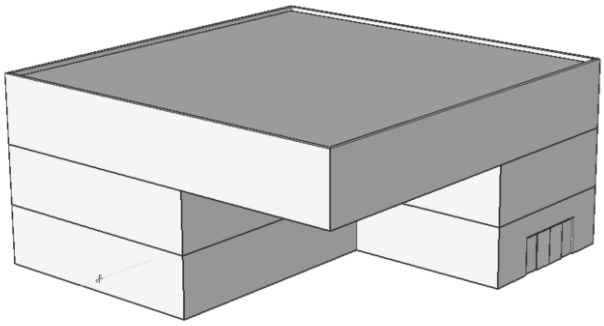
5 To make the overhang you should remove parts of the outer walls of the second floor. First, snap the Quickdraw cursor in the corner of the two outer walls, inside the building. Then click and drag the cursor outside.

6 The outer walls will turn red to indicate that the walls will be removed.



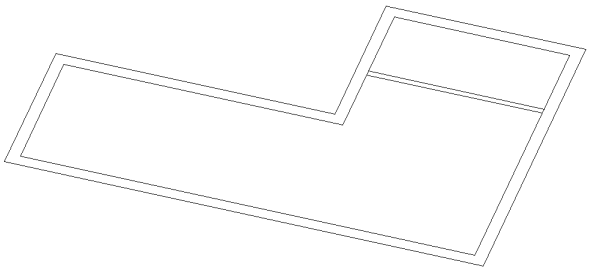


7 Use the same method as in step 2, selecting option 3 (Finish the building with a flat roof).



20.3.21.8 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

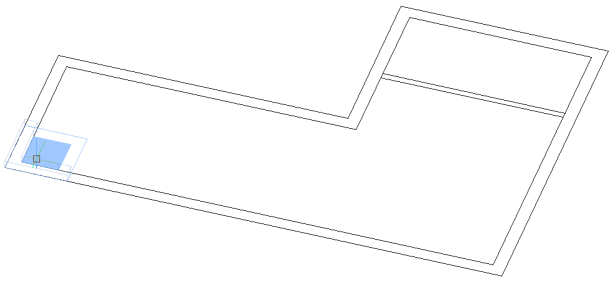


Uwaga: It is also possible to draw a simple 2D floor plan with the (POLY)LINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

Uwaga: 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.

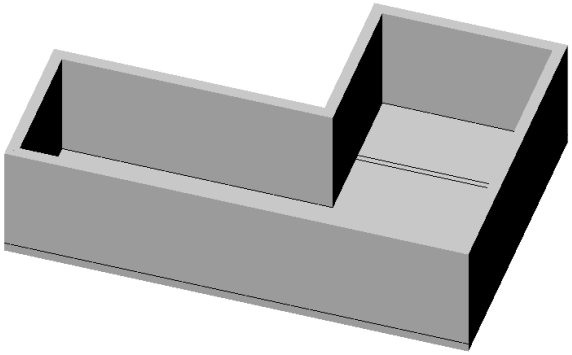
3 Hover over one of the outer walls with the cursor.



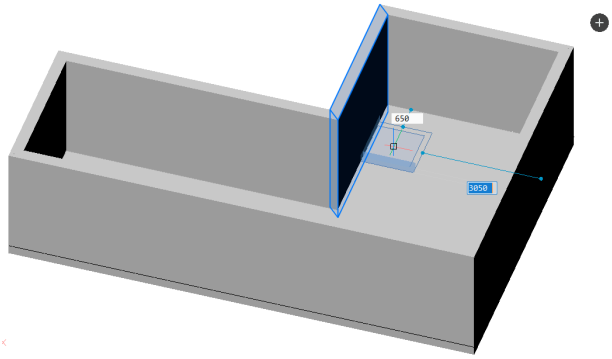
Uwaga: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

Uwaga: 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.

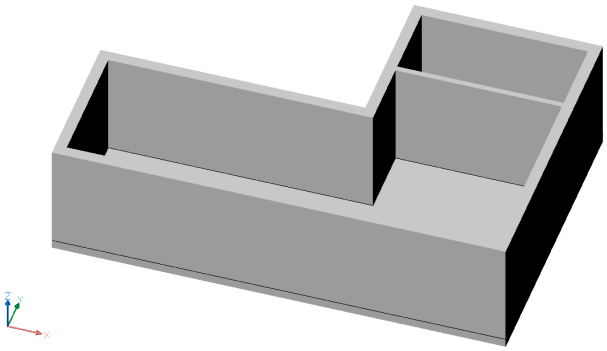


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Uwaga: 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.



Uwaga: 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

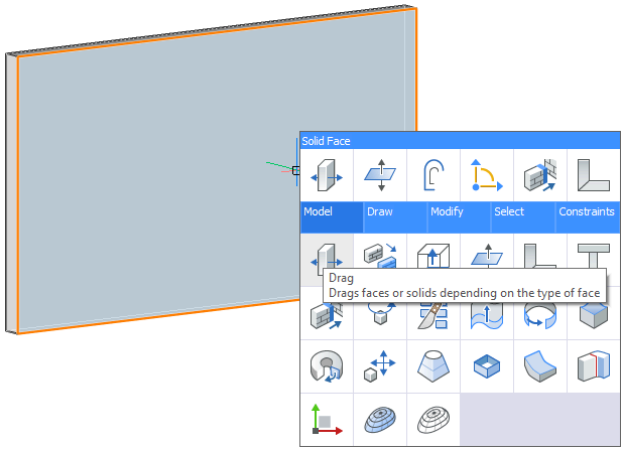
DRAG

20.3.22.2 About

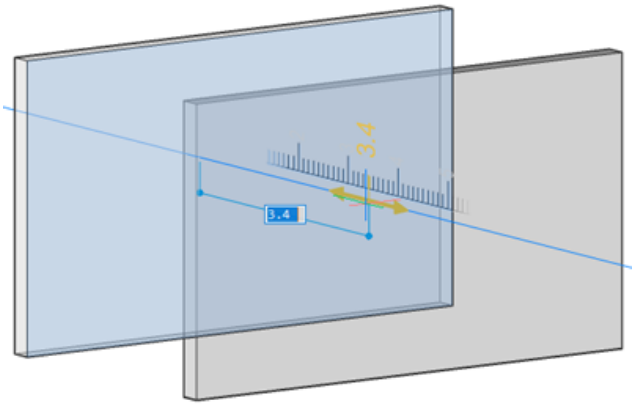
The **Drag** tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, the **Drag** tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids 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.

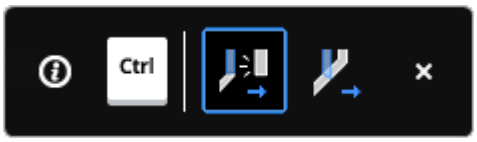


2 Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter.



Uwaga: 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.

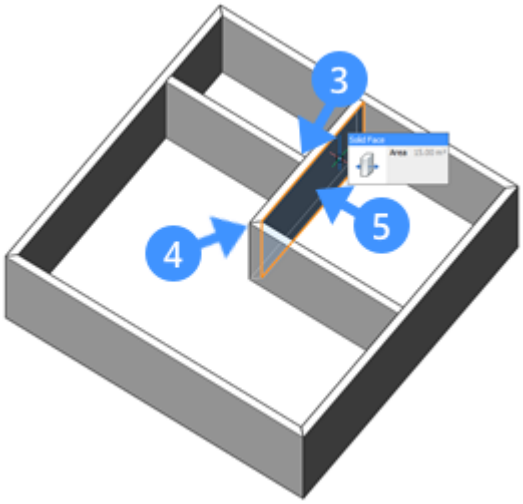


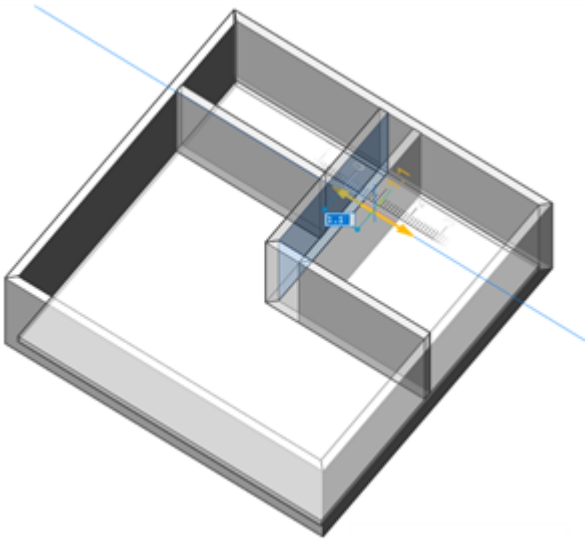
Uwaga: The current mode is indicated by a blue frame.

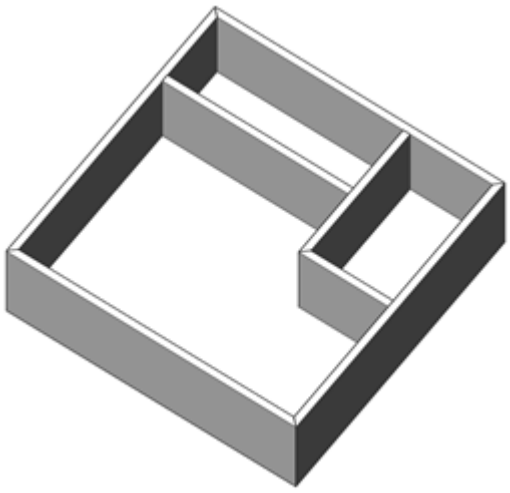
Hit the CTRL-key to toggle the connectivity mode. Alternatively, type D+Enter to disable connectivity or type E+Enter to enable connectivity.

- Select the **Enable connectivity mode** option, to move the solid while preserving the connections between solids.

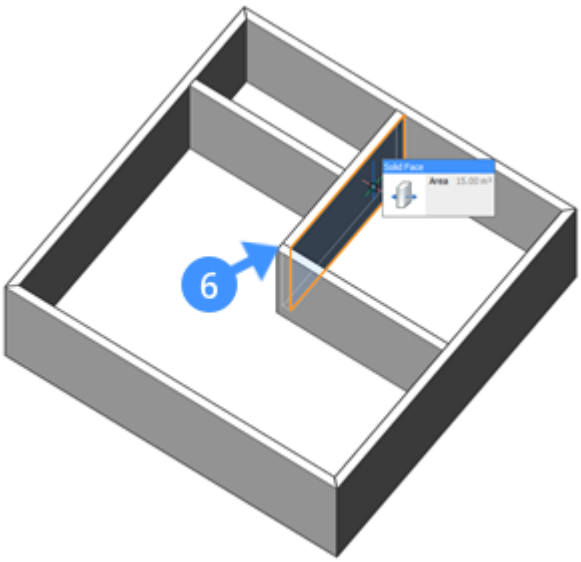
In the following illustration, the solid behavior is shown before and after applying the DRAG in **Enable connectivity mode**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5).

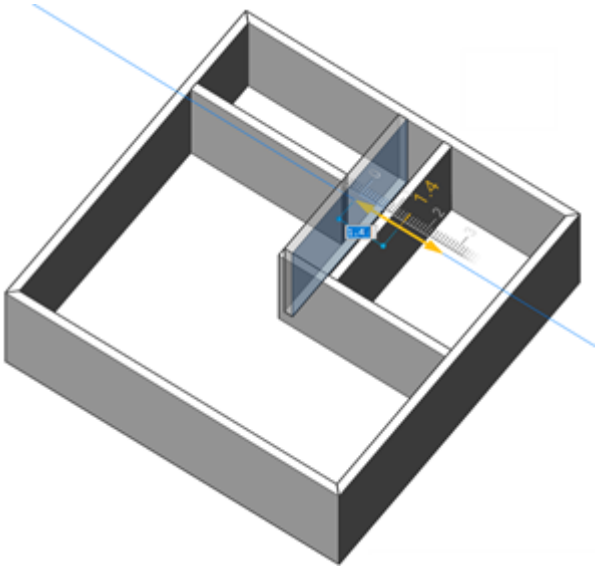


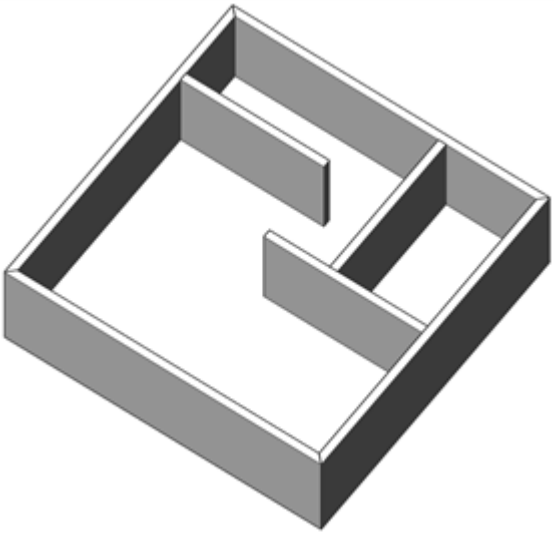




- Select the Disable connectivity mode (default) option to move the solid while breaking the connections between solids.
- In the following illustration, the solid behavior is shown before and after applying the Drag in "Disable connectivity mode". The connectivity between walls, i.e. T- (3) and L- (6) 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.

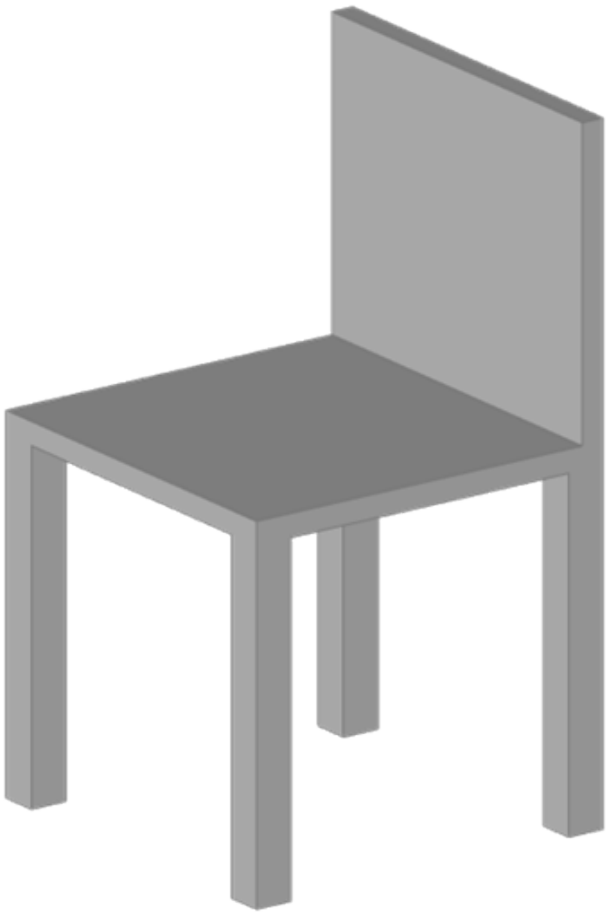
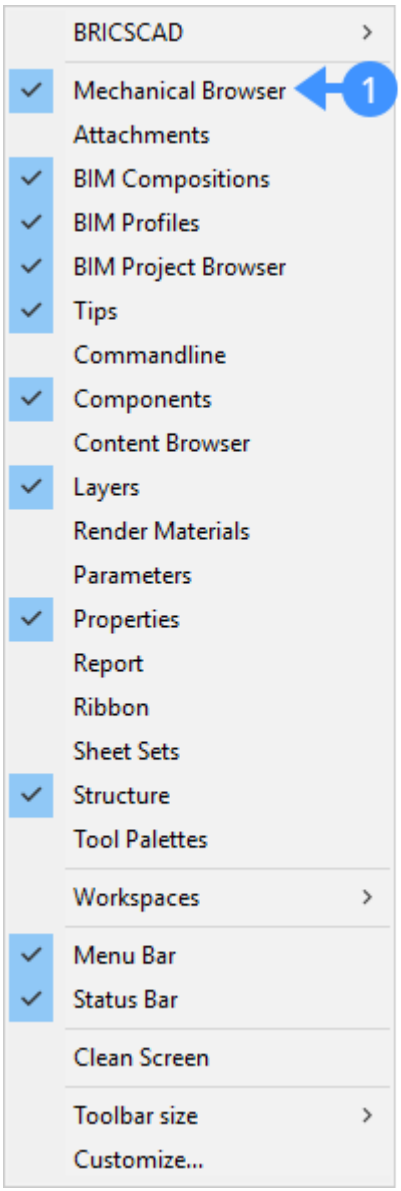


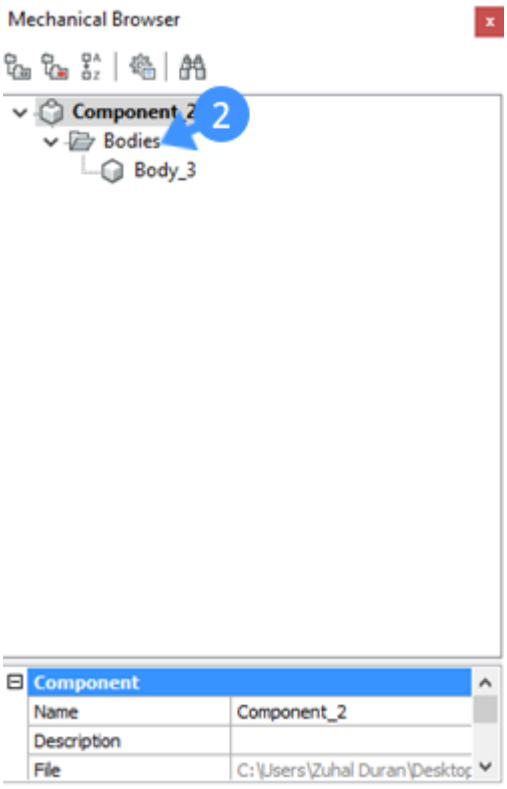
Figure 1, a geometric model of the sample chair.

2 Open the **Mechanical Browser** by clicking its icon in the **Toolbar** panel or type the MECHANICALBROWSEROPEN in the Command line.

Uwaga: 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.

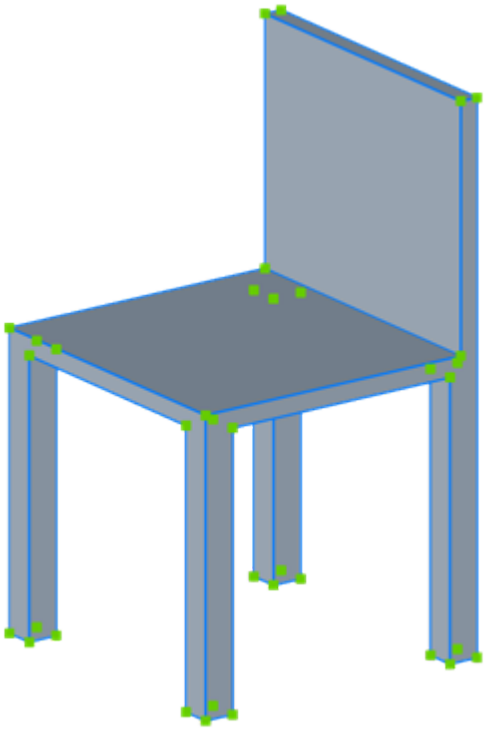


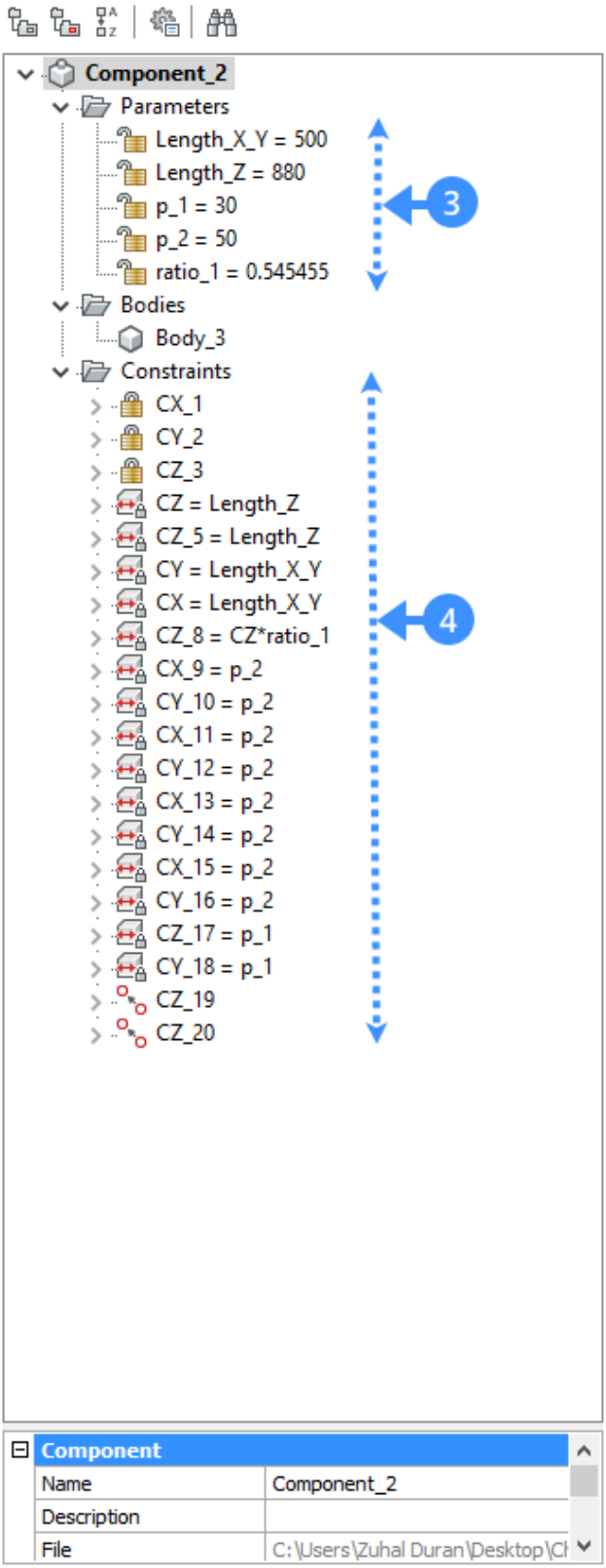
Figure 2, selected object.

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.

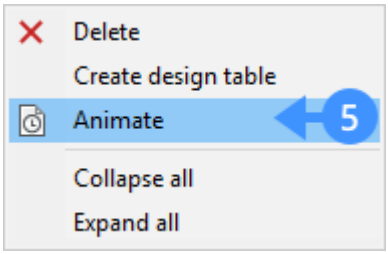
These parameters and constraints are shown in the **Mechanical Browser** (3 and 4).


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

Uwaga: **Parametrize** creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints depending on the geometric model.

Uwaga: 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.



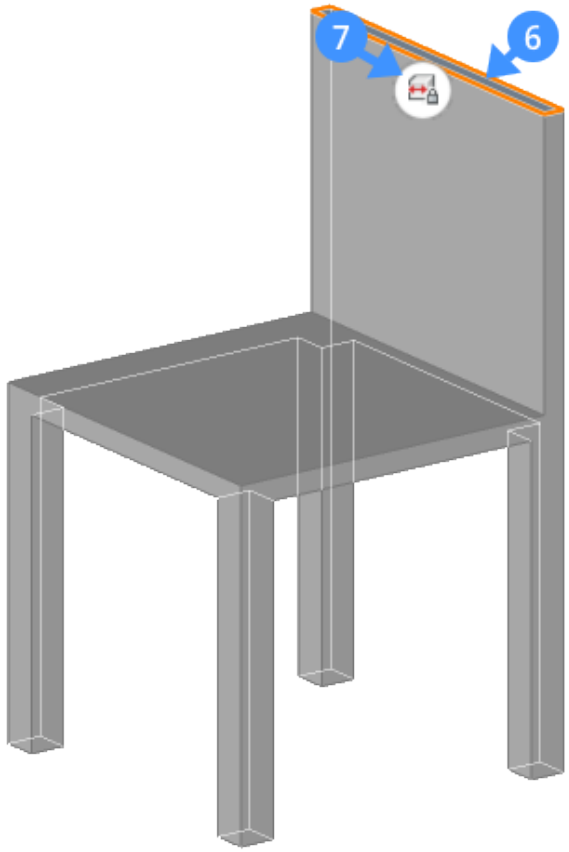
Uwaga: In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the **Mechanical Browser**. Each parameter corresponds to one measurement in the chair geometry. For instance, **Length_X_Y** controls the width and depth, **Length_Z** controls 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

Uwaga: After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

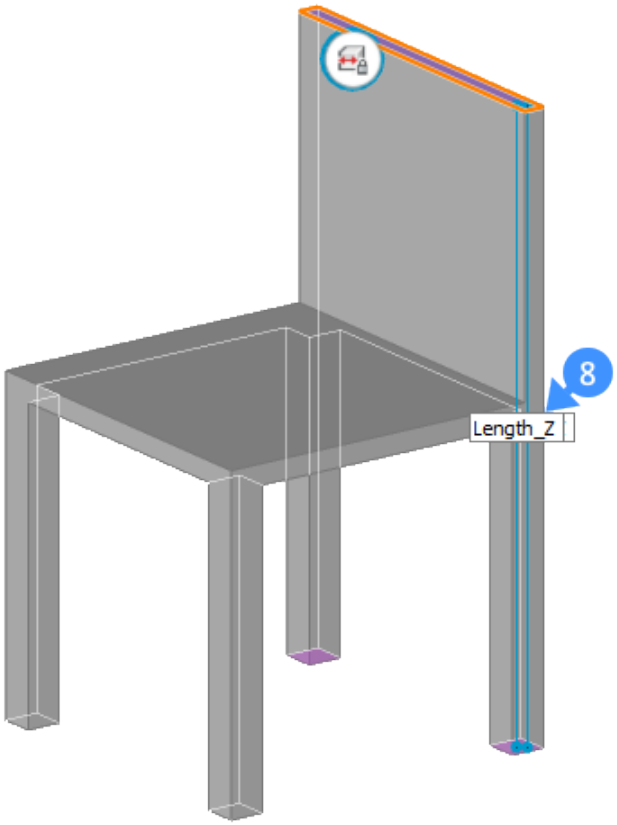
1 Select one of the faces (6) of your component. The constraint type (e.g. distance, fix) regarding the

face is displayed.

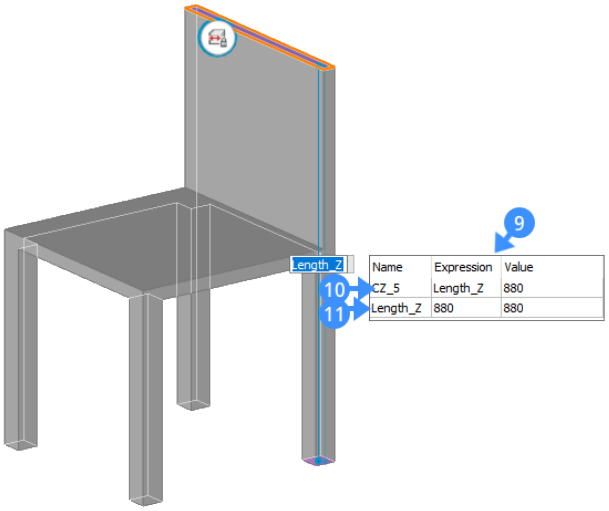


2 Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

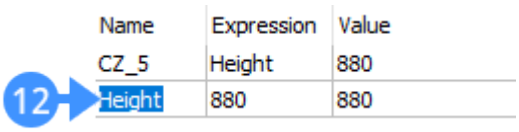
3 Click the editable dimension box (8).



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.



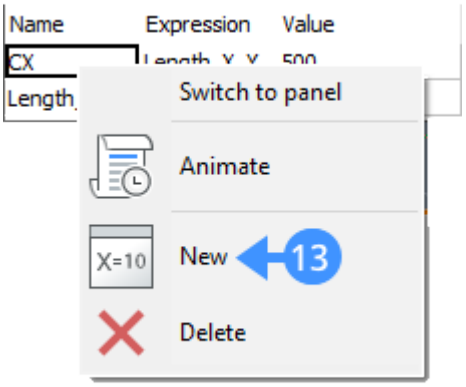
4 Change the parameter name by clicking its field (12) in the parameters manager.



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

Uwaga: The above procedure can also be done using the **Mechanical Browser**.

20.3.24 Working with parameters and constraints

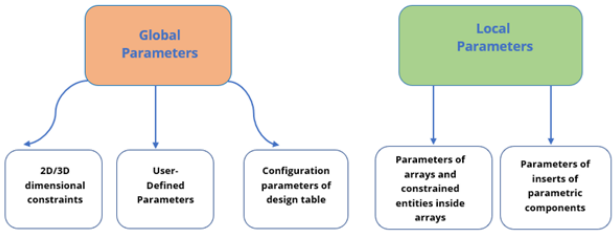
20.3.24.1 Parametric modeling

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.

Uwaga: 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.

Uwaga: 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

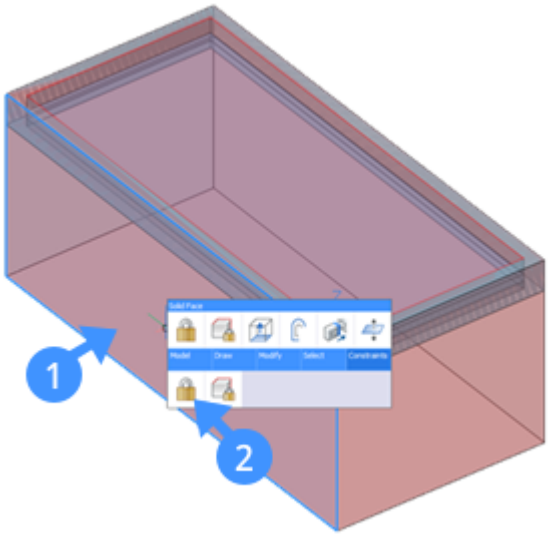
Command: DMFIX3D

1 Turn on the **BIM_Subtract** layer.

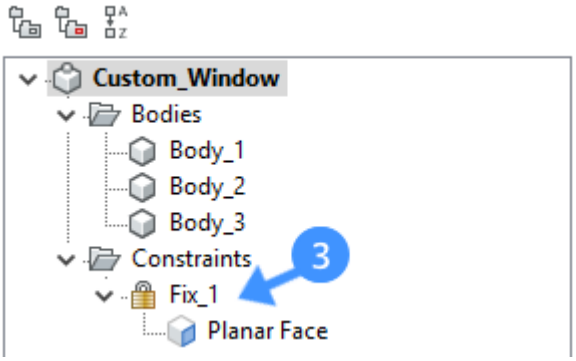
2 Select the entities (1) you want to add a fixation to.

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

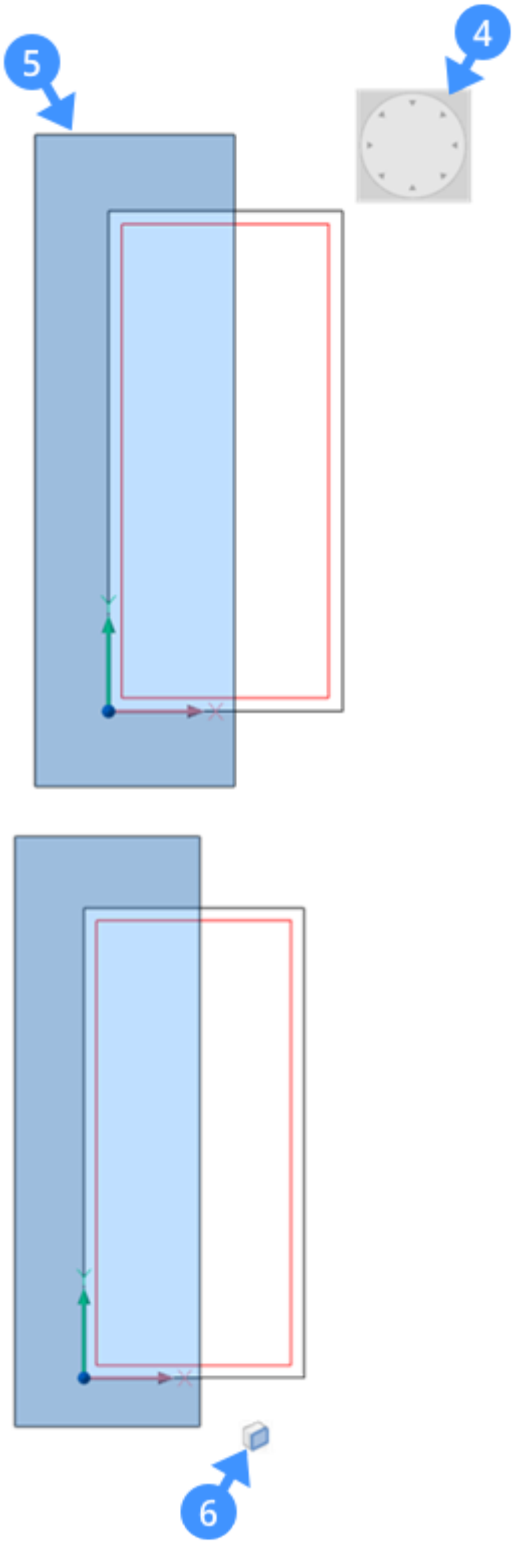
1 Select the entities to apply the **Rigid Set** constraints.

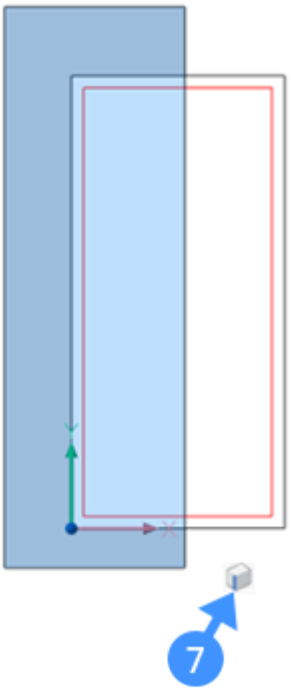
The entities remain in the same position with respect to each other when they are in a Rigid Set.

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).

Uwaga: 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.

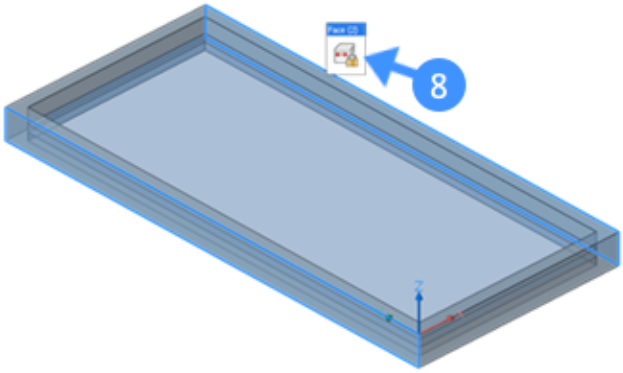
2 Select the first face.

3 The face highlights.

4 Hover over the second face.

5 Hit the TAB key to highlight an obscured face.

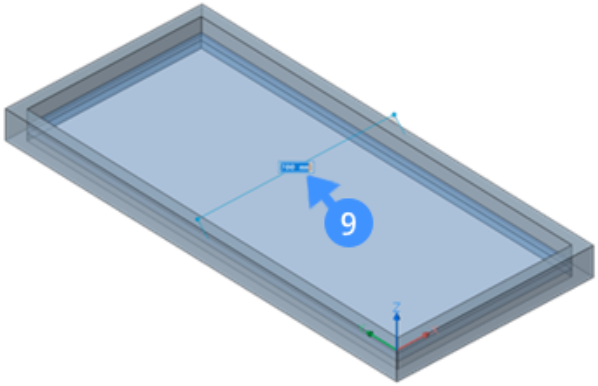
6 When the second face highlights, choose **Add Distance** (8) in the 3D Constraints command group in the Quad.



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**.

Uwaga: 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.

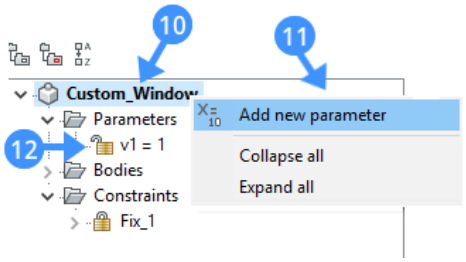
Step 5: Applying parameters

We will create a parameter that controls the width of the window.

1 Right-click the file name (10) in the **Mechanical Browser**.

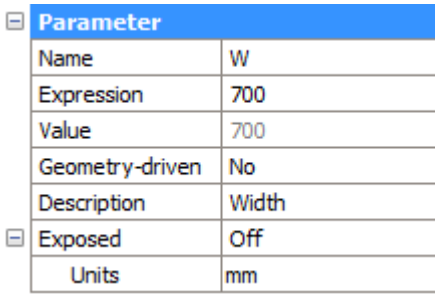
2 Select **Add New Parameter** (11) in the context menu.

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 Dane budynku

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.

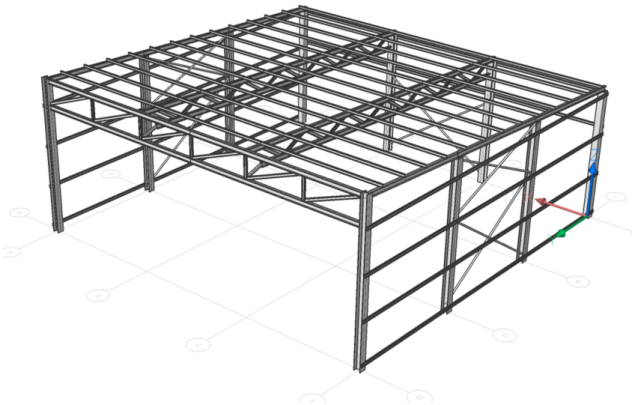
20.4.1.3 Procedure: creating an analytical model with strict deviation settings

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.

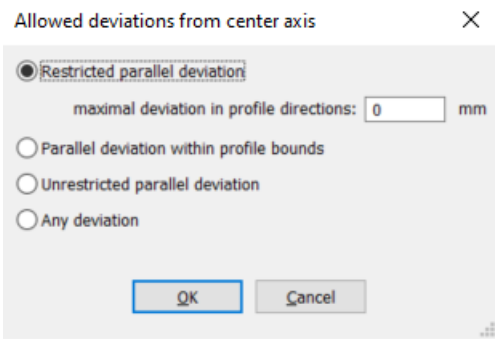
You are prompted: *run Bimify on entire model or [Advance] <model>*

2 Press Enter if you want to classify everything in your model or type **A** if you want a selection of elements.



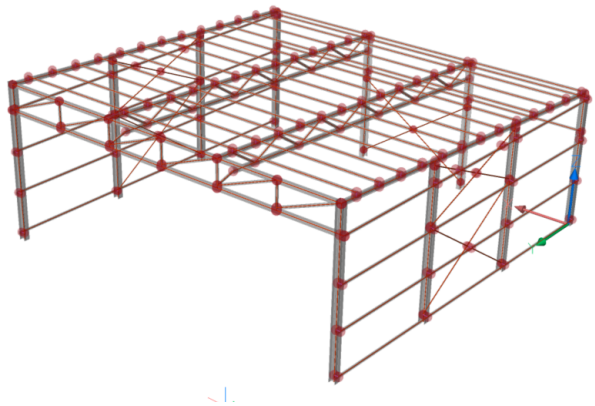
3 Launch the BIMANALYTICALMODEL command in the Command line.

4 A pop-up window opens.

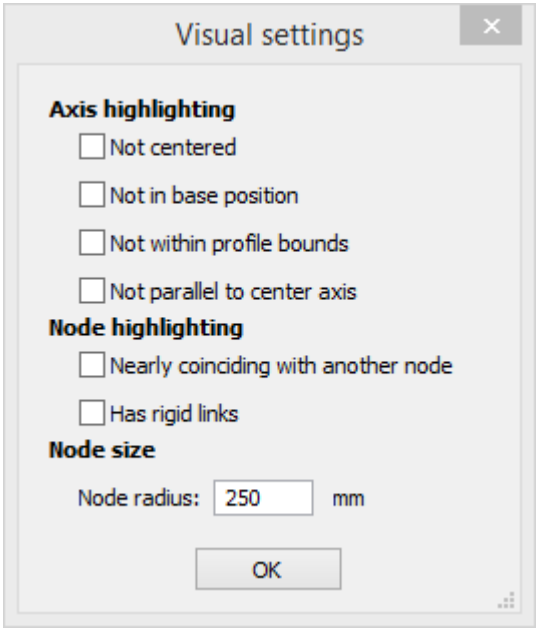


This window allows you to set the level of freedom used when generating an analytical model. When hovering over one of the options, a tooltip shows you what type of restrictions are imposed for that option. In this case, choose the **Restricted parallel deviation** with a maximal deviation in profile directions of 0 mm. No axis is allowed to deviate from its central position in the profile.

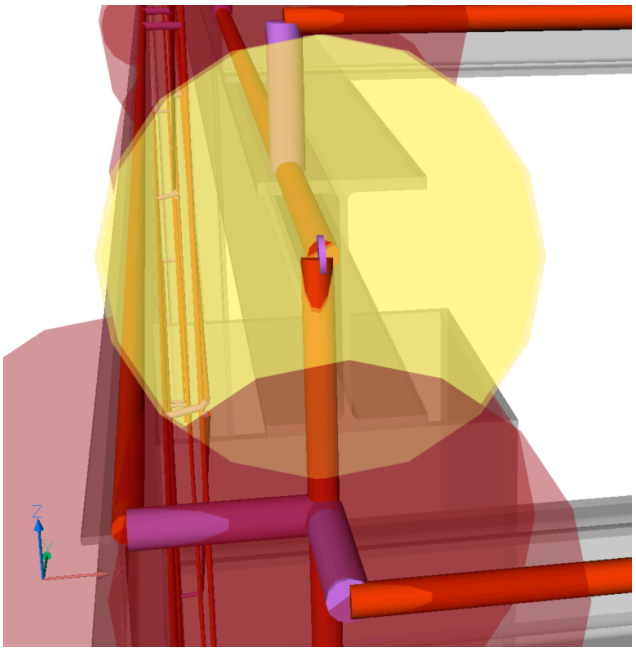
5 An interactive preview of the analytical model is constructed based on the chosen global deviation setting.



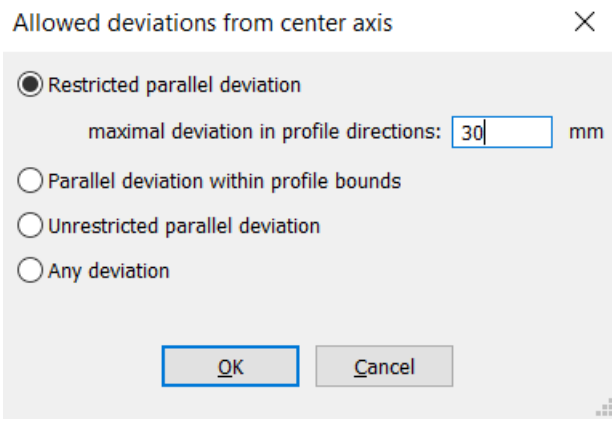
Also, the visual settings window appears. In this window, you can check and uncheck different features of axes and nodes to highlight (in yellow) the axes and nodes which correspond to those features. This allows you to explore the current properties of the axis and nodes of the analytical model. Based 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.

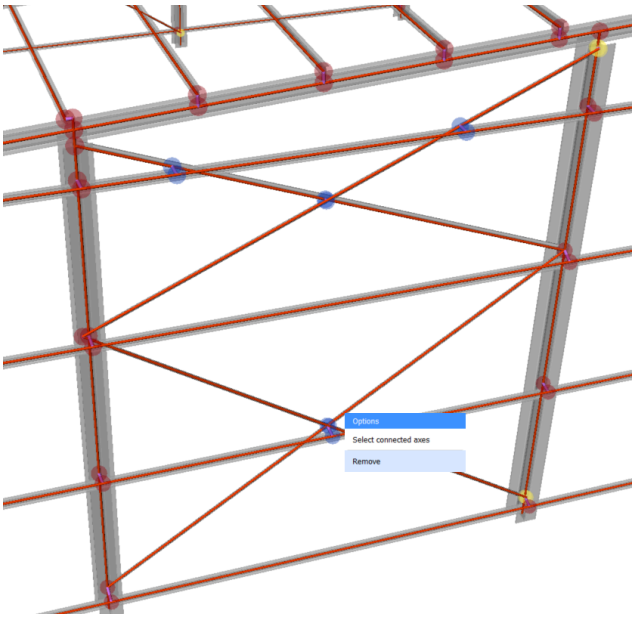


To avoid this, restart the command with a slightly more flexible setting. In the **Allowed deviations from center** window, choose again for **restricted parallel deviation** but this time with a maximal deviation in profile directions of 30 mm.

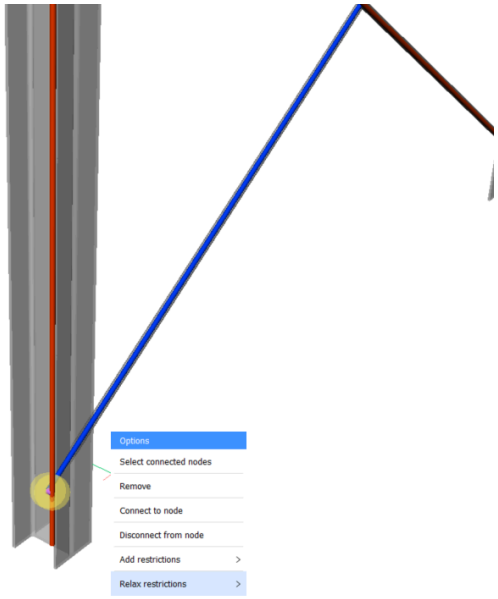


7 For the detection of nodes in the preview of the analytical model, the proximity of structural elements is used. Because of this, it is possible that there arise nodes in the model where tie rods cross. You can easily remove these nodes by clicking on the node and select **Remove**.

Uwaga: You can select several nodes at once using box selection.

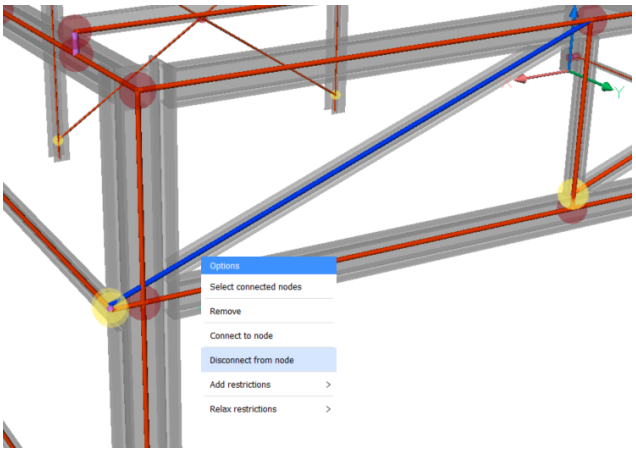


8 If you take a look at the remaining coinciding nodes, you see that the tie rods are not aligned perfectly with the center lines of the columns and small rigid links are formed. You can remove these rigid links by relaxing the position of the axes of the tie rods to a non-parallel position. You can 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.

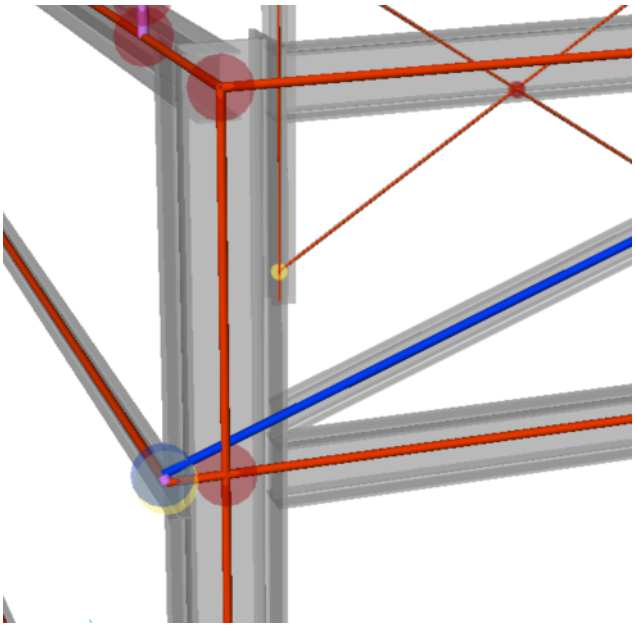
10 You can disconnect the tilted member by clicking on it and choose **Disconnect from node**.



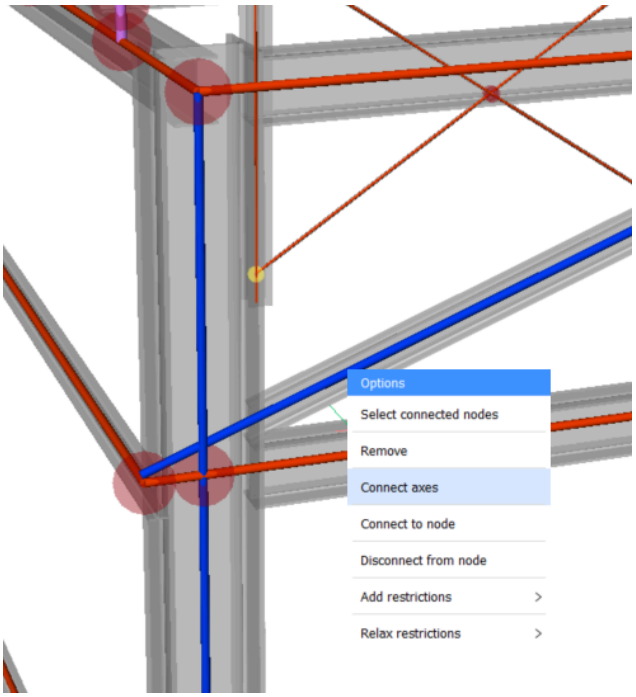
You are prompted:*Select node disconnect from axes [Ok]*

Select the node you want to disconnect from.

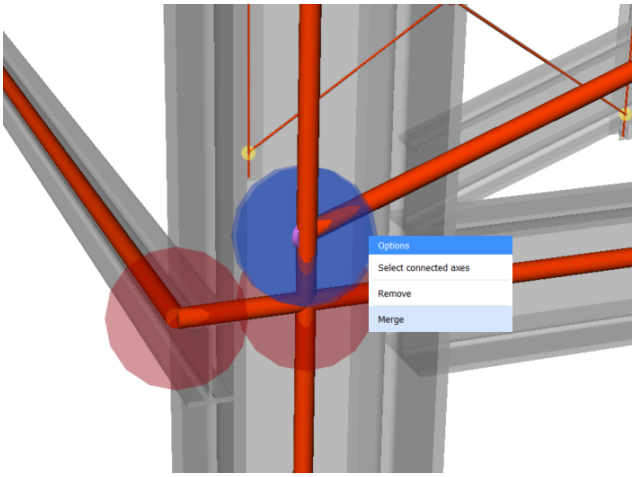
Type **O** to choose Ok.



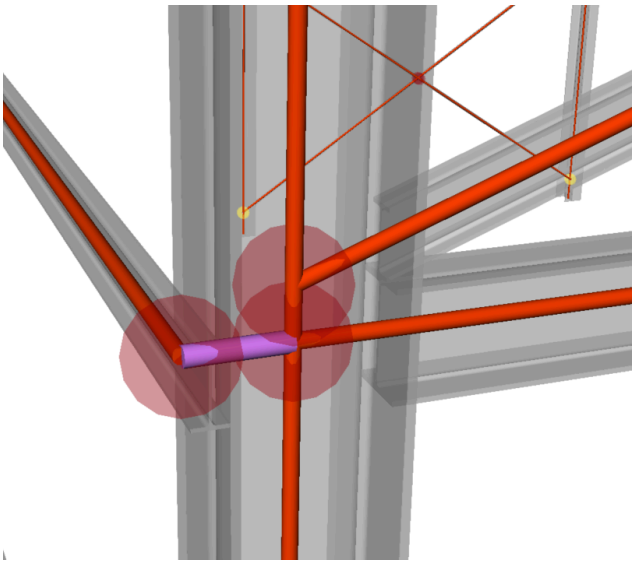
Now select the two axes that you want to connect and click **Connect axes**.



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**.

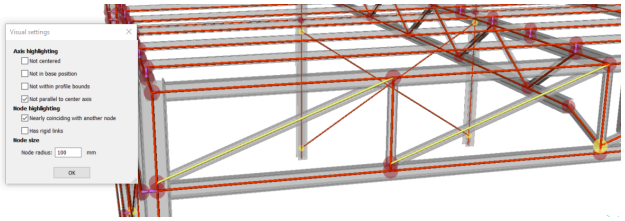


11 You see that the side member also is attached to the front beam instead of the column. Repeat step 9 for the side member.



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.

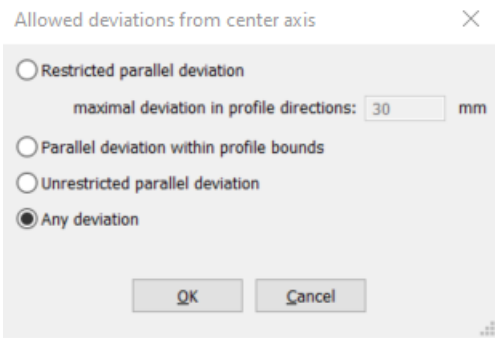
Uwaga: If you take a look at the axis **Not parallel to center axis** by using the **Visual settings** window, you will see that the tilted member in the front truss has been allowed to be non-parallel to the center axis. So, you could also remove the small rigid links by relaxing the tilted member to 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

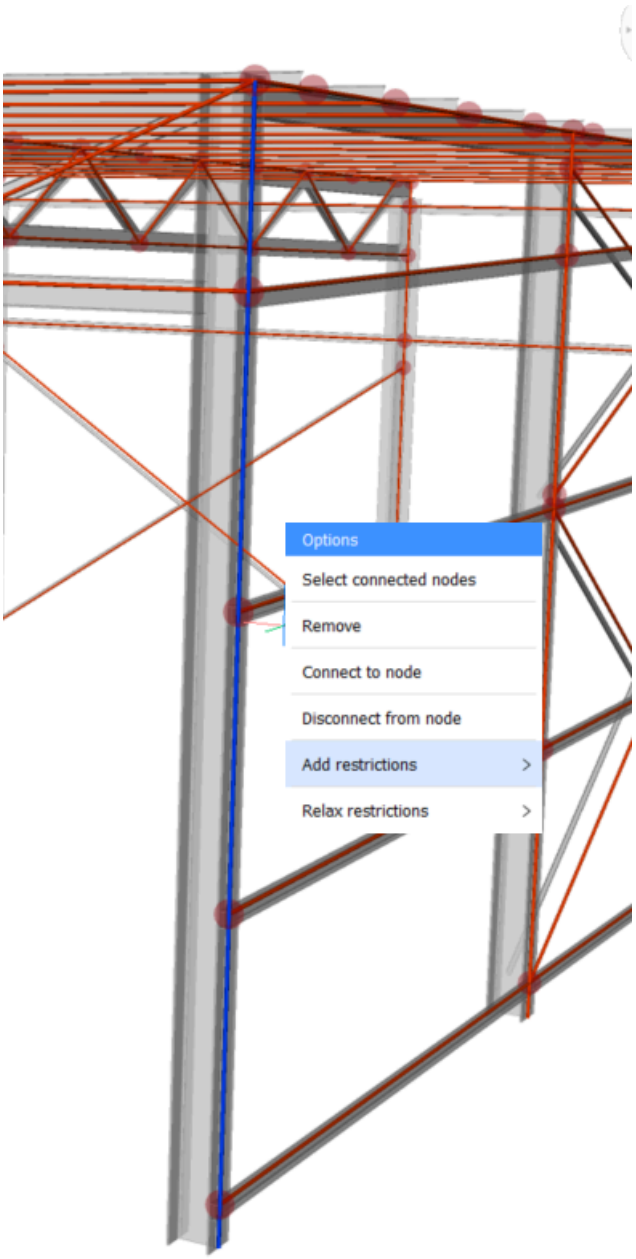
1 Repeat the steps from the previous procedure until step 4.

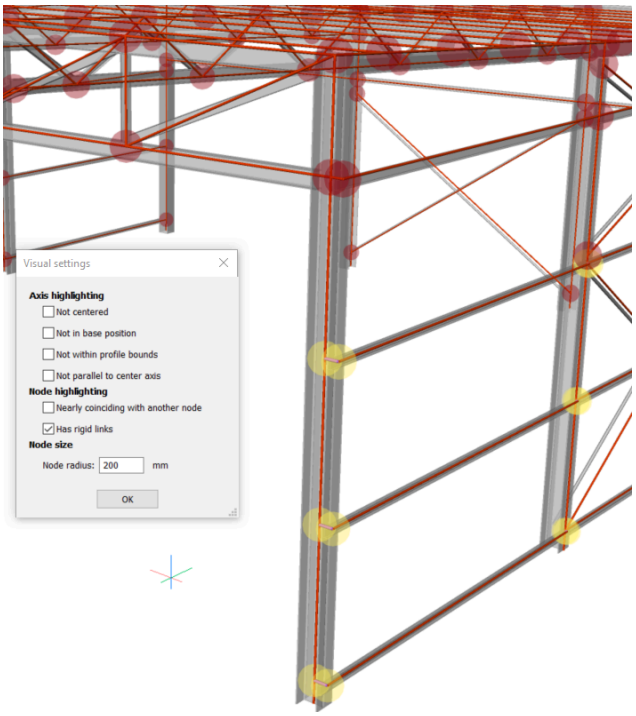
2 Choose **Any deviation** in the **Allowed deviations from center axis**. This allows all kinds of freedom to avoid any rigid links. In the preview of the analytical model, you will see that some axes have been tilted. For example, the small members in the trusses are not centered and the column axis 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.

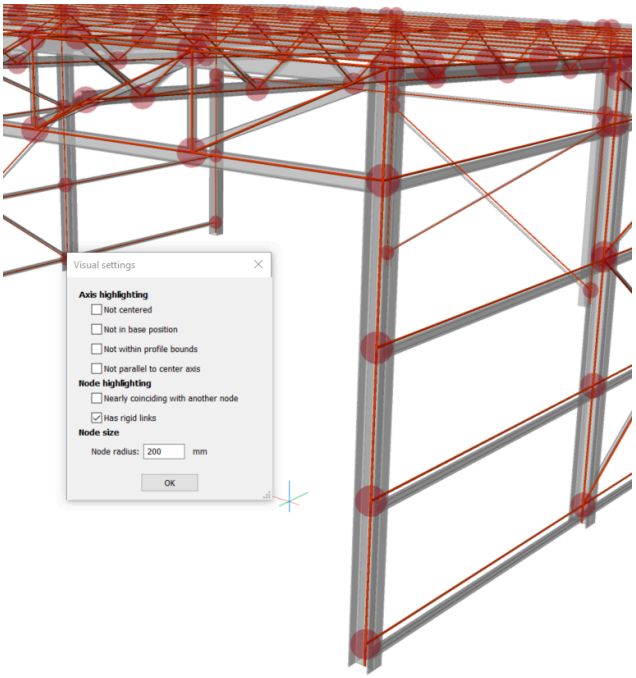
4 In the interactive preview of the analytical model, the column axes were moved towards the profile bounds. In this step, you will center the axis of the columns. You can do this by clicking on the axis of the column, click **Add restrictions** and click **To center**. This introduces rigid links very 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**.

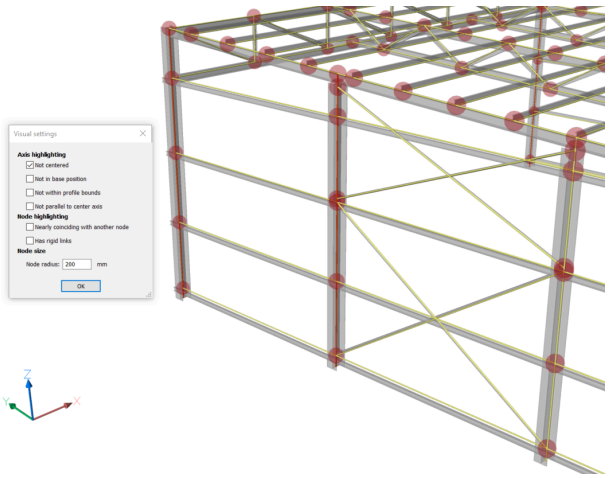
Type **R** in to choose **Recalculate**.



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

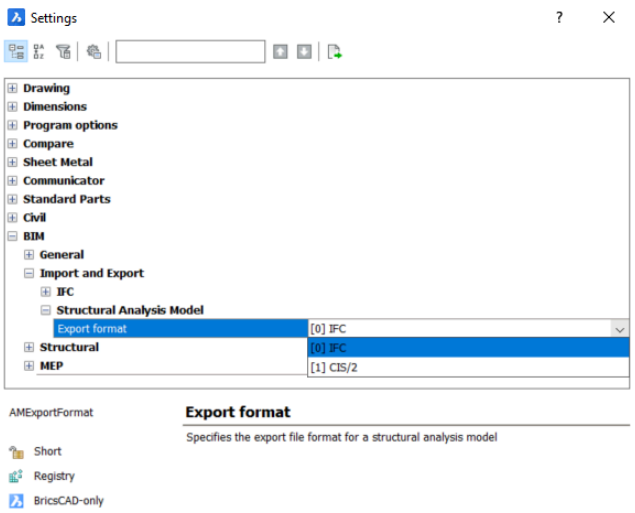
restriction on this column as well.



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.

Uwaga: 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.



Uwaga:

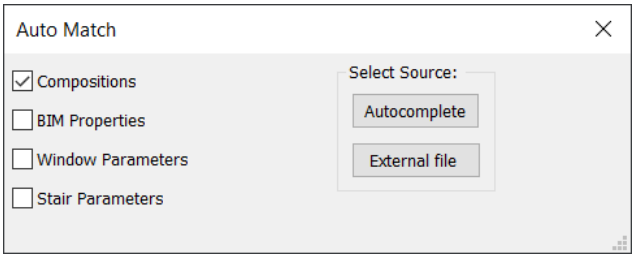
Before you use BIMAUTOMATCH make sure that you have Bimified ( ) everything. AutoMatch works best with a fully Bimified model.

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:



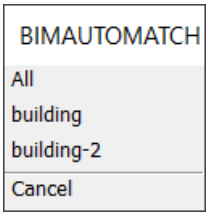
2 Select one of the following:

- **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.

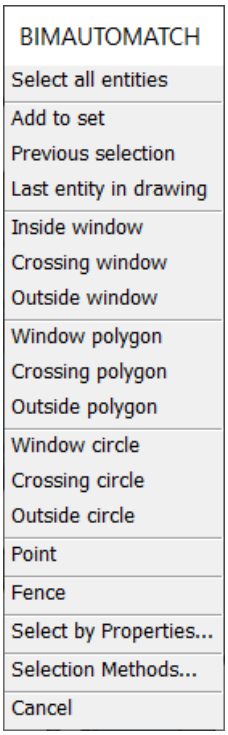
5 **You are prompted:**



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.

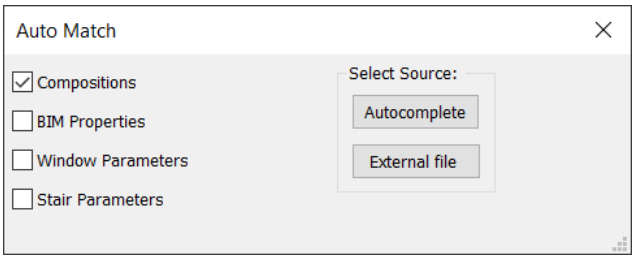
7 **You are prompted:**



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:



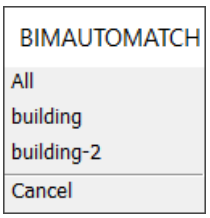
2 Select one of the following:
- **Compositions**: Copy compositions.
- **BIM Properties**: Copy BIM Properties.
- **Window Parameters**: Copy parameters of windows.
- **Stair Parameters**: Copy properties of stairs.

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.

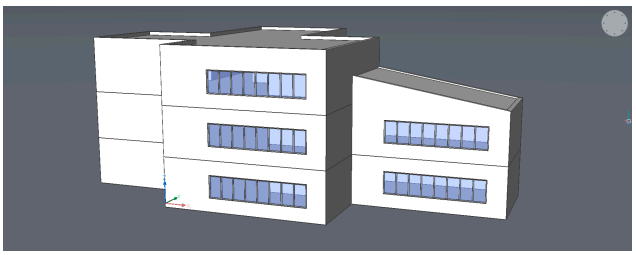
6 **You are prompted:**



7 Select the building from the menu.

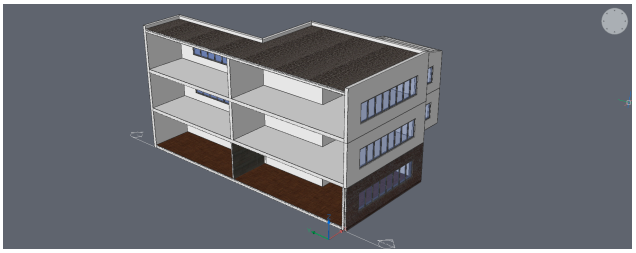
20.4.2.5 Procedure: using autocomplete in one file

1 Open up a file to start from. Here we use a basic rectangular building that looks like this:



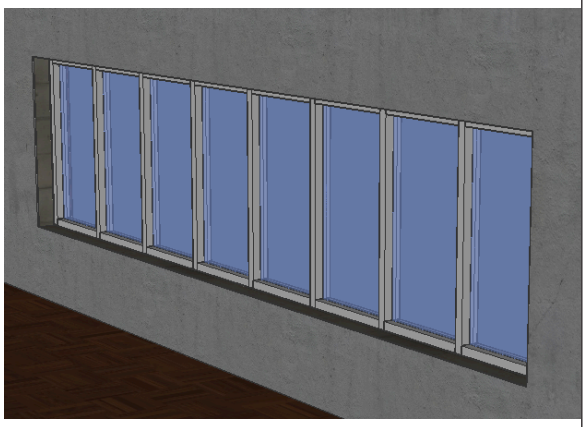
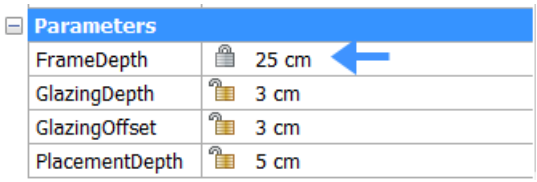
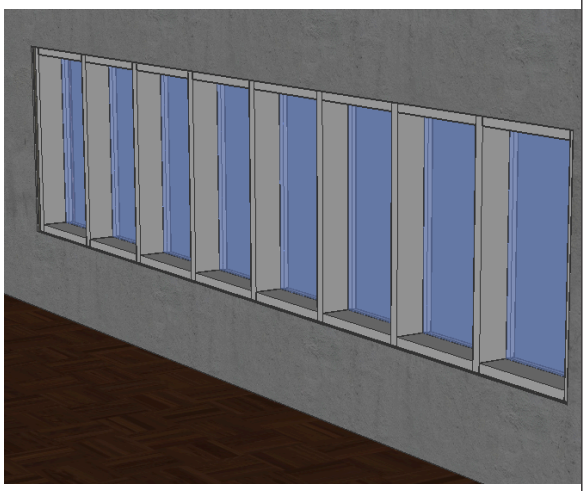
Uwaga: No BIM data or compositions should be added yet.

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:



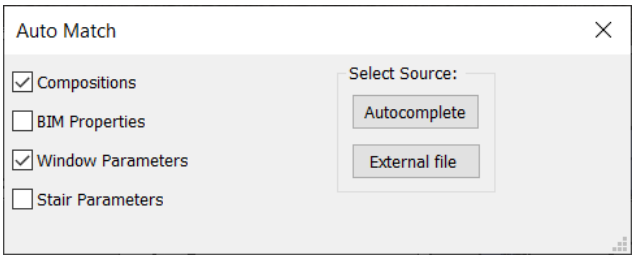
Uwaga: It could be that you have different types of outer walls, for example the wall facing north that needs more insulation. In this case, you give one of these northern walls a different composition.

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.

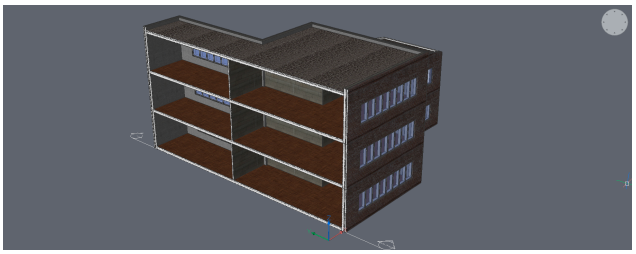
4 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

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:

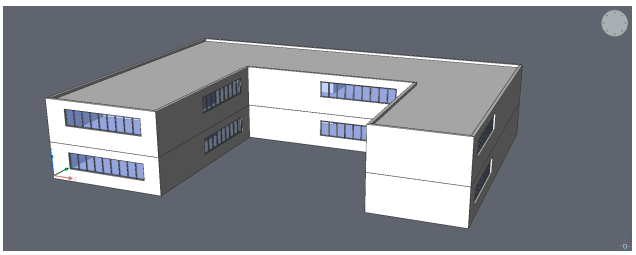


By using BIMAUTOMATCH, we very quickly have all our compositions and window parameters as we want them, without having to worry about forgetting anything.

Uwaga: You can see that different types of outer walls are no problem. All the walls facing north have been given the correct adjusted composition.

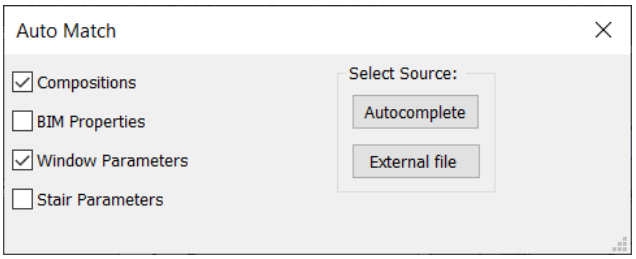
20.4.2.6 Procedure: using autocomplete with an external file

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.

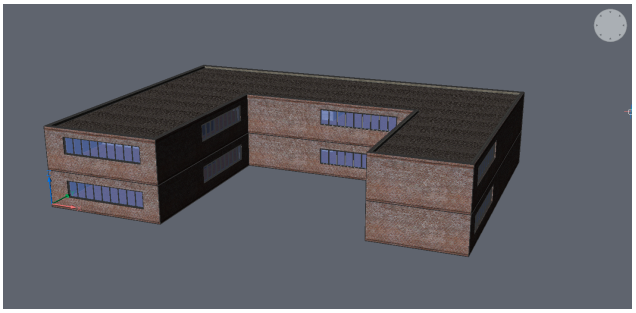
The following window appears:

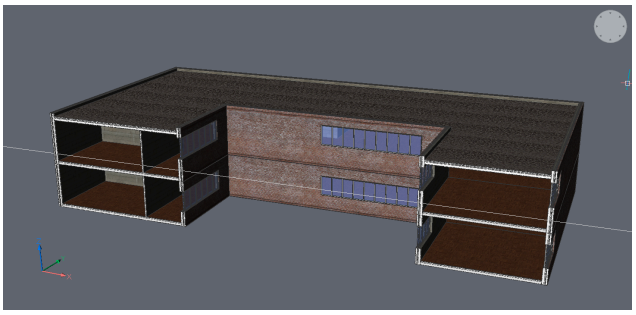


We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

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:





Uwaga: 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:

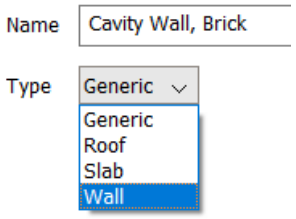
- Type a composition name in the Command line and press Enter.

  Uwaga: 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.

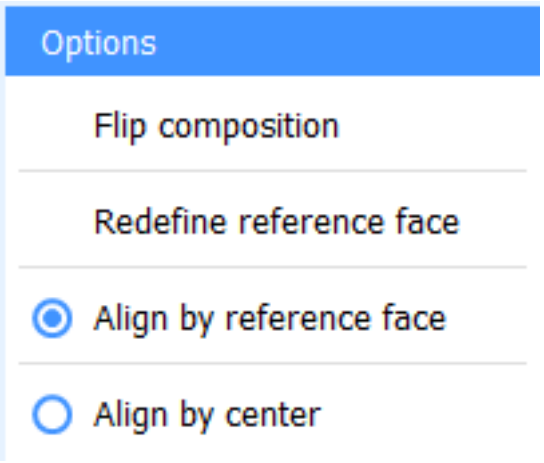
  If you want to select the composition without opening the menu, double-click it.

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).

Uwaga: 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.

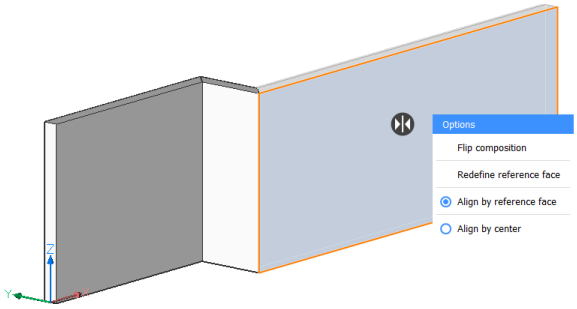
2 Hold down the left mouse button and drag the composition on the solid.

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.

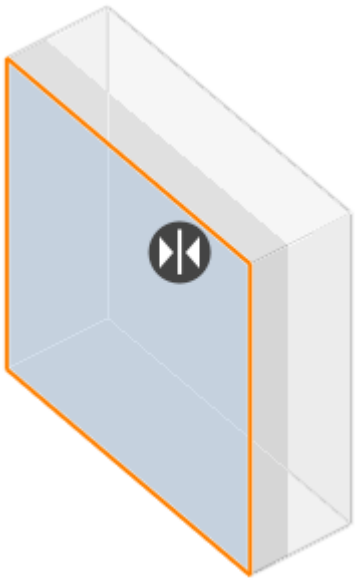
5 A widget appears that allows you to choose the reference face.

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.

Uwaga: 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

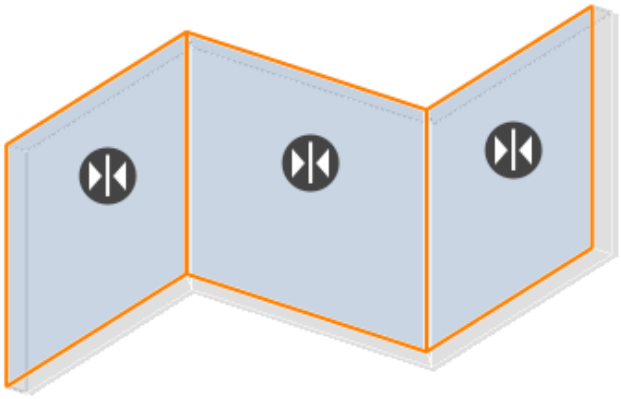
1 Select the solids you want to apply the composition to.

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.

Uwaga:

- 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.

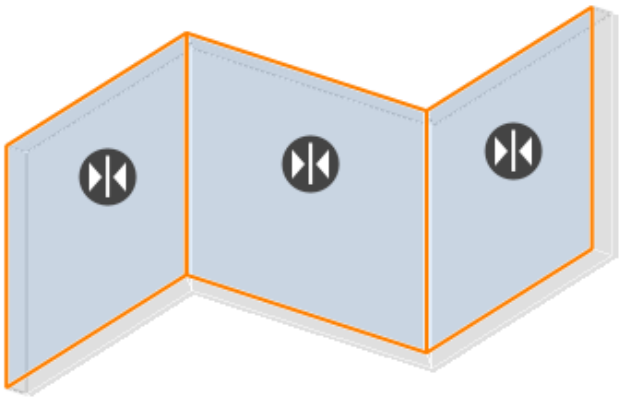
2 Select the **Entity** option.

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.

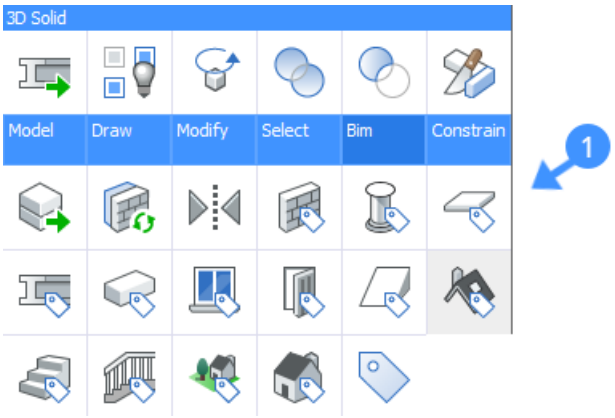


For more information about this command, visit the Command Reference article BIMCLASSIFY.

20.4.4.3 Classifying entities

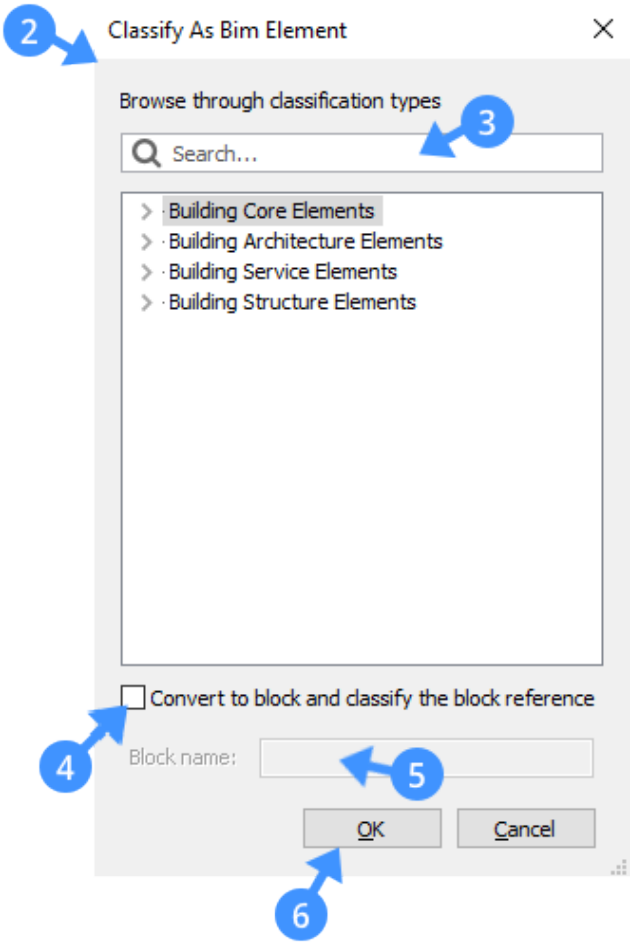
1 Select an entity in the drawing to classify.

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):



Uwaga: 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

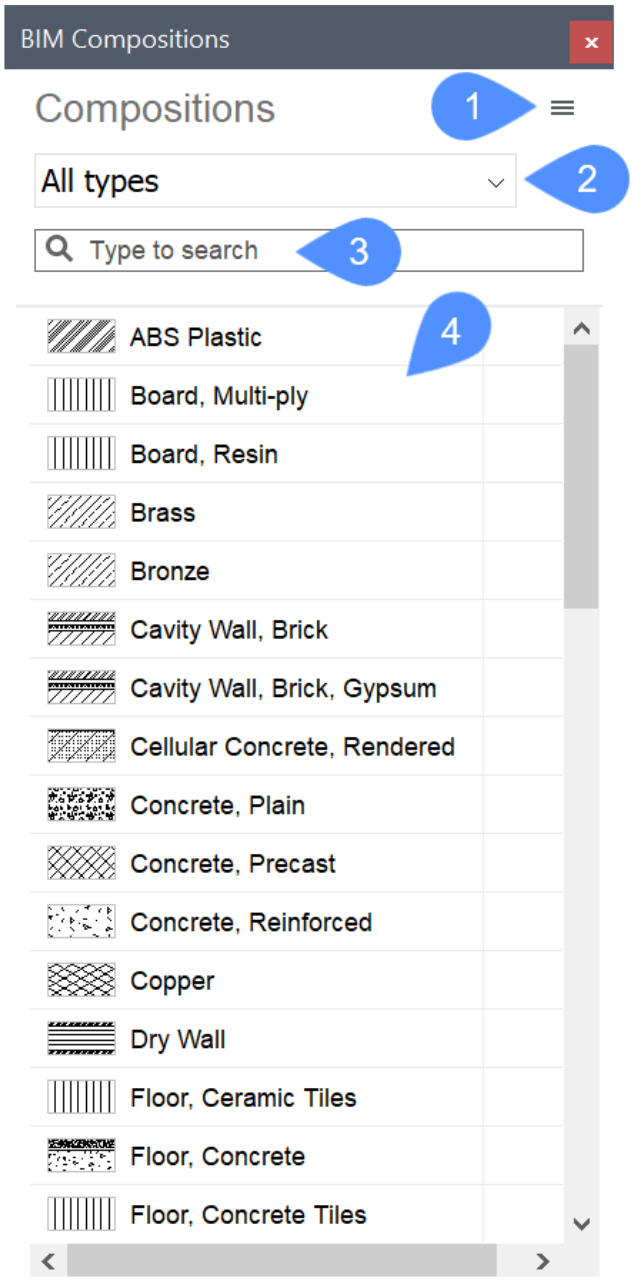
20.4.5.2 About

BIM compositions:

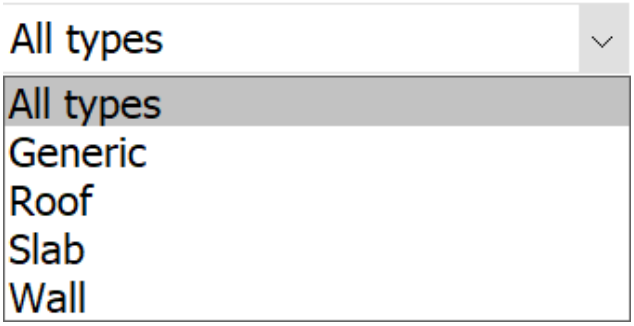
- Are stored in the library database.
- Are either multiply or single-ply.
- Ply materials are managed in the **Physical Materials** editor ( ).
- Can be assigned to any solid in a BIM model.
- Are listed in the **BIM Compositions** panel ( ).
- Are managed in the **Compositions** editor ( ).

For more information about Attaching Compositions, see the procedure article Attaching Compositions.

20.4.5.3 Using the BIM compositions panel



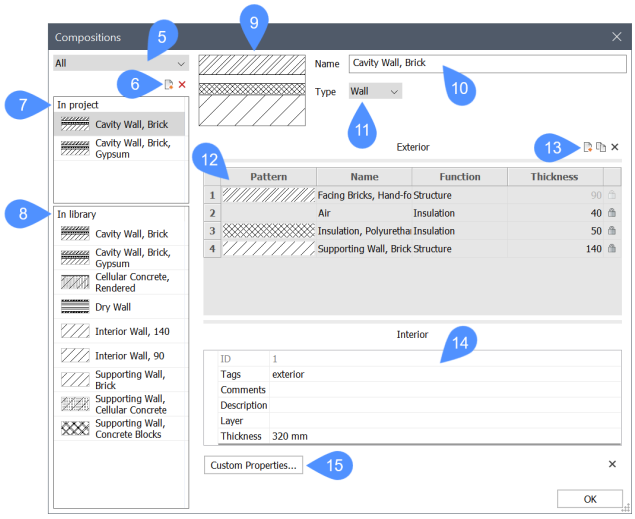
(1) Menu	Open material dialog... Open compositions dialog... Open project dialog... Show only compositions in project...

(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.

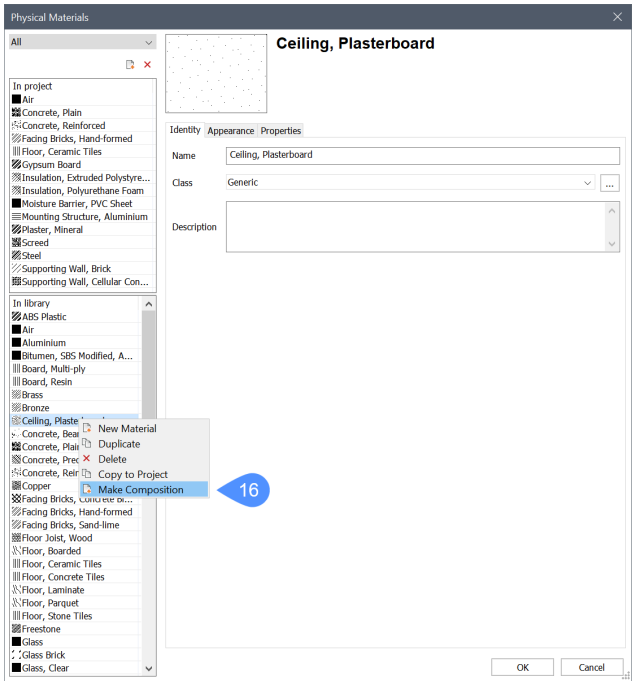
2 Type a new name in the **Name** field for the composition.

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.

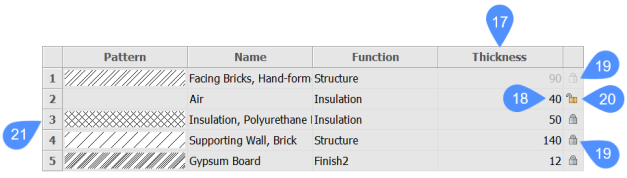
Step 2: Changing the plies

- **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.

Uwaga: 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.

  Uwaga: 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.

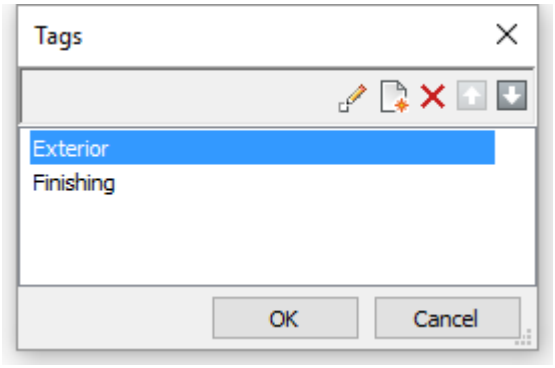
  The thickness of a ply can only be unlocked if the **Allow Custom Thickness** property of the material is set to **Yes**.

- **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: 

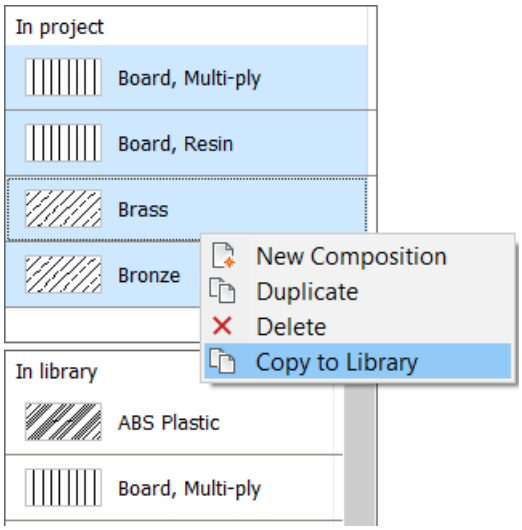
When LEVELOFDETAIL=0, the major faces of a solid are rendered using the material of the outside plies. The minor faces (for example top, bottom, start and end face of a wall) are rendered using the material of the exterior ply.

20.4.5.8 Copying compositions

You can use the following procedure in both the **BIM Compositions** panel and the **Compositions** editor:

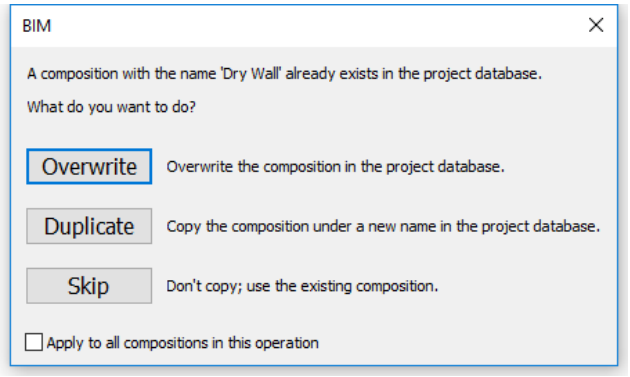
1 Select the composition(s) in the source database. Press and hold the Ctrl key to select multiple compositions.

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).

Uwaga: 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

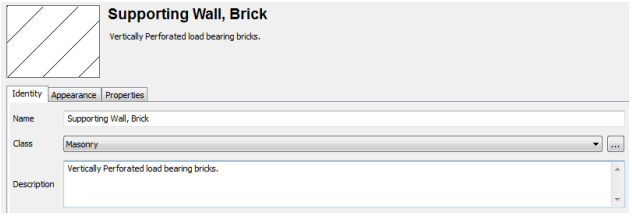
The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic** Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical 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:



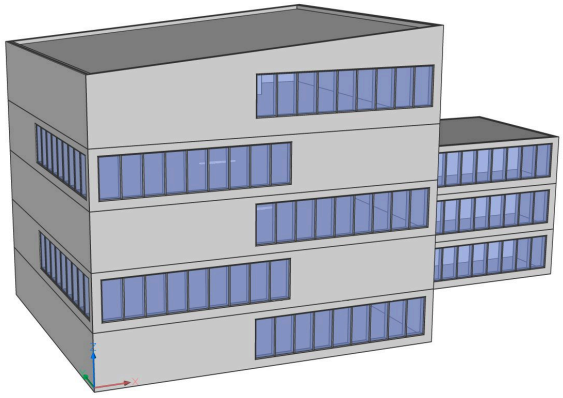
For more information about this setting visit the article LEVELOFDETAIL.

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.

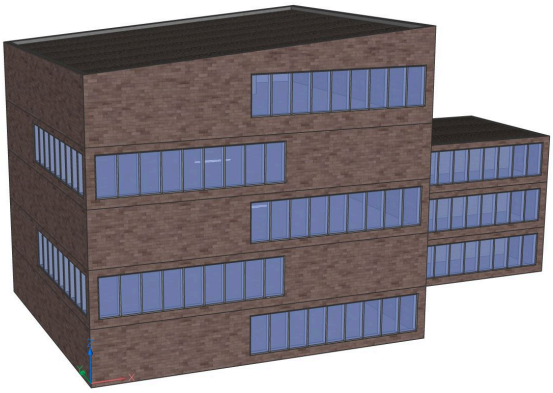
To toggle the value of the RENDERCOMPOSITIONMATERIAL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:



For more information about this command, visit the article RENDERCOMPOSITIONMATERIAL.



RENDERCOMPOSITIONMATERIAL = 0 (off)



RENDERCOMPOSITIONMATERIAL = 1 (on)

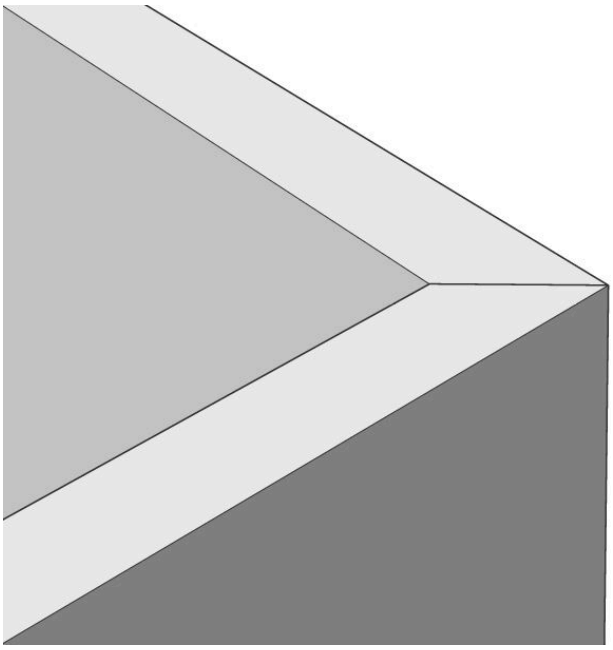
Uwaga: Only when Visual Styles have their **Face Settings / Material Display settings** set to **Materials and Textures** (e.g. **BIM**, **Modeling**, and **Realistic**), a visual representation of materials is displayed. Composition plies are displayed in the model as well as render materials.

Uwaga: 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.

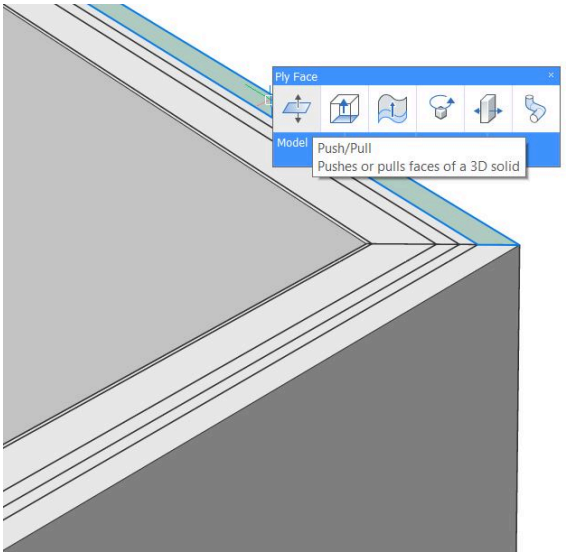
- **LOD =0, RENDERCOMPOSITIONMATERIAL = 0**

  The material definition of the solid is used and no materials are displayed.



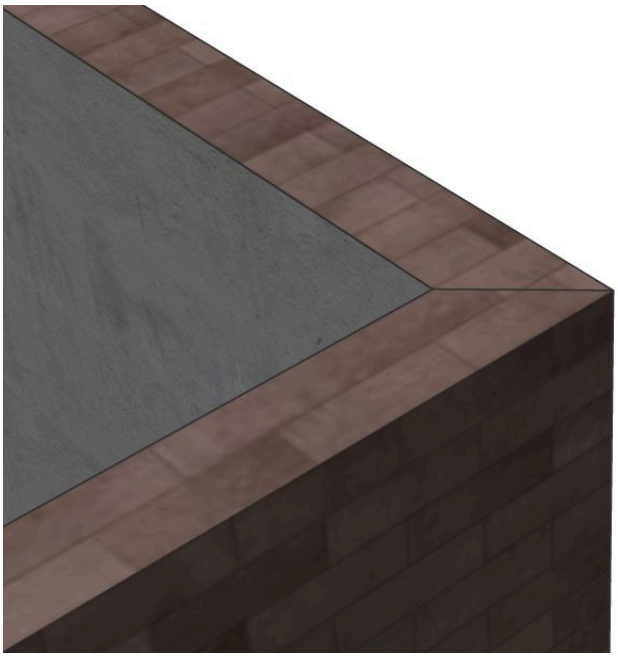
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 0**

  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.



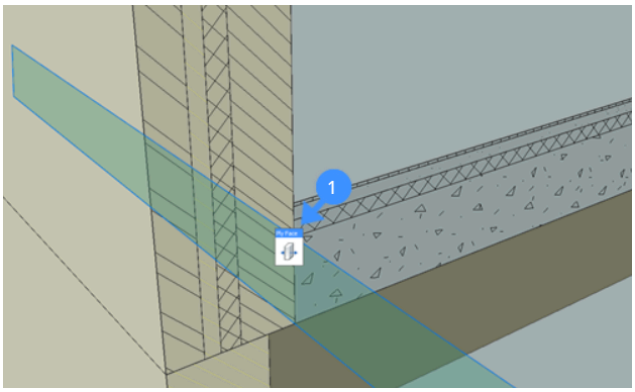
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 1**

  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

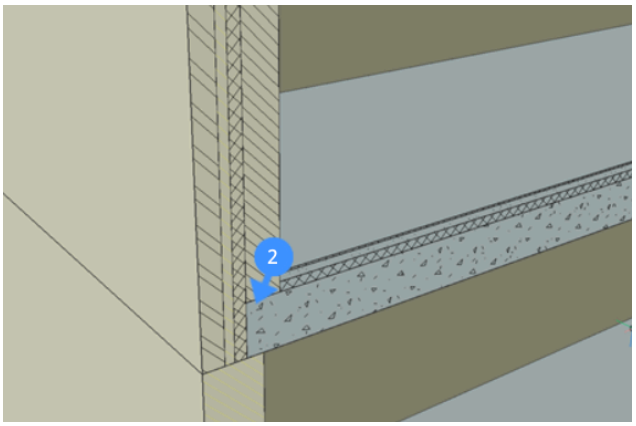
1 Make sure the value of **LEVELOFDETAIL = 2 (High)**.

2 Place your cursor near the composition ply you want to edit and press the Tab key until the **Ply face** (1) highlights.



Uwaga: 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).



Uwaga: Make sure DMPUSHPULLSUBTRACT = 1. This ensures that plies being push/pulled are subtracted from any of the other 3D solids they intersect.

Uwaga: 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.

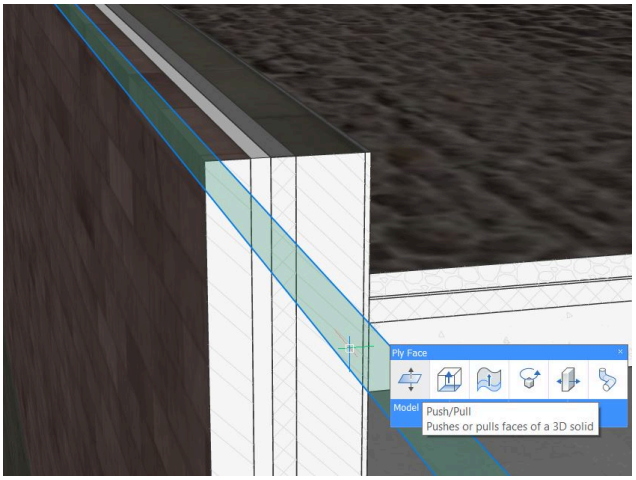


2 Click the **Face detection** icon in the ribbon to enable face detection.

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.

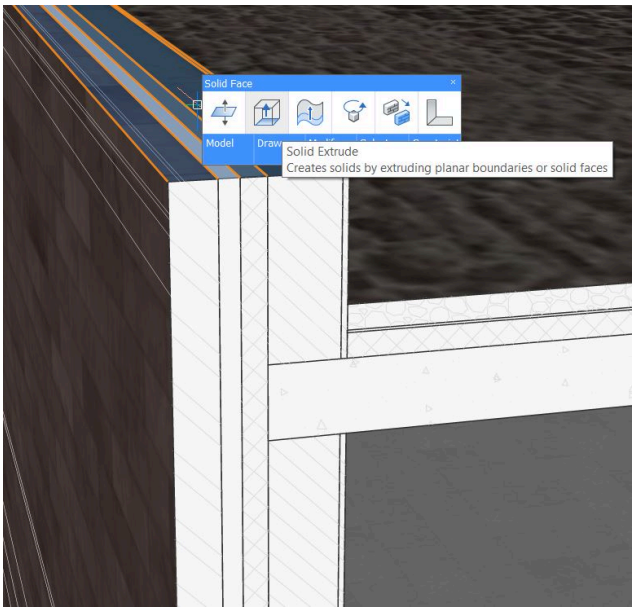
5 Choose **Push/Pull** in the **model** tab of the Quad.



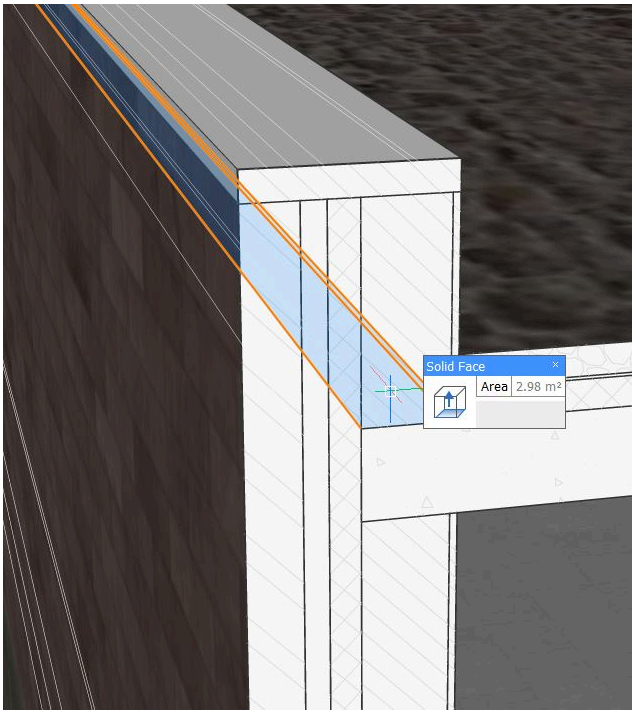
6 Push/Pull the face to the next ply of the wall.



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

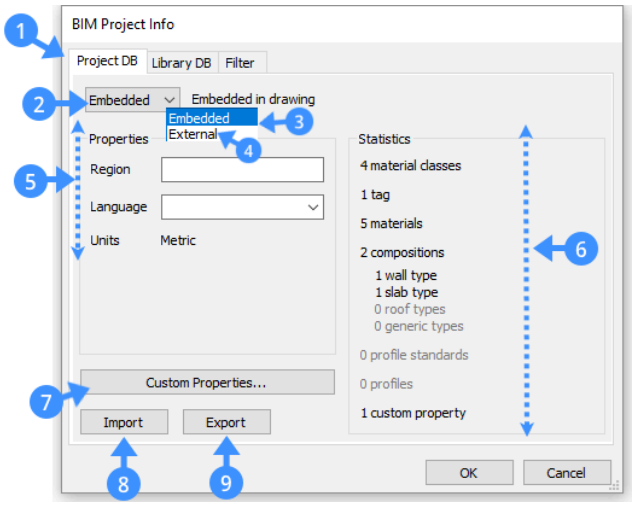
The **BIM Project Info** allows you to control the location, properties, and content of the Project and Library databases. When you launch the BIMPROJECTINFO command in BricsCAD, the BIM project info displays in a dialog box that includes certain components and tabs. Project and library databases of 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.

Uwaga: 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.

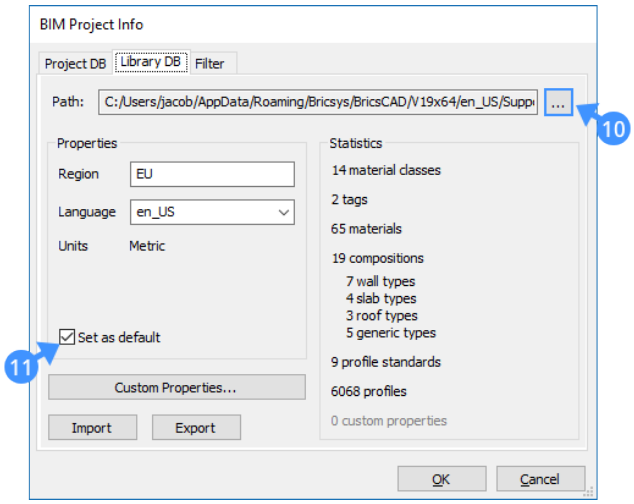
3 To specify the Project Database properties, such as Region and Language, fill out the **Region** field and choose one of the language options in the drop-down list.

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

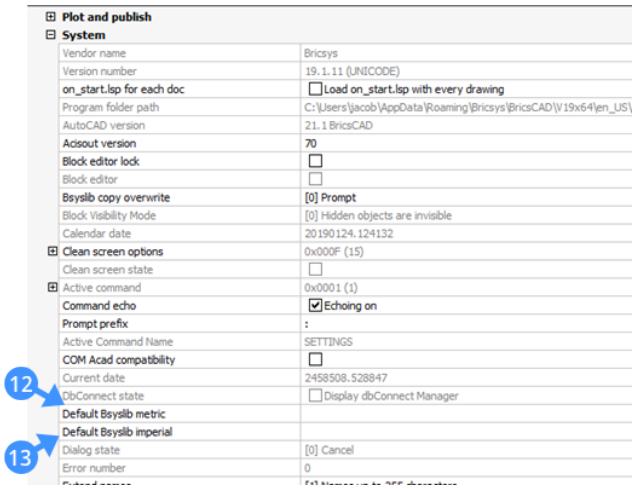
1 Click the **Library DB** tab page in the **BIM Project Info** dialog box. The following illustration shows the Library Database section in the dialog box.



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.

Uwaga: You can edit the default library settings (12, 13) in the **Settings** dialog under **Program options** > **System**.



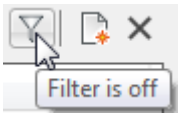
20.4.8.5 The project filter

To each Building Material or Composition, any number of tags can be attached. Any string can be entered as a tag.

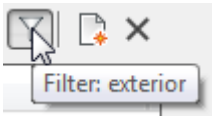
Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed.

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.

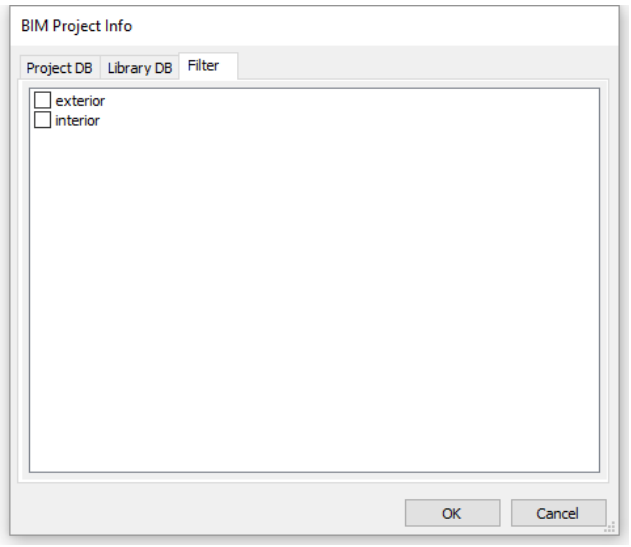


A tooltip shows the current project filter.

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.

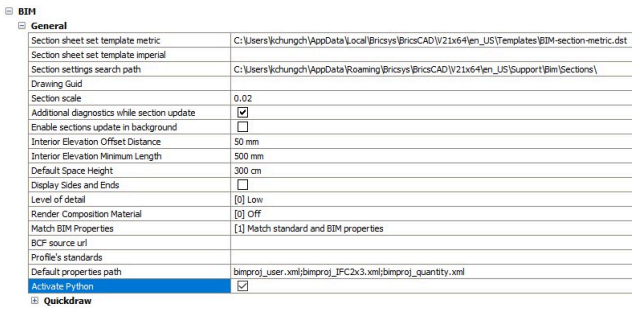
However, If the value shows **<0 for OFF>**, type **1**in the Command line and press Enter to confirm.

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)
```

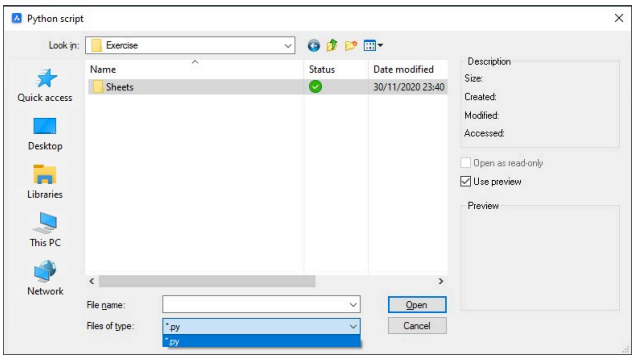
For more information on the BricsCAD API and its various classes, please visit the API chapter below.

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.



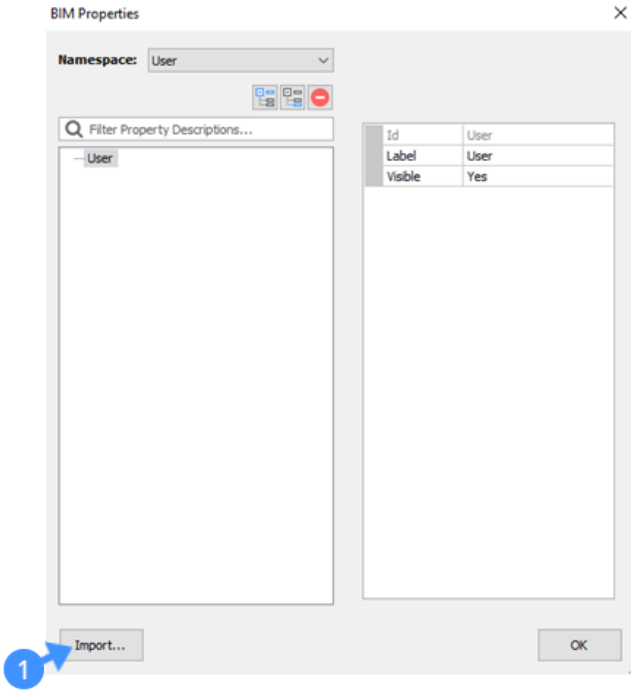
For more information about this command, visit the Command Reference article BIMPROPERTIES.

20.4.10.2 Assigning classification codes to the BIM

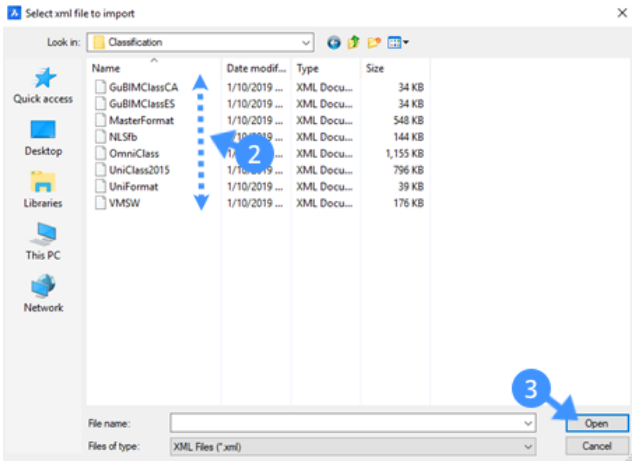
1 Open the BIM Properties dialog box from the Quad (**BIM** > **Model**).

Uwaga: 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.

Uwaga: 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.

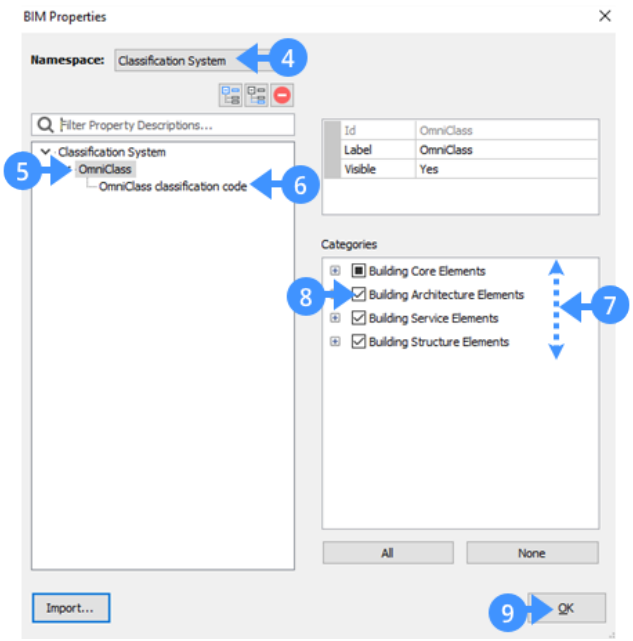
By default, the classification system is applied to all sorts of building elements (7).

Optional: Deselect the categories of building elements to which the classification system is applied.

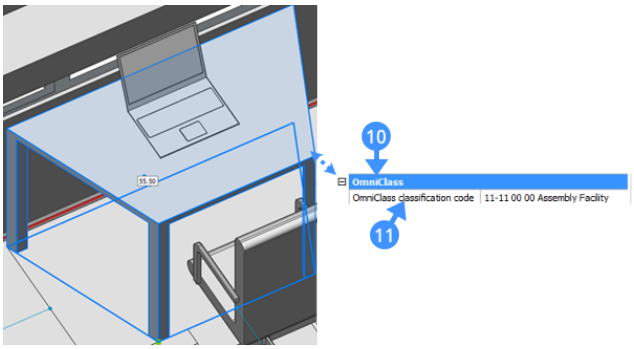
To do so, check the box next to the category name (8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only.

Click the **OK** button (9) to finalize the procedure.

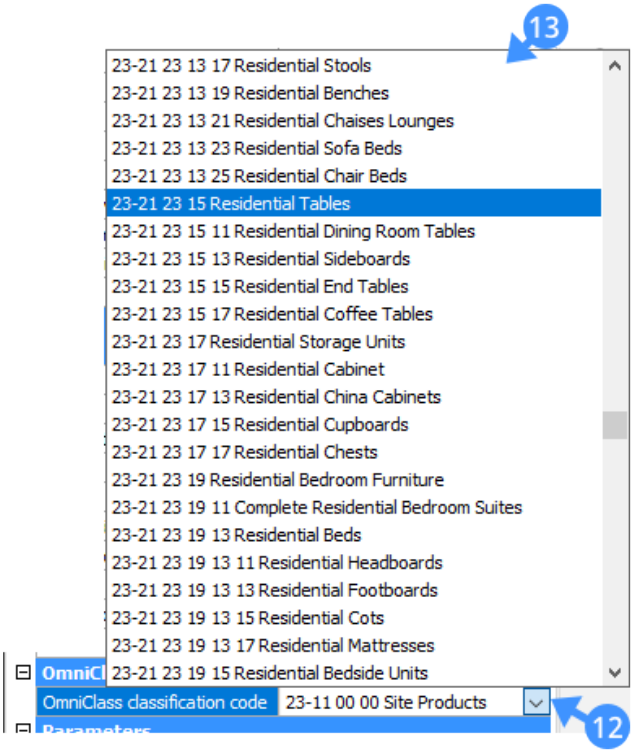
Orimport another classification system following the previous steps.



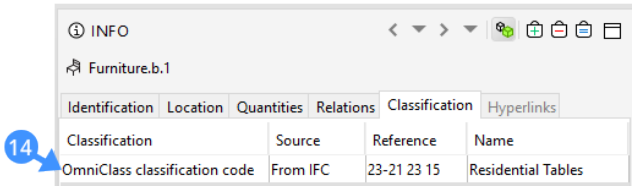
5 Select the element to which you want to assign a classification code, and open the properties panel. The classification system (10) and code (11) should be available in the properties panel.



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).



Uwaga: 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.

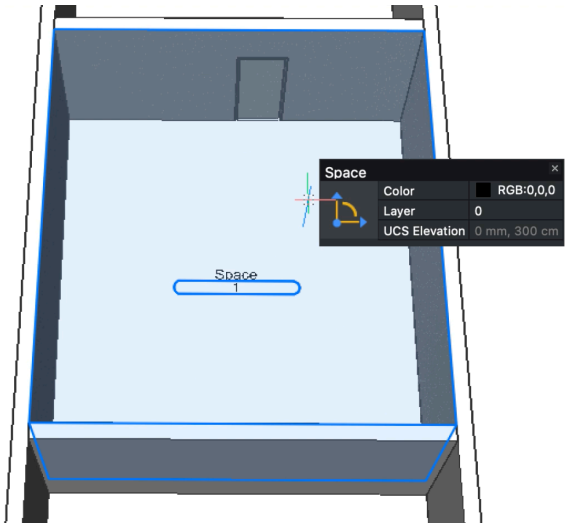
You are prompted:*Pick a point inside the space or [Edit];*

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.



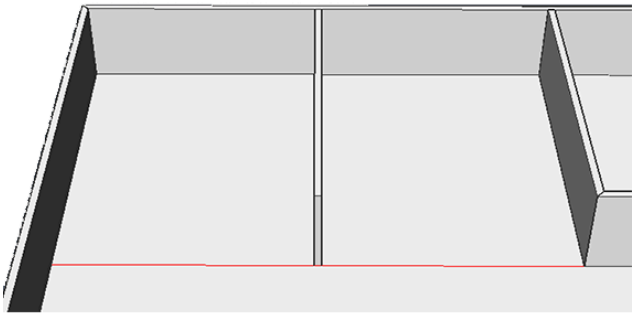
6 Edit the properties of the BIM Space in the properties panel.

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.

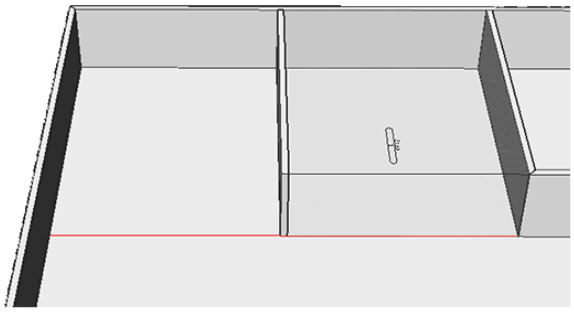


2 Select the dividing elements (the red line).

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.



Uwaga: 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.

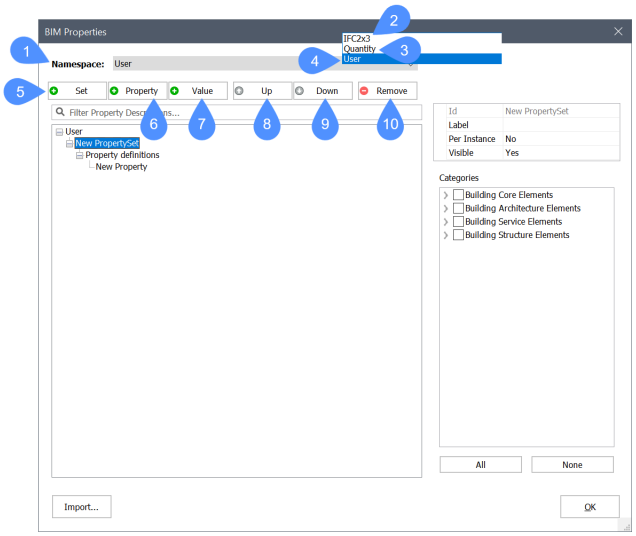
For more information about this command, visit the Command Reference article BIMPROPERTIES.

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 ***Uwaga:*** 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:

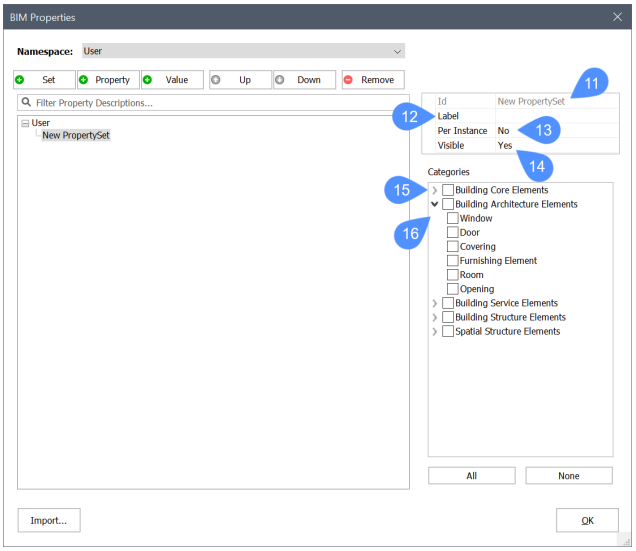
- Type a name in the **Id** (11) field.
- Type a name in the **Label** (12) field.

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.

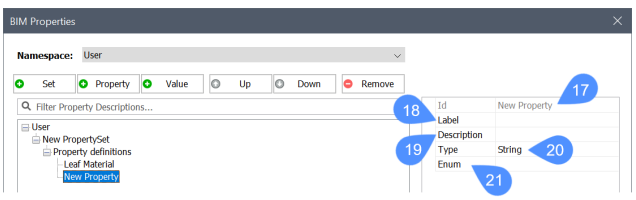


3 To add a property to the currently selected property set, first, select a property set, then click the Add Property button to add a new property.

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.



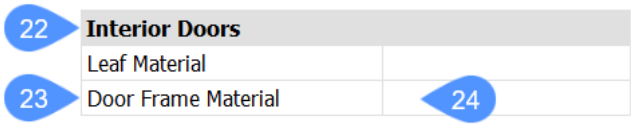
5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button to remove.

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

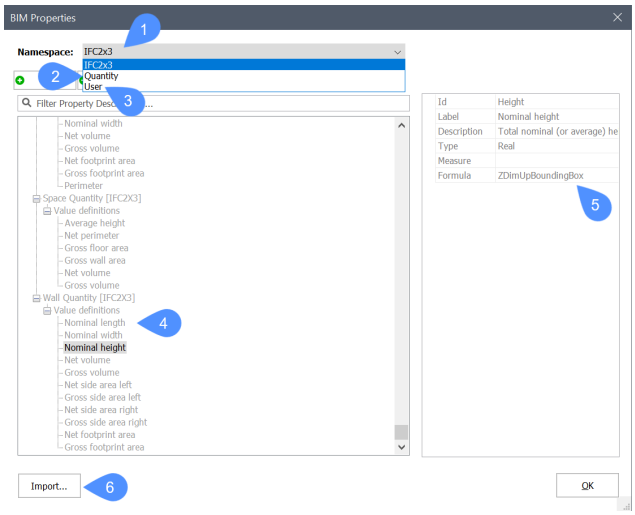
In the BIM Properties dialog, by default three namespaces are available: **IFC2x3** (1), **Quantity** (2) and **User** (3). The **IFC2x3** namespace contains properties that are defined by the IFC2x3 schema. This contains both common properties (e.g. Wall Common properties, such as **Fire Rating** and **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.

> ***Uwaga:*** 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.



Uwaga: 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

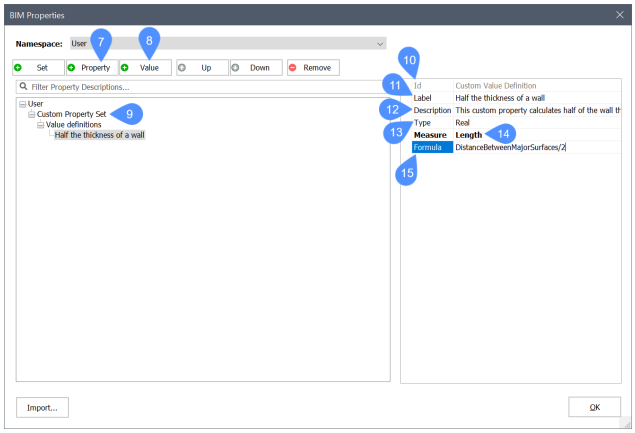
In the **User** namespace it is possible to create custom properties inside a custom property set using the **Property** (7) button, see Procedure article Custom Properties.

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**?

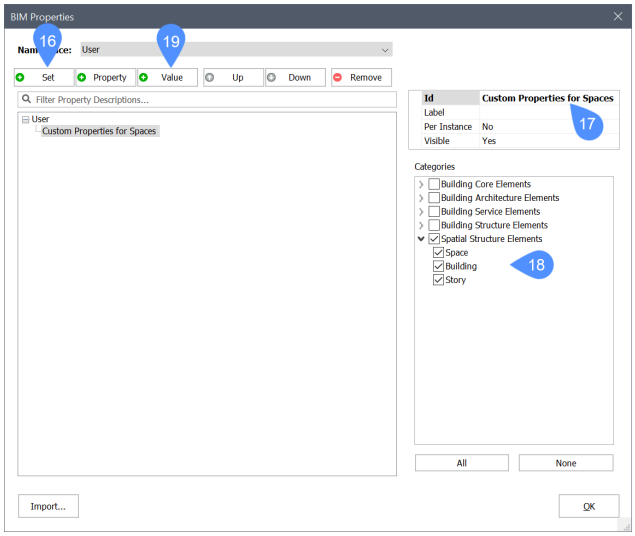
- **Formula** (15): the actual expression of the quantity. Here Core quantities can be used, as well as mathematical expressions as defined in the Procedure article Working with Parameters and Constraints.

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).

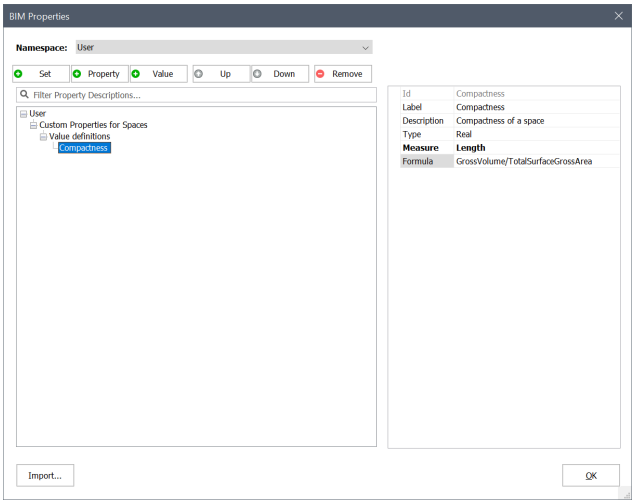
6 Give this Value Definition an **Id** and a **Label**, e.g. **Compactness**.

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)

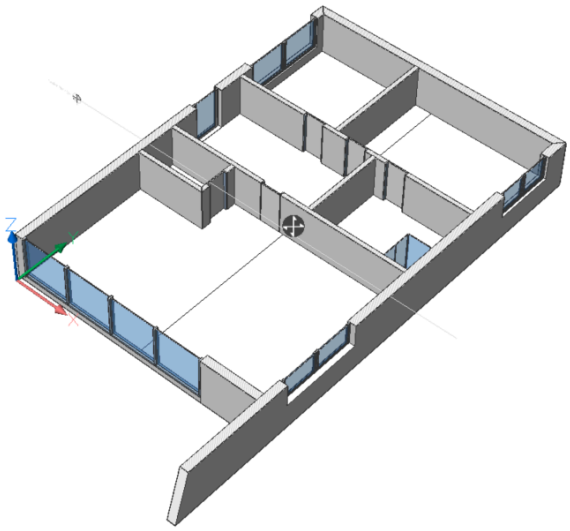
Only solids which are classified as a **BIM Slab** can be processed by the BIMSETLOADBEARINGDIRECTION command. Use the BIMIFY or BIMCLASSIFY command to classify a solid as a **BIM Slab**.

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

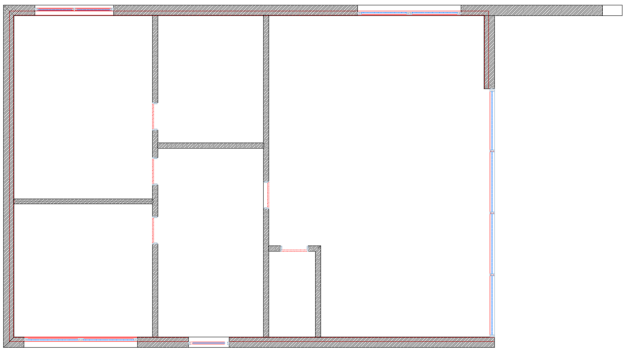
1 When you insert the load bearing tag in a plan section, you need to select a line, which is generated by the slab. Since slabs usually overlap with walls, you need to set the **Hidden Lines** > **Show** property of the **BIM Section** entity to **Yes**.

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.

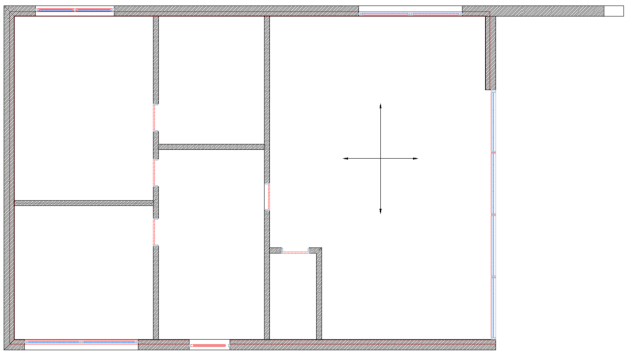
5 Choose the **Change Tag Type** option.

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

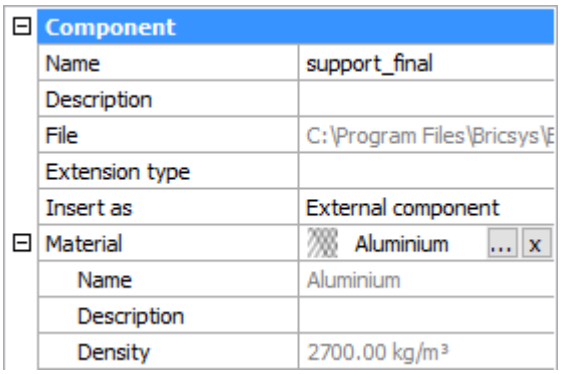
20.4.15.1 Commands

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.

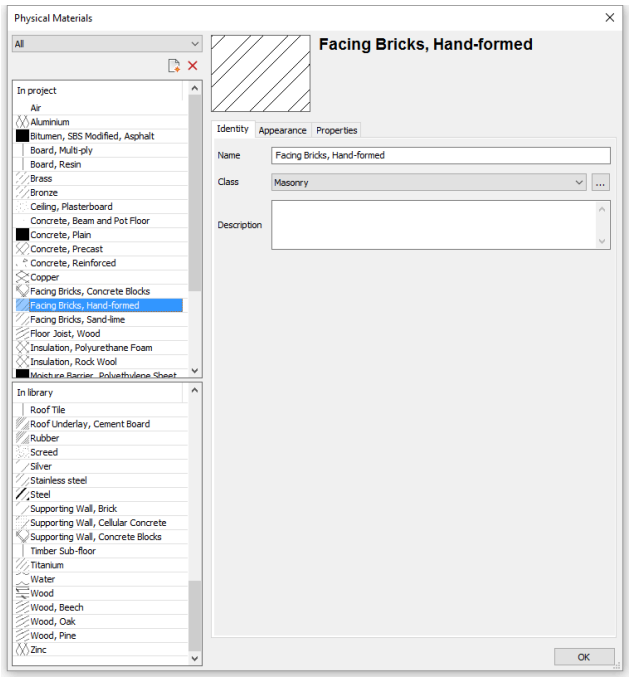
20.4.15.4 Editing a physical material

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

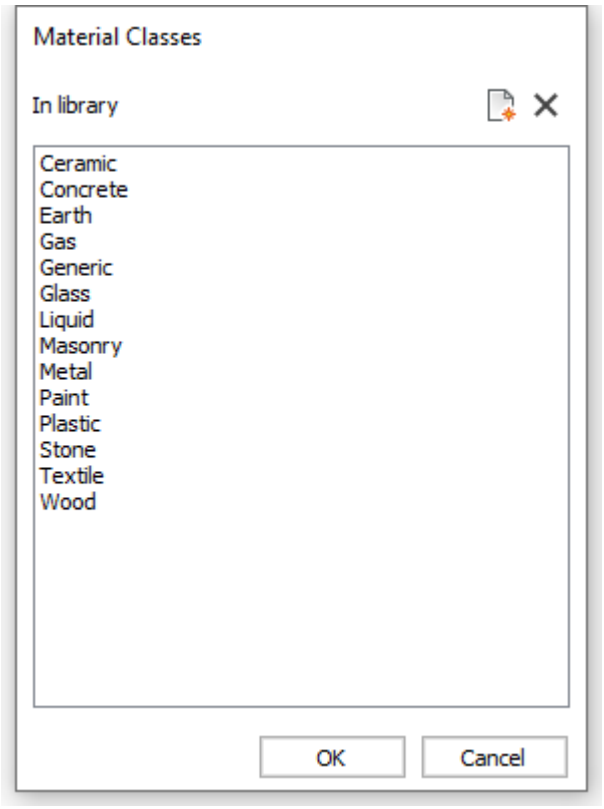


1 Type a **Name** for the material.

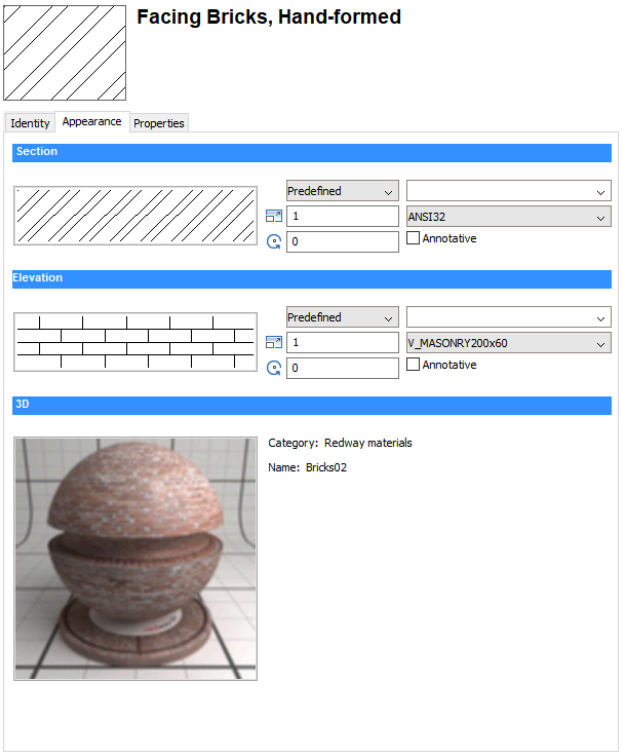
2 Select a **Class** from the drop-down list*.

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.

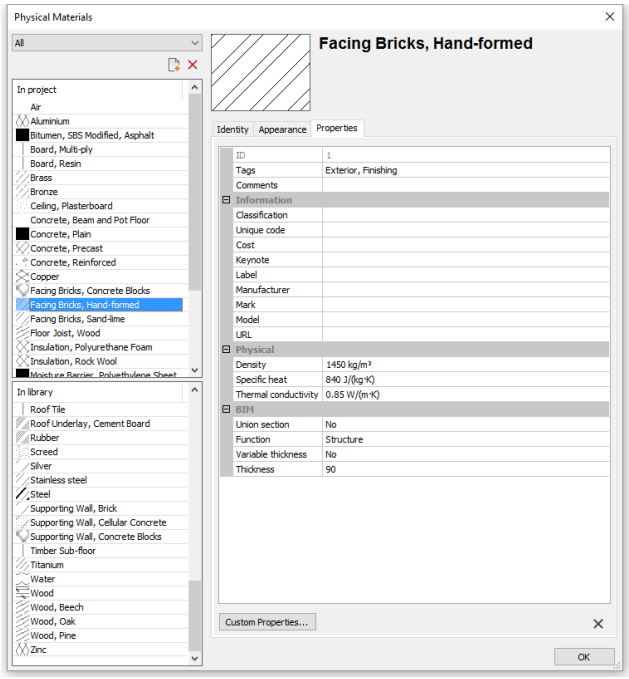
  See **Section** settings above, except for the **Scale** setting. The elevation hatch is independent of the **Scale** property of the **BIM** section.

- **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.

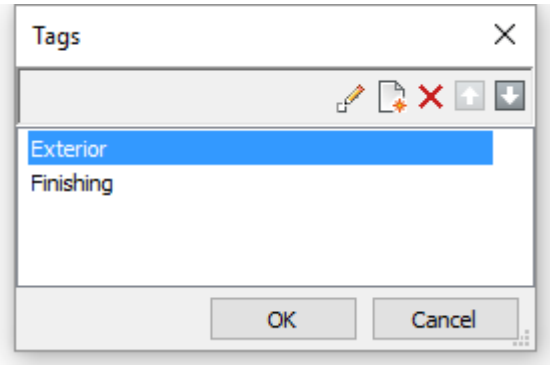
Properties tab page



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.

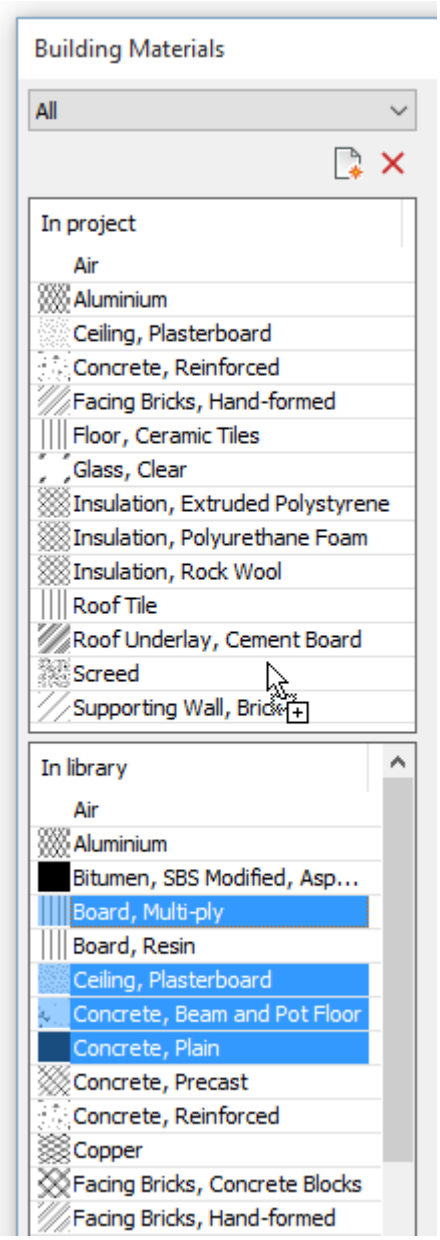
- **Union Section**: Select **Yes** to remove connection lines in sections between different solids of this material. Select **No** to keep the connection lines.
- **Union Elevation**: Select **Yes** to remove connection lines in elevations between different solids of this material. Select **No** to 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.

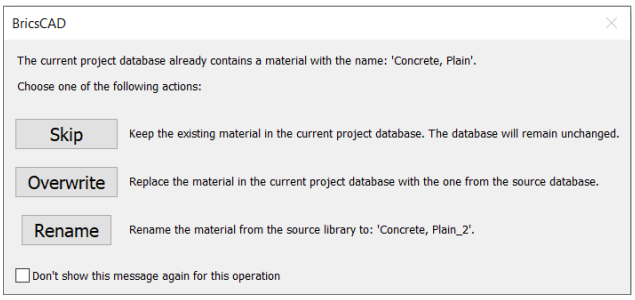
1 Press and hold the Ctrl-key to select multiple materials.



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**.

Uwaga: You cannot delete materials that are used in a composition or a mechanical component.

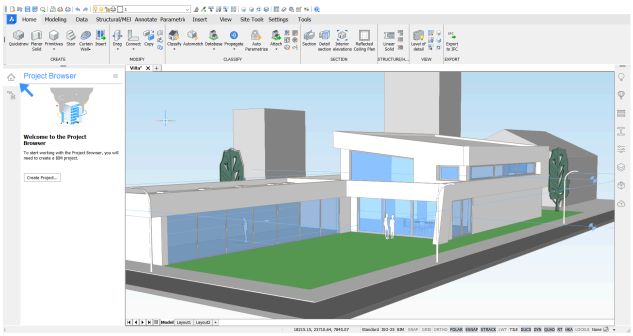
20.4.16 Project browser

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.



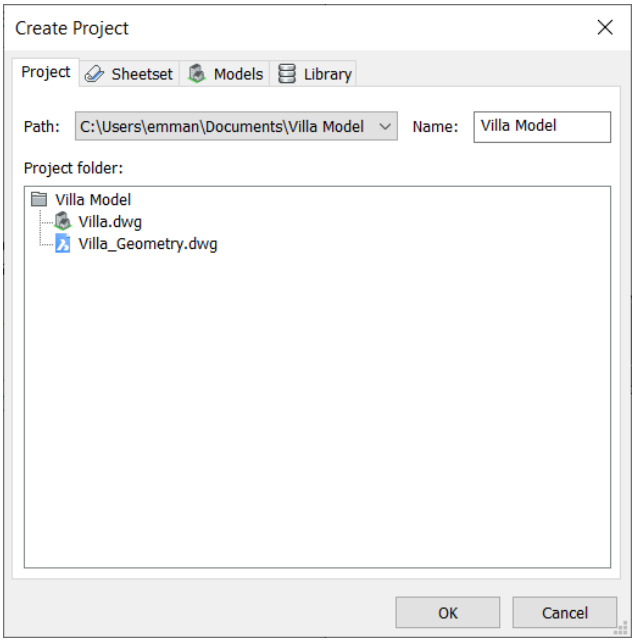
Uwaga: If the **Project Browser** doesn't display in the panel, right-click in the panel and select **Panels** > **BIM Project Browser**

20.4.16.3 Create a BIM project

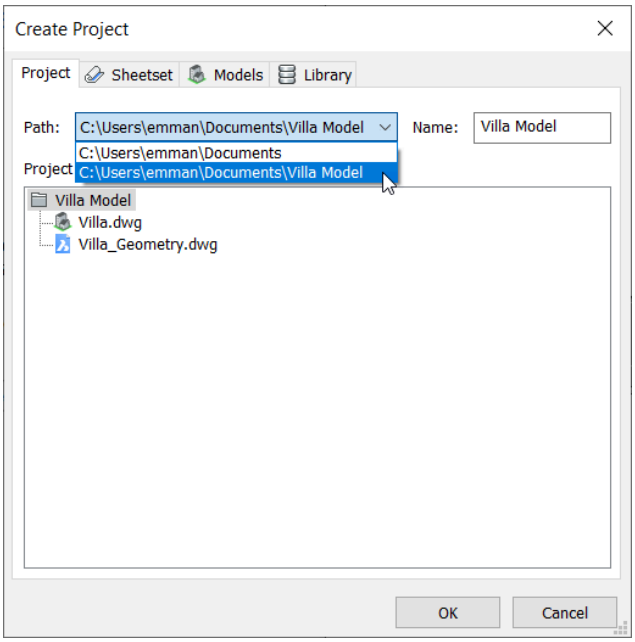
To create a BIM project, you must select one or more model drawings.

Project

1 Click the **Create Project...** button in the **Project Browser**. A dialog box appears.



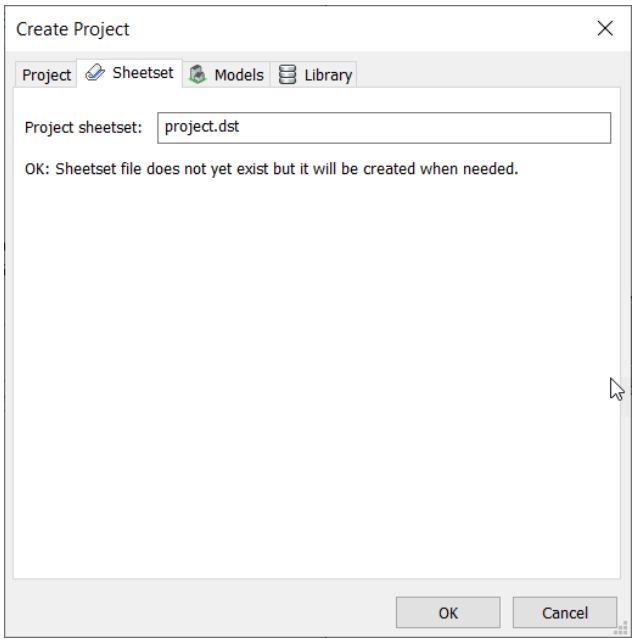
2 **(Optional)** Click the **Path** list button and select a parent folder.



3 **(Optional)** Type a name in the **Name** field (the default name is the name of the parental folder).

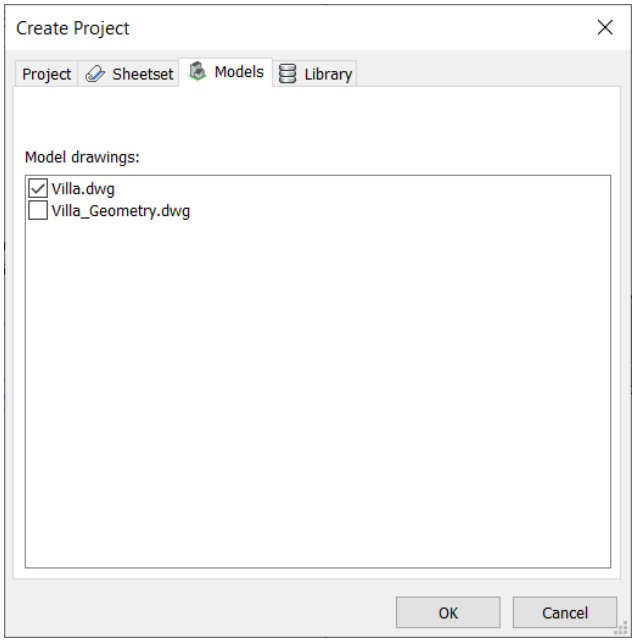
Sheetset

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.



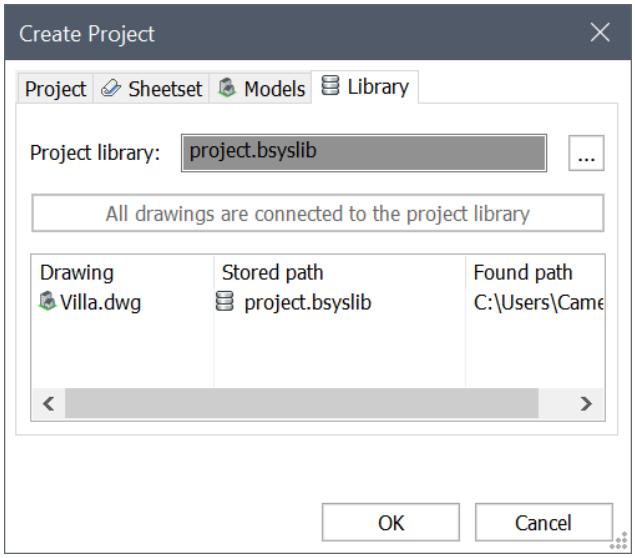
Library

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:

Uwaga: 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.

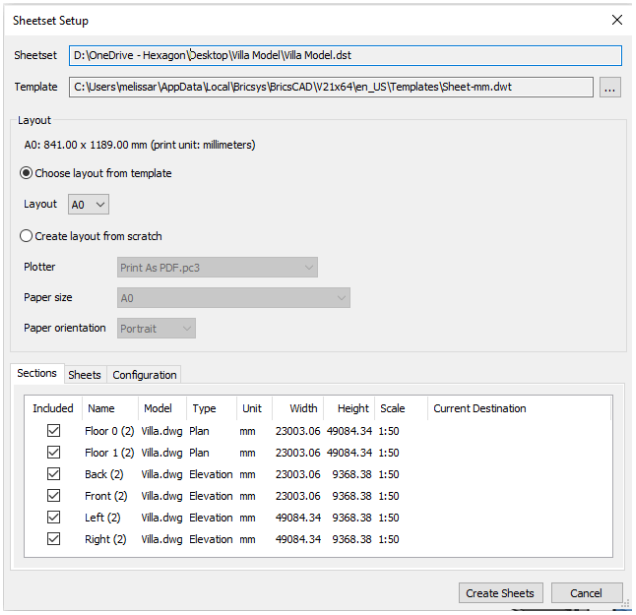
20.4.16.4 Sheetset setup

Sheetset: Shows the path of the sheetset file.

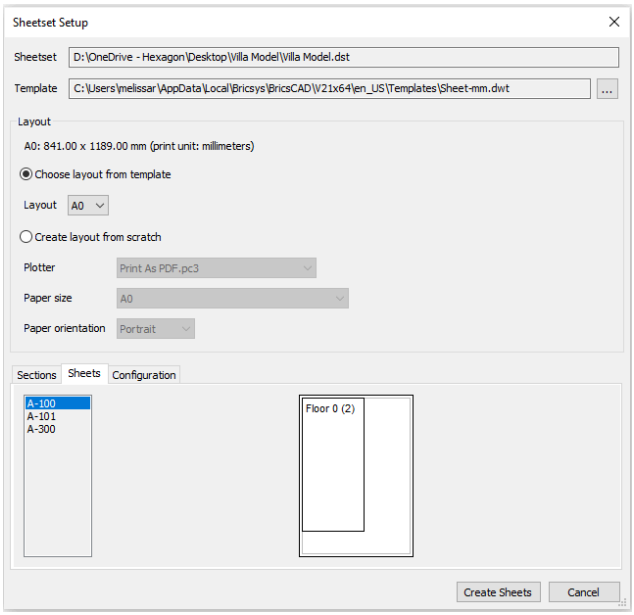
Template: Shows the path of the current template. Click **Browse** (...) to change the template.

Layout: Selects which layout (different paper sizes) the section results will be generated.

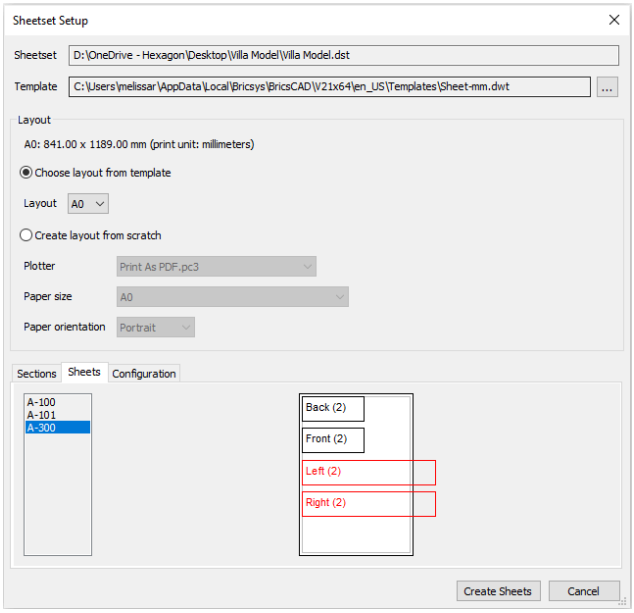
1 The **Sections** tab lists all sections. For each section with an empty **Current Destination** field, a viewport and sheet view are generated.



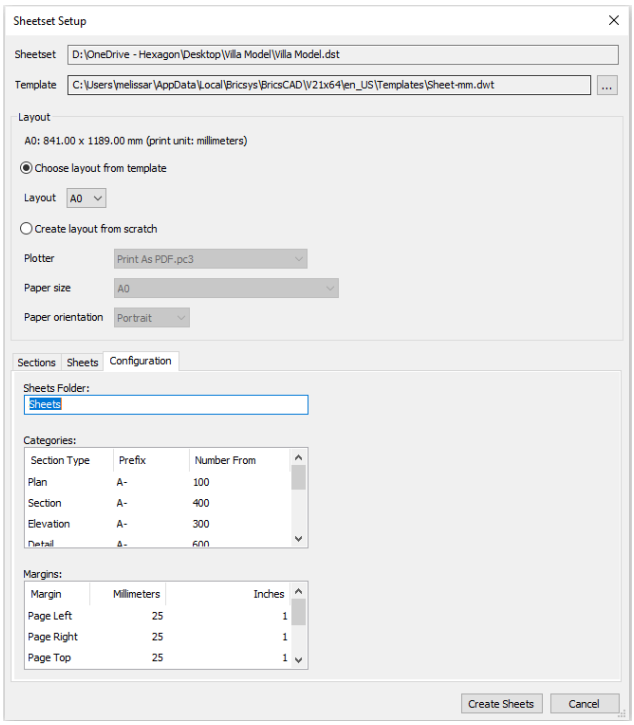
2 Click **Sheets** to preview each sheet (within the selected layout).



Uwaga: When the viewport is larger than the layout, it displays in red.

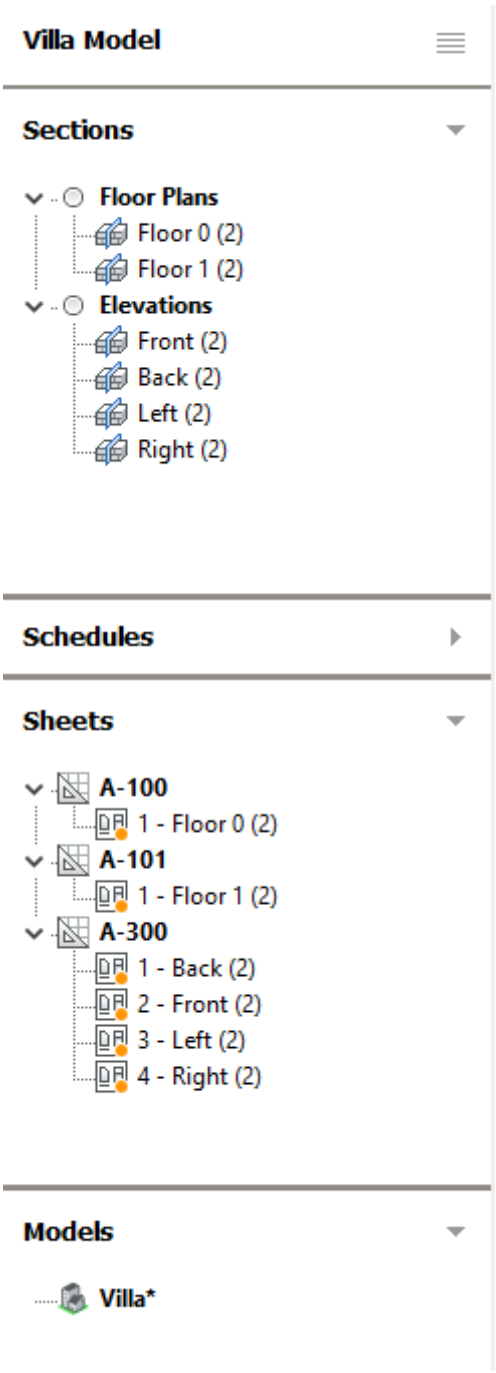


3 Click the **Configuration** tab. Slowly double-click (or click and hit Space/Enter) to change the properties. Under **Sheets Folder** you can change the name of the folder where the sheets will be stored.



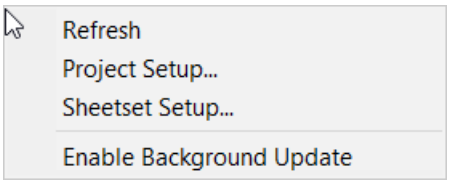
4 Click **Create Sheets**. An overview of the project displays and the sheets are generated.

Uwaga: 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.

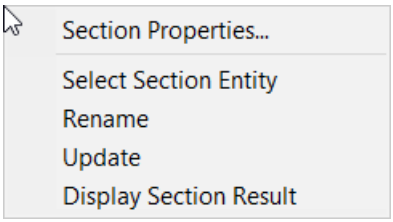
20.4.16.6 Creating schedules

See the Schedules article for more information about Schedules.

20.4.16.7 Working in the project browser

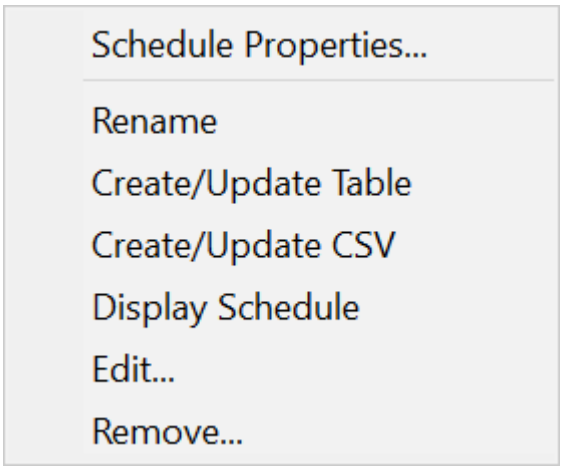
Right-click an item in the BIM **Project Browser** to display a context menu.

Sections



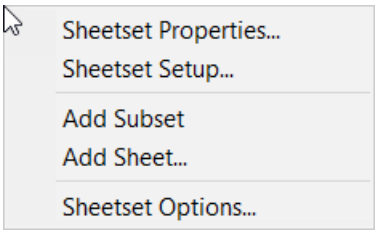
- **Section Properties...**: Opens the **Section Properties** dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: Selects the section entity in the project model drawing.
- **Rename**: Renames the section entity.
- **Update**: 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



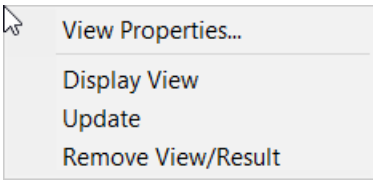
- **Schedule Properties...**: Opens the **Schedule Properties** dialog box that allows you to modify the properties of a schedule.
- **Rename**: Renames the schedule.
- **Create/Update Table**: Creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: Creates a CSV file with the 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.

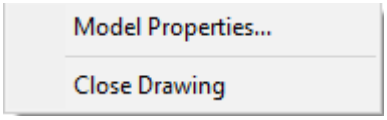
Sheet views



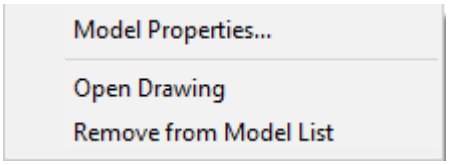
- **View Properties...**: Opens the **Sheet View Properties** dialog box that allows you to modify the properties of a sheet view.
- **Display View**: Displays the corresponding result and zooms in on its viewport.
- **Update**: Updates the sheet view. It executes the BIMSECTIONUPDATE command for the 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.

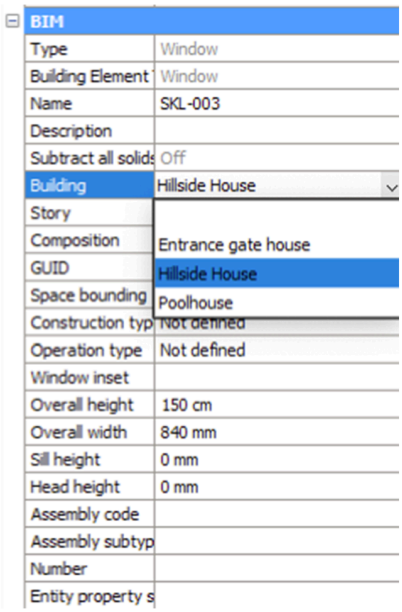
7 **(Optional)** Click **Import Spatial Locations** button ( ) to import a spatial location from a .txt file.

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.

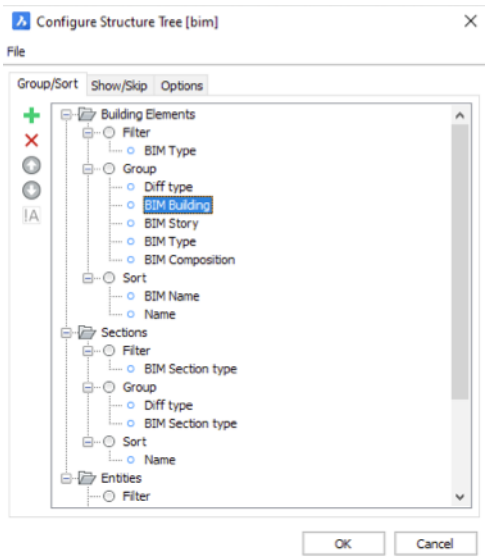
10 Do the following to assign the building and story to these building elements:

- 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.



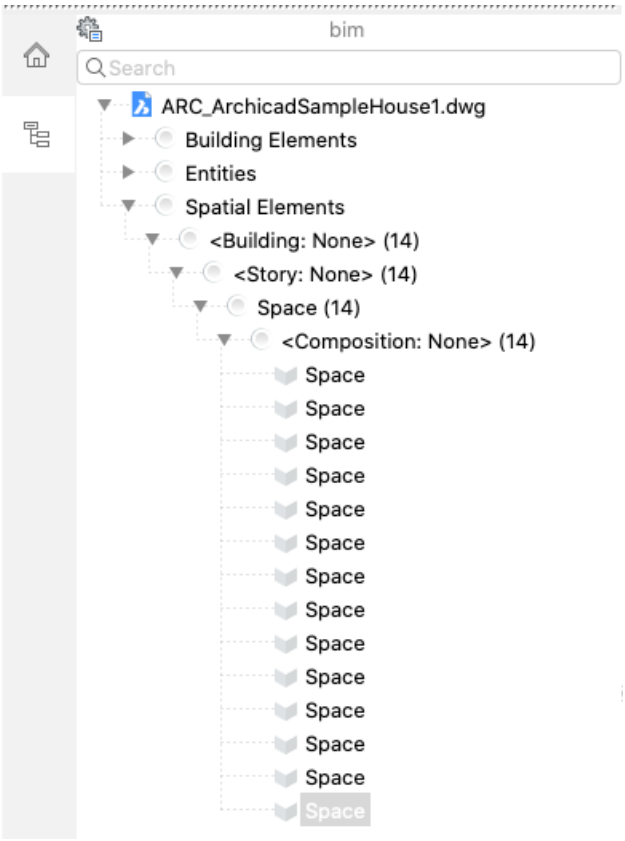
If there is no **BIM Building** or **BIM Story** in the **structure browser**, you can add them manually. See Structure Panel for more info.

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.



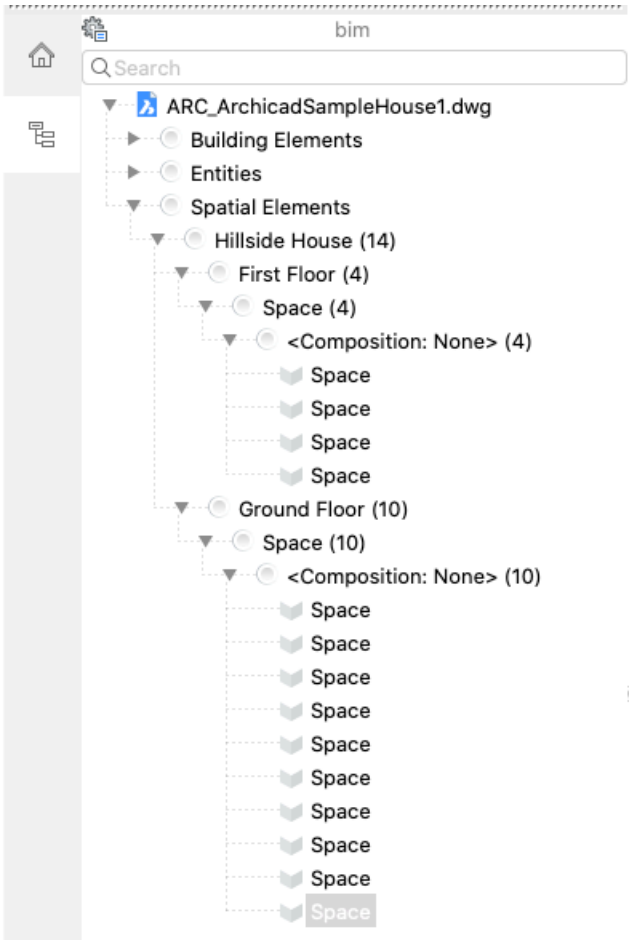
20.4.17.6 Procedure: attaching spatial locations

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).

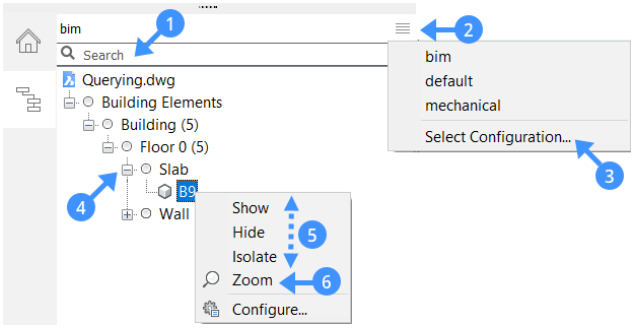
2 Click the horizontal stripes button (2) to load a configuration file in the list. Click **Select Configuration...** to select a .cst file, which specifies a configuration (3). The current configuration displays on the left. Based on the selected file the BIM model will be organized in a tree-structure.

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

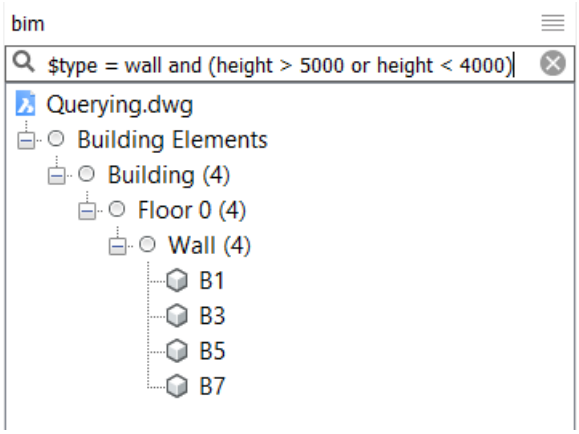
1 Typing something in the **Search** field at the top of the **structure browser** will issue a simple Text search: e.g. if you type 'wall' in the search field and press **Enter**, the **structure browser** will be filtered to everything with 'wall' in its name, including walls and curtain walls.

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).

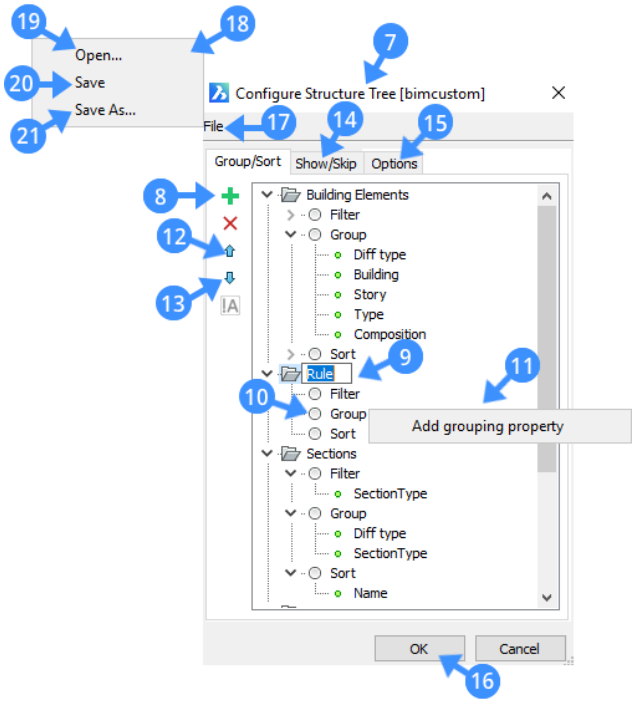
5 Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the **Structure Browser**.

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:

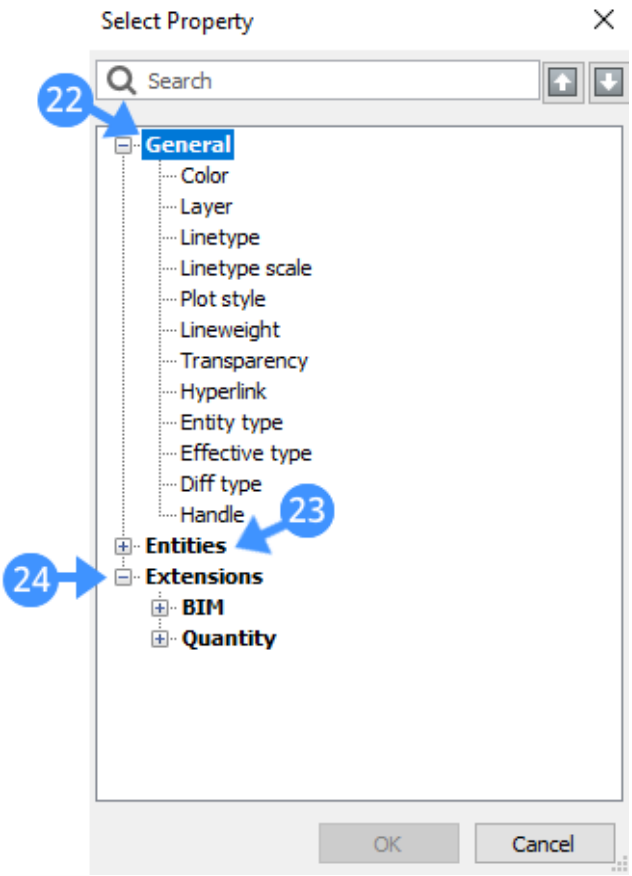
- Click the **OK** (16) button.
- Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

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.

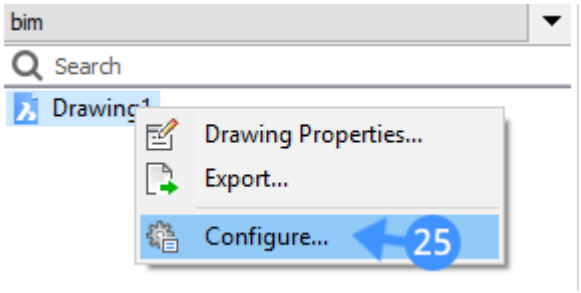
20.4.18.6 Procedure: managing the tree-structure with custom properties

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.

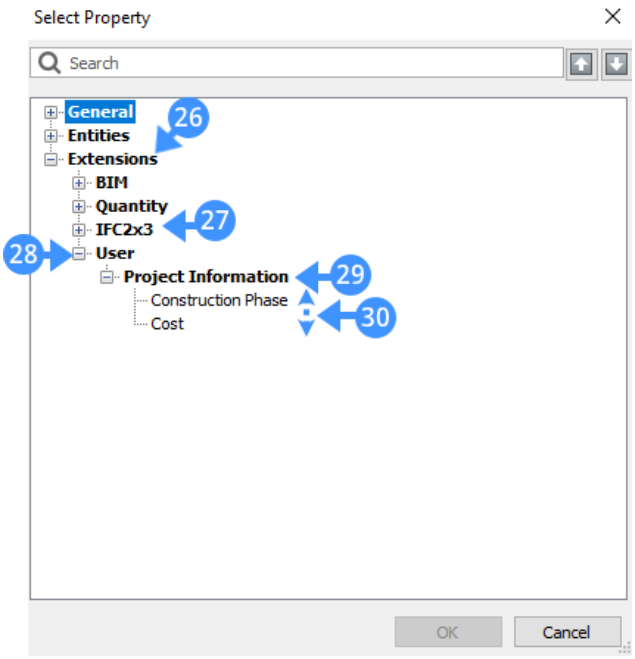
2 Select the **Group** node of the rule, and click the **Add rule or property**(+) icon.

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.

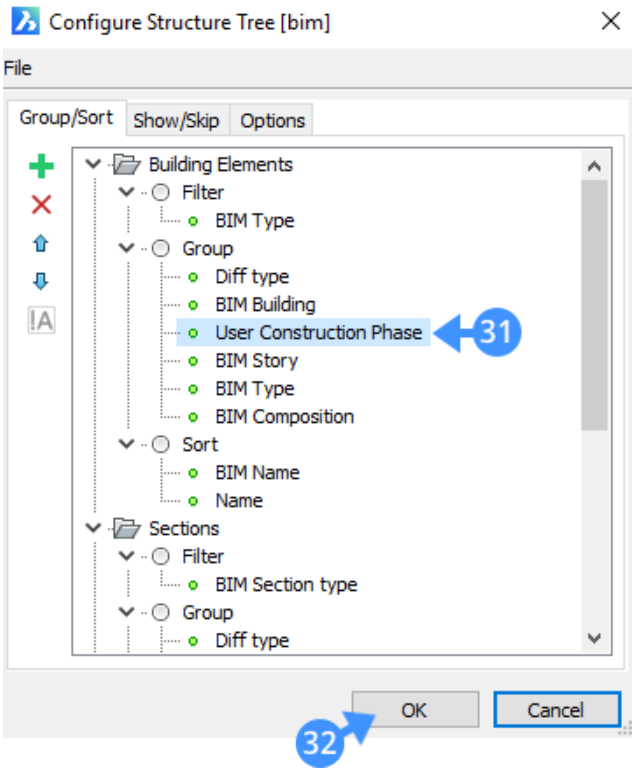
For example, the **User** namespace is expanded to display the property set (29) and properties (30) that are in this node.



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 Pomoc w projektowaniu

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.

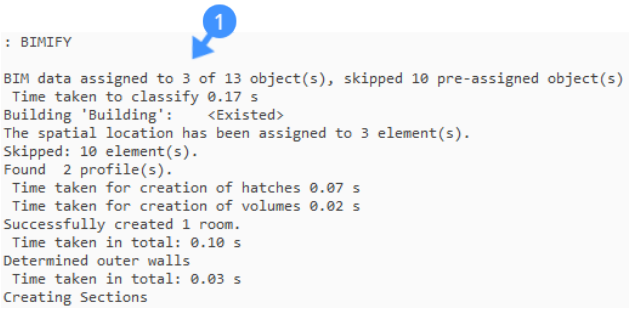


For more information about this command, visit the Command Reference article BIMIFY.

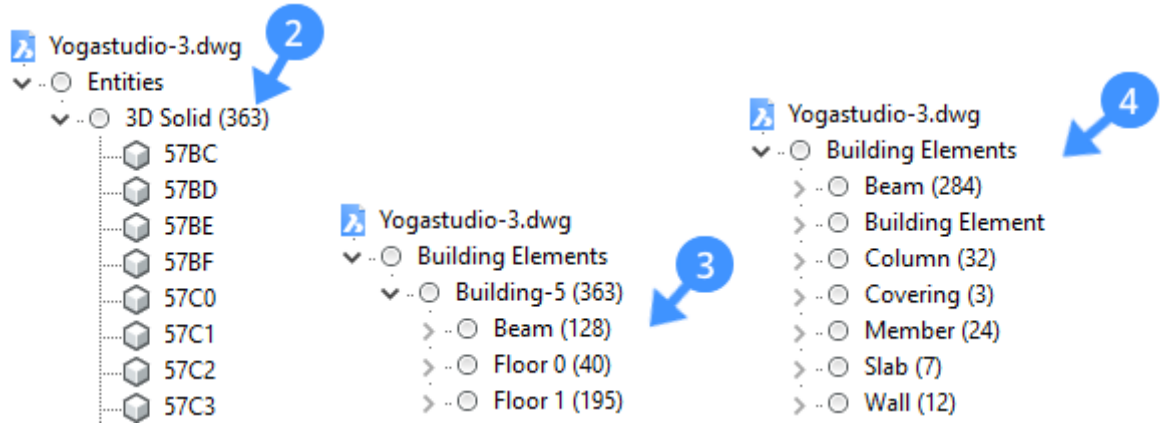
20.5.1.3 Procedure: bimify your entire drawing

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.



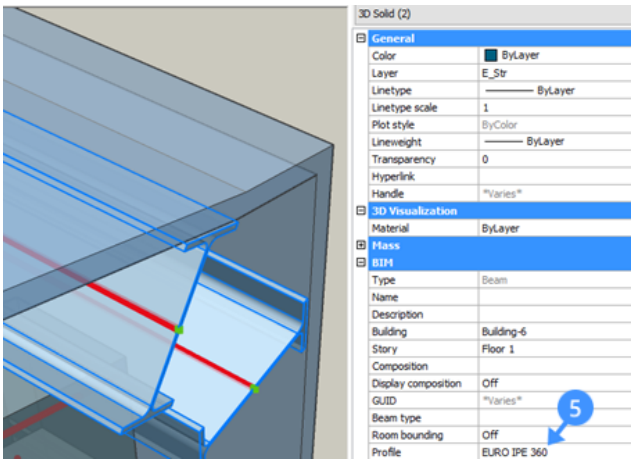
Bimify creates new spatial locations if necessary.



Left to right:

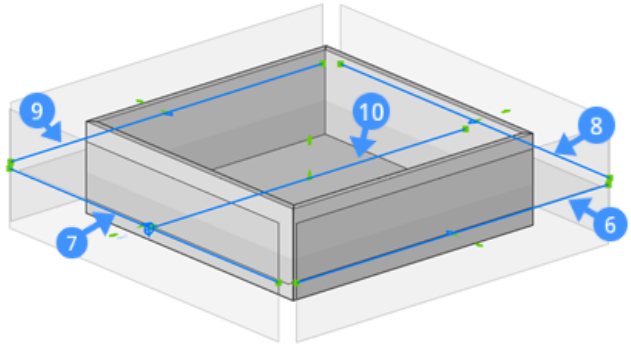
- ***Before Bimify:*** *The structure tree displayed only 3D solids (2)*
- ***After Bimify:*** *The entities are sorted into their corresponding building, story (3) and building element type (4).*

Uwaga: The **structure browser** can be customized to control the content of the tree.



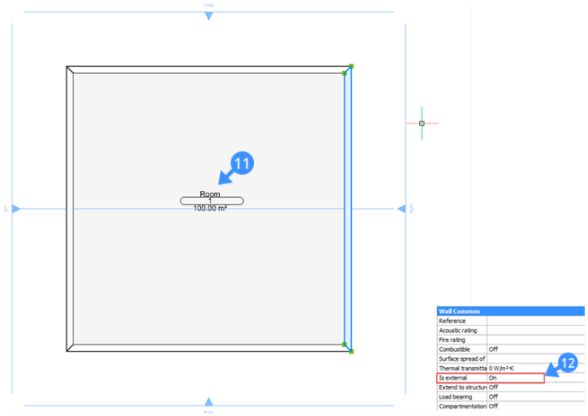
The profile data of a beam in the properties panel (5). BIMIFY recognized the profile of these model elements as structural members and detected the axis location of the profile.

The elevation views and floor plan sections are created last.



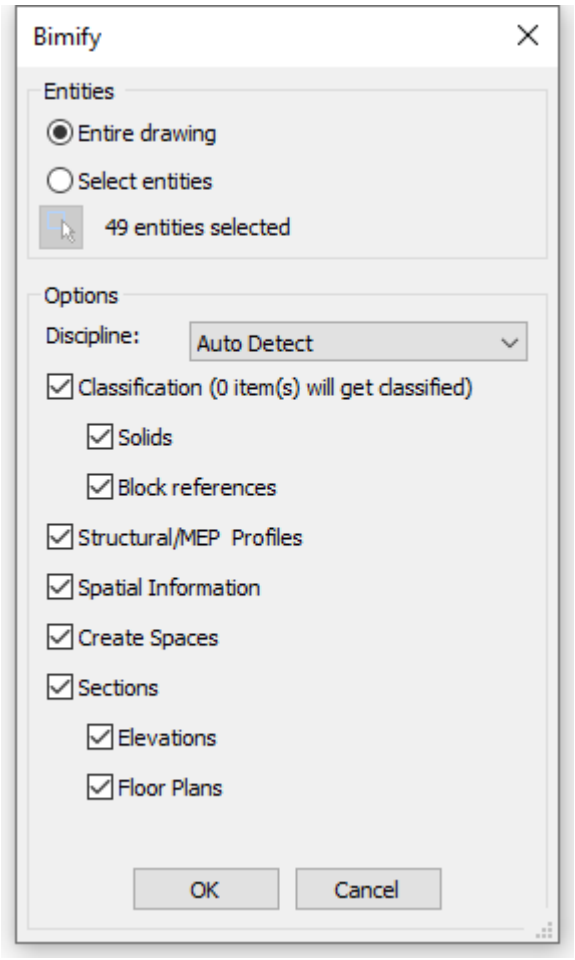
This building contains front (6), Left (7), Right (8) and Back (9) views as well as a floor plan (10).

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:



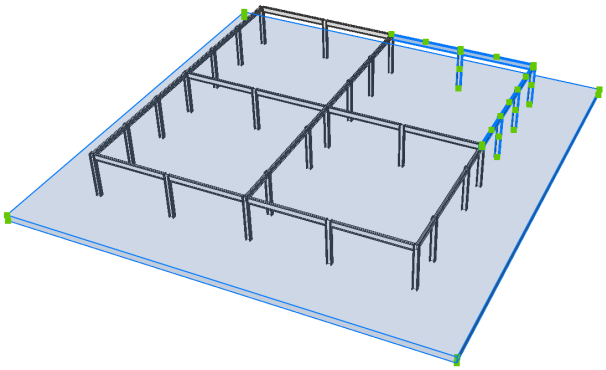
Discipline	Auto Detect Architectural Structural MEP Mixed model Defines the discipline of the drawing manually. This information helps improve the accuracy of Bimify.
Classification	Classifies **Solids** and **Block References** automatically.
Structural/MEP profiles	Assigns columns, beams, members or 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.

Uwaga: 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.

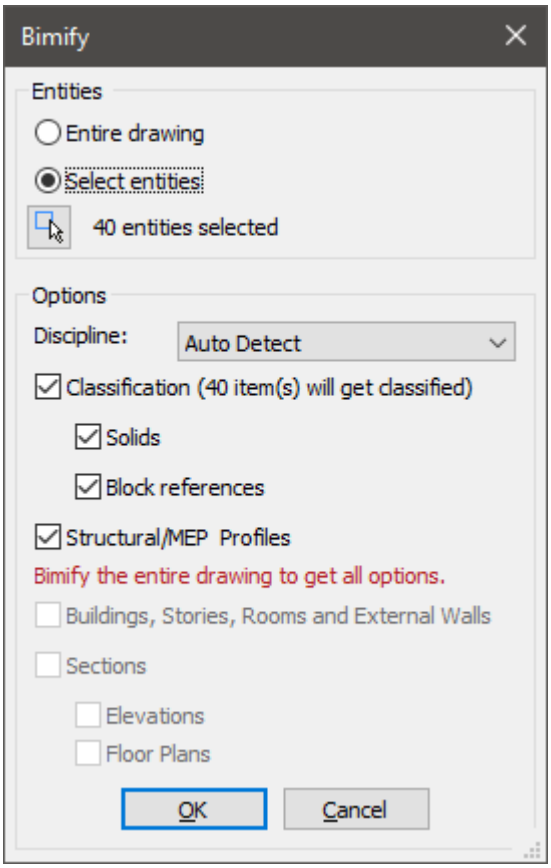
20.5.1.4 Procedure: bimify a selection of your BIM Model

1 Select the part of your drawing you want to Bimify.



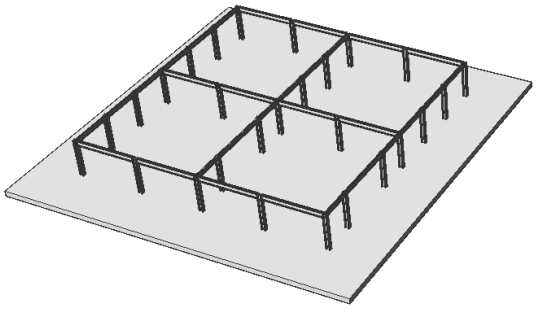
2 Type BIMIFY in the Command line and type in **A** for Advanced.

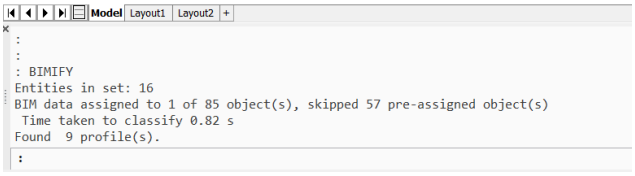
A dialog box displays:



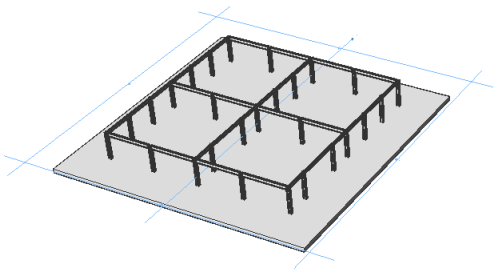
Uwaga: If you only select a part of your model, only solids, blocks and profiles can be classified. BricsCAD can only detect entities like stories, spaces, and sections when the whole drawing is selected.

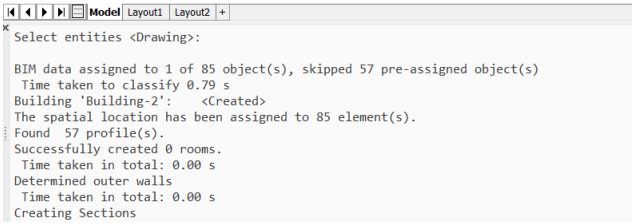
3 When the Bimify process is completed, an overview of the result is provided in the **Prompt History** panel.





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.

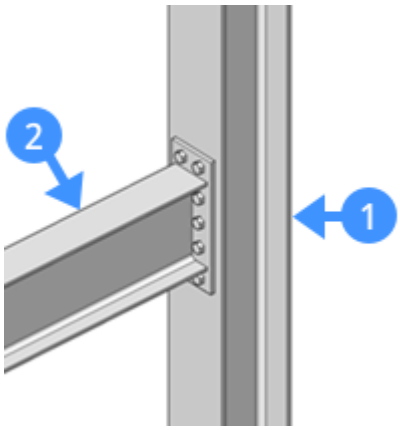
The general workflow of BIMPROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

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.



Uwaga: This is an example of a situation that can be solved by BIMPROPAGATELINEAR.

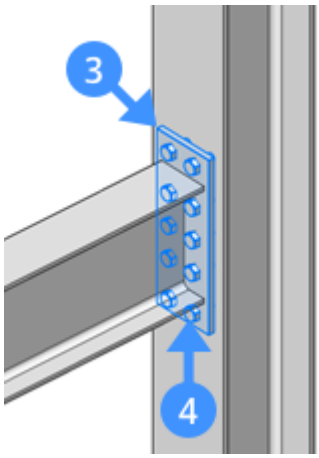
Press **Enter** to stop selecting reference solids.

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).

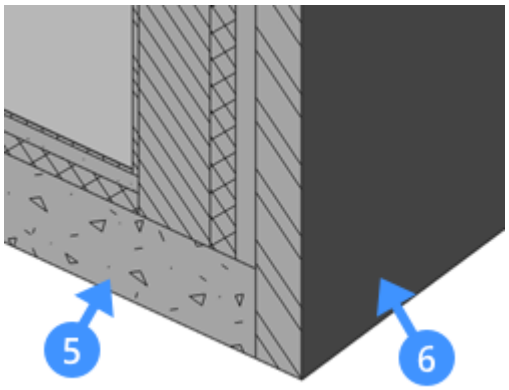
You are prompted:

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).



Uwaga: 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.



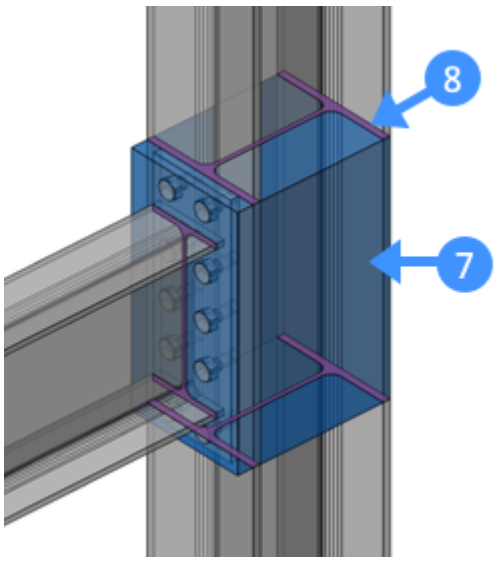
If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects.

Uwaga: 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.

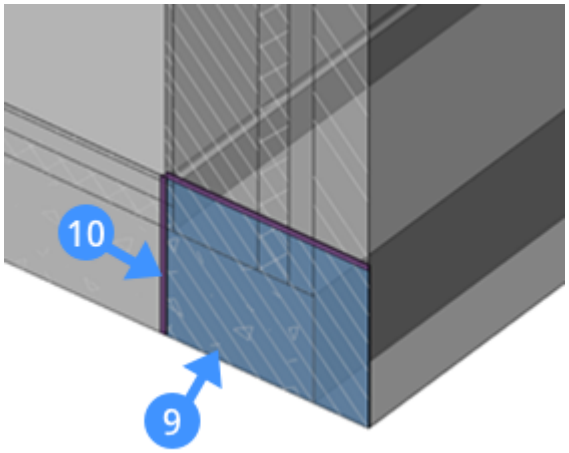
A detail volume can be either a **3D volume** or a **2D slice through a linear detail**.

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).



Uwaga: The detail of BIMPROPAGATEPLANAR and BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

You are prompted:

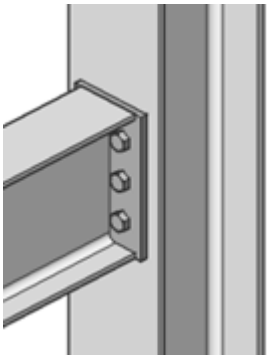
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

You are prompted: Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>:

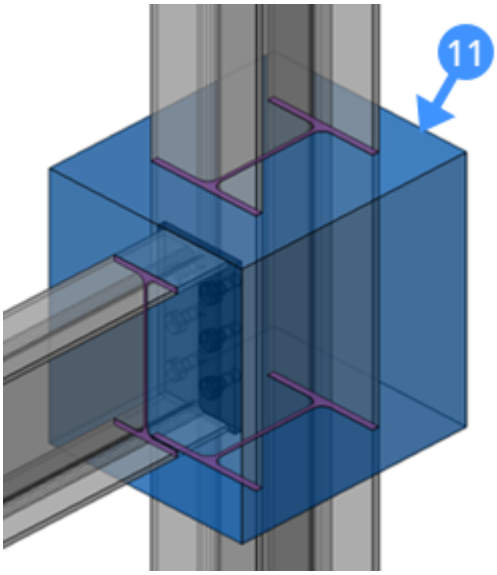
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

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:

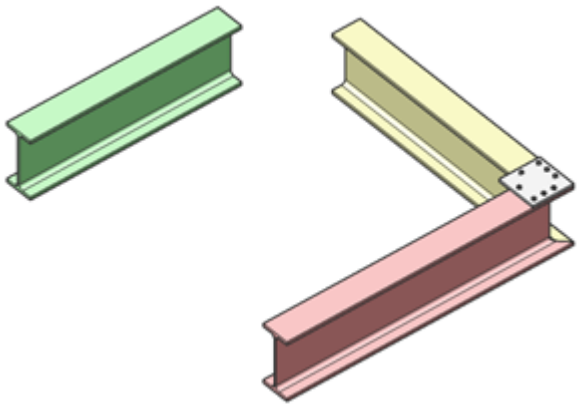
- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following:

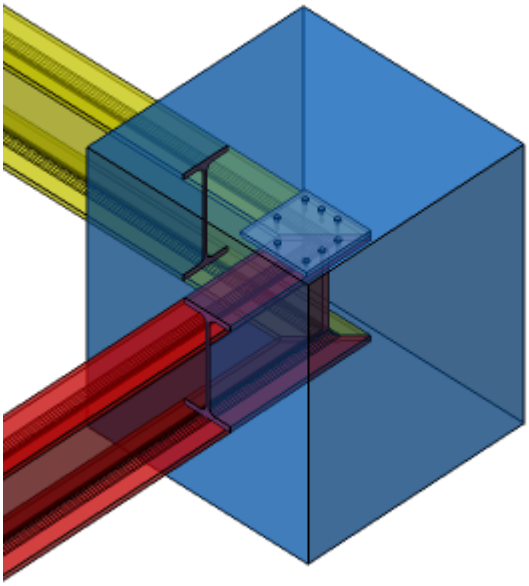
  In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail:

- 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

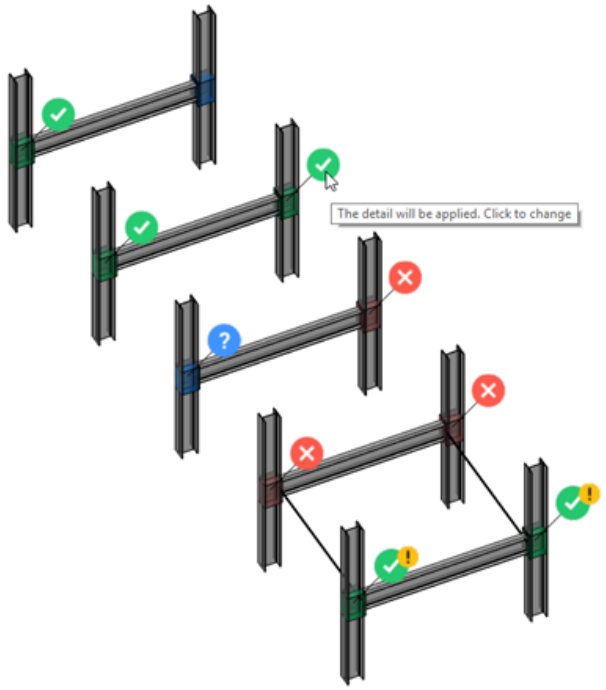
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

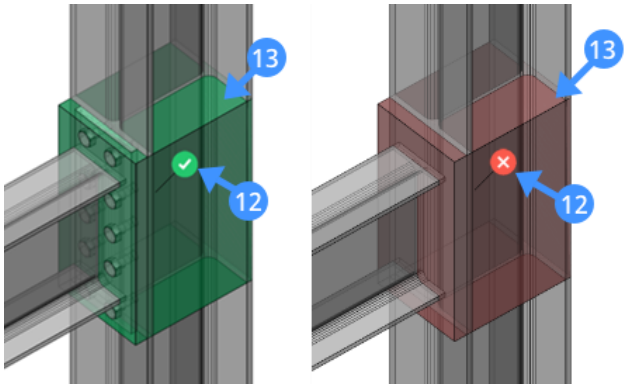
In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

	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:**

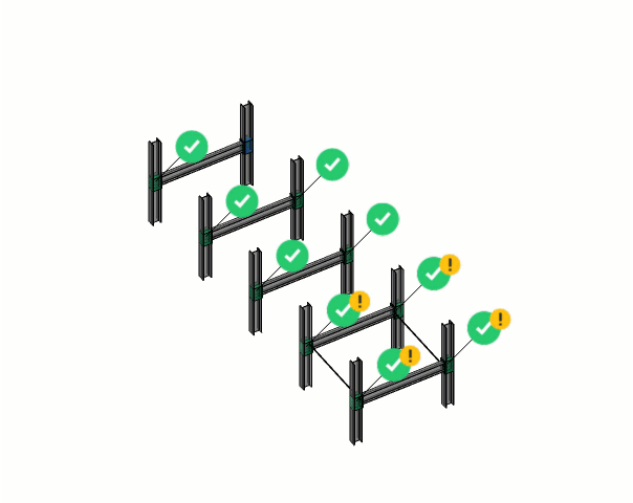
You are prompted:

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:

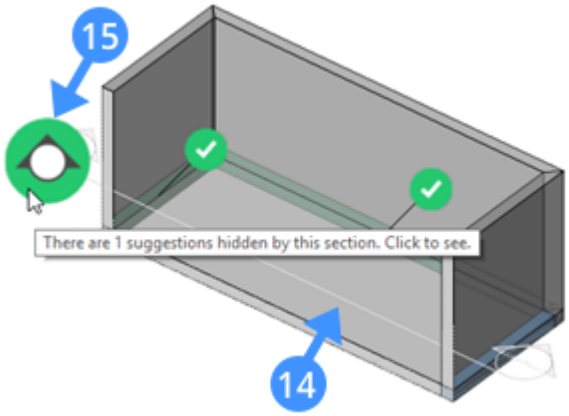
- Four widgets are selected and toggled off
- Enter is pressed: the four selected suggestions were all toggled off, so they are not applied, and the four suggestion are removed from the view
- Four more suggestions are selected
- Enter is pressed: the four selected suggestions are applied, and removed 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.



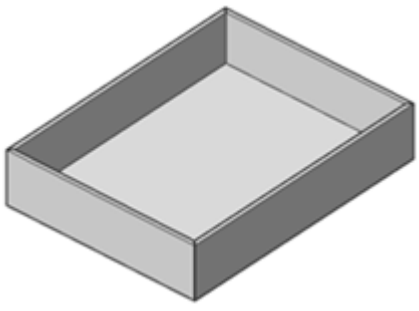
20.5.3 Propagate planar

20.5.3.1 Commands

BIMPROPAGATEPLANAR

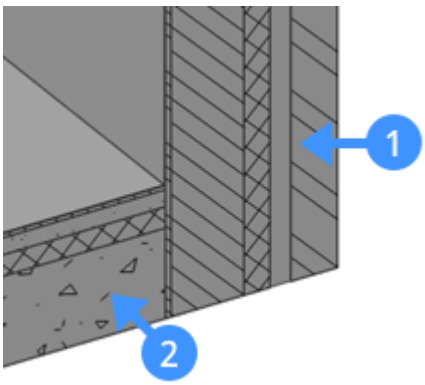
20.5.3.2 Procedure: propagating a wall-slab connection

1 Consider the following situation of four walls and one slab, created with **QuickDraw**.

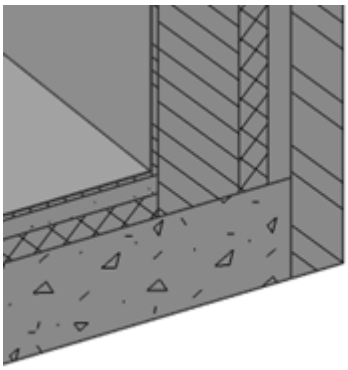


2 The four walls were given a composition, as well as the floor slab. Without any further editing or modeling, creating a vertical section through the building will show the connection one of the walls (1)

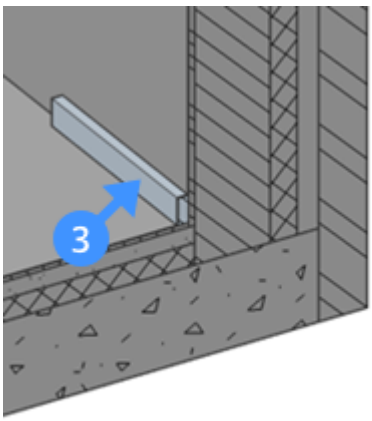
and the slab (2) to be something similar to the following image.



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.

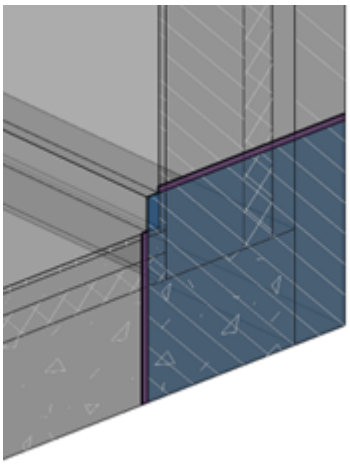
7 **You are prompted:**

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:**

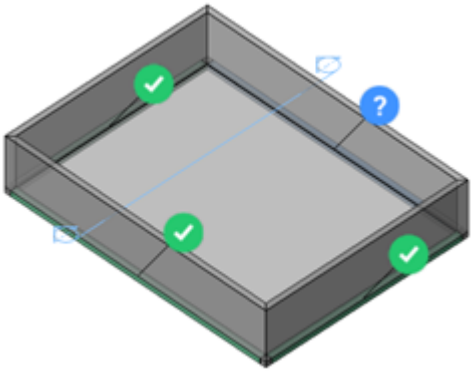
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

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.

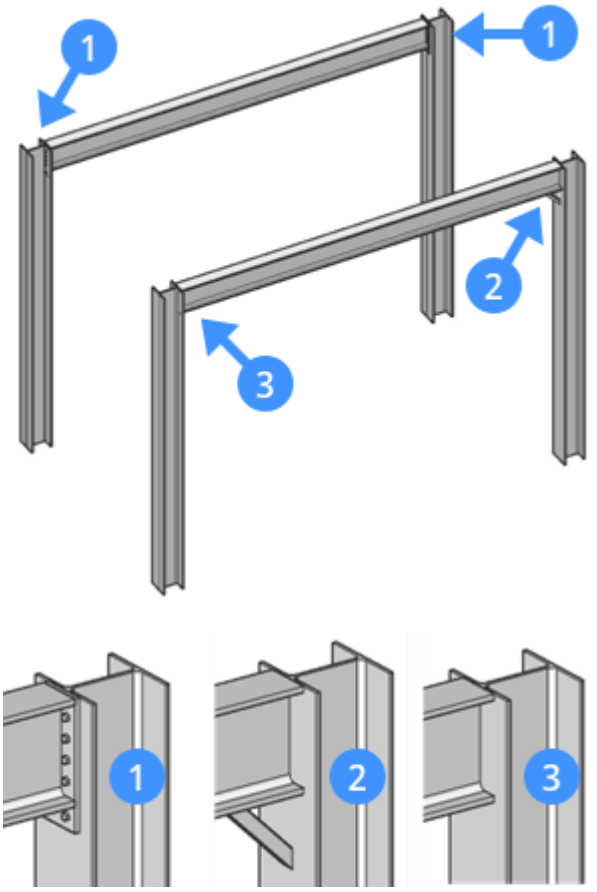


For more information about this command, visit the Command Reference article BIMPROPAGATELINEAR.

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>:

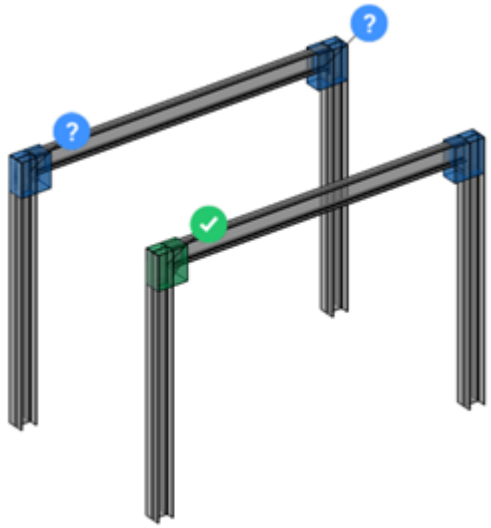
You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the Using Propagate article. Choosing **No** will exit the command.

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

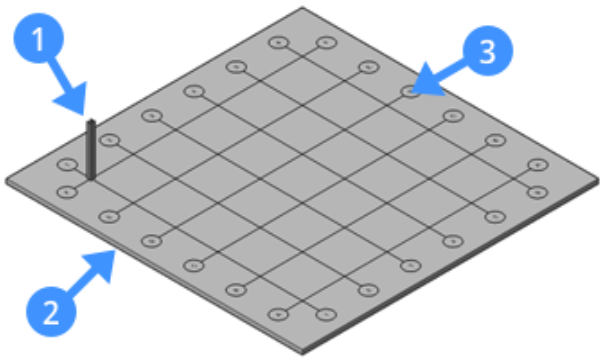
20.5.5.1 Commands

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.

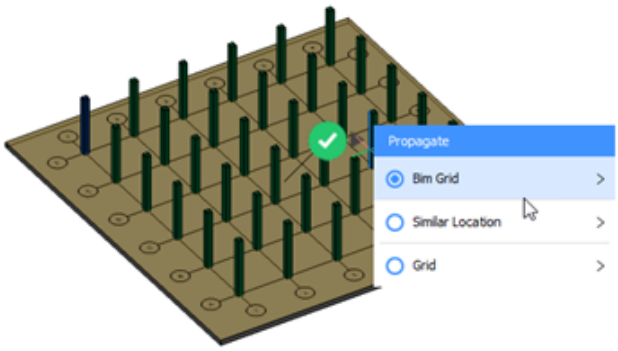
7 **You are prompted:**

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.

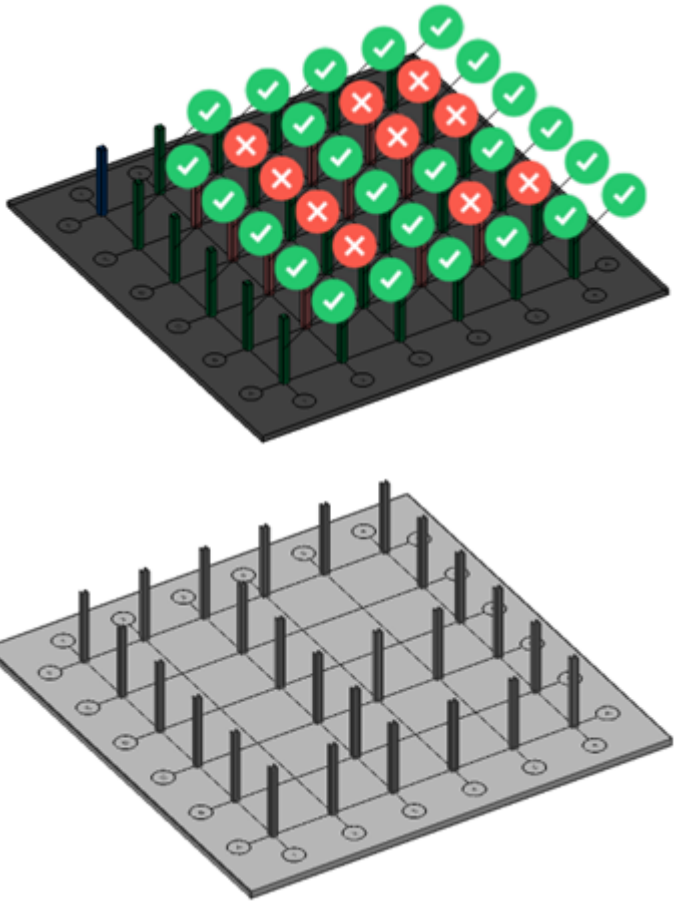
Press **Enter** to accept.

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:



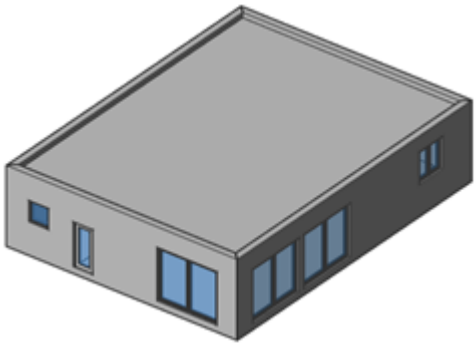
20.5.6 Propagate edges

20.5.6.1 Commands

BIMPROPAGATEEDGES

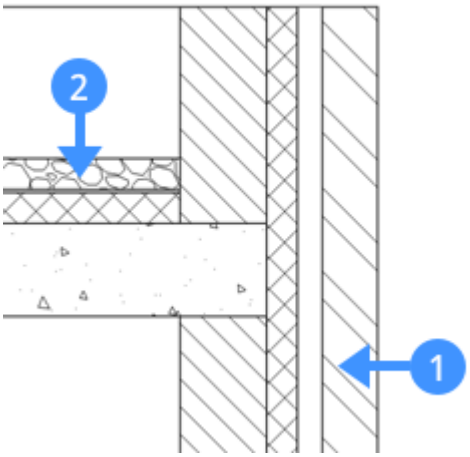
20.5.6.2 Procedure: propagating a wall cap

1 Consider the following situation of a one-story building with a flat roof.

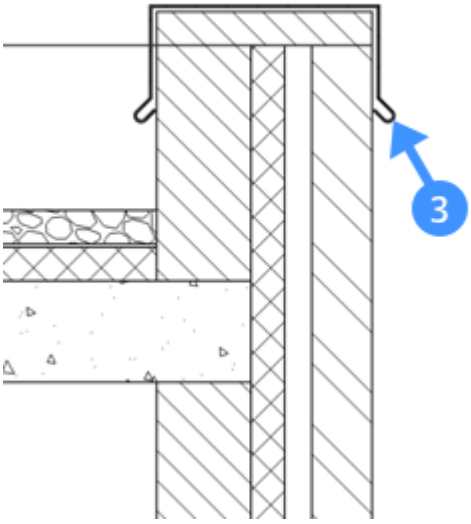


2 Let's create a vertical section plane to cut through our building and take a look at where the wall (1)

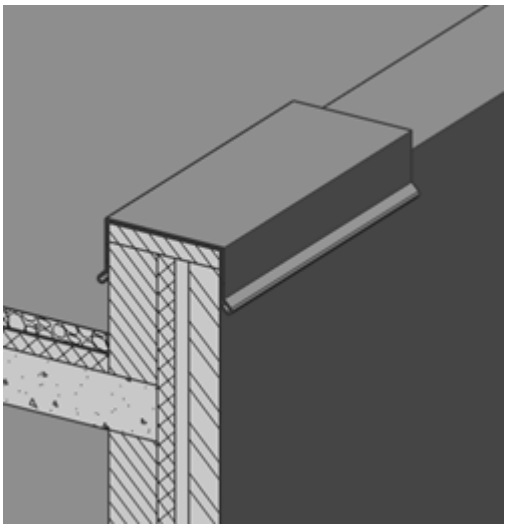
and the roof slab (2) come together.



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:**

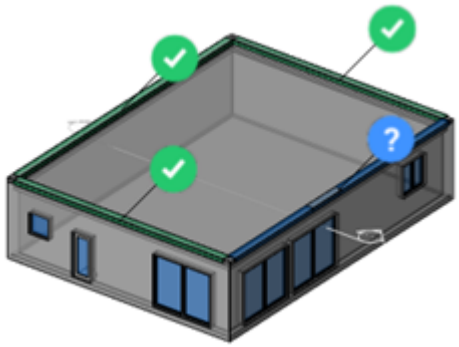
Choose location of detail relative to edge: [Outside reference solid/Inside reference solid] <Outside reference solid>:

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.

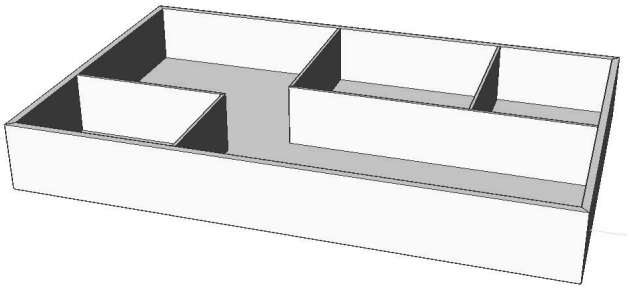
20.5.7 Propagate corner

20.5.7.1 Commands

BIMPROPAGATECORNER

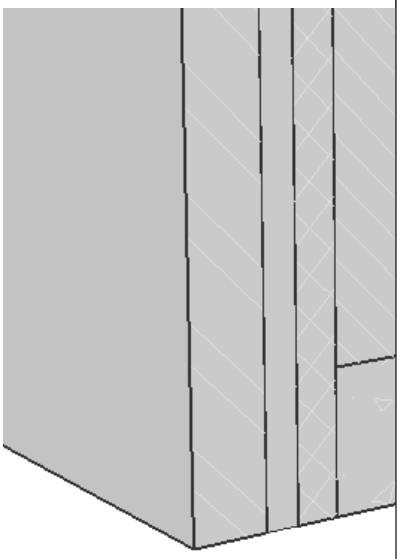
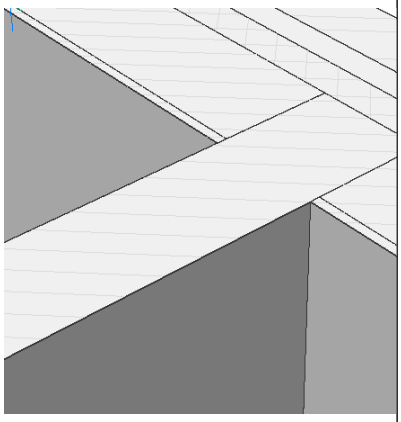
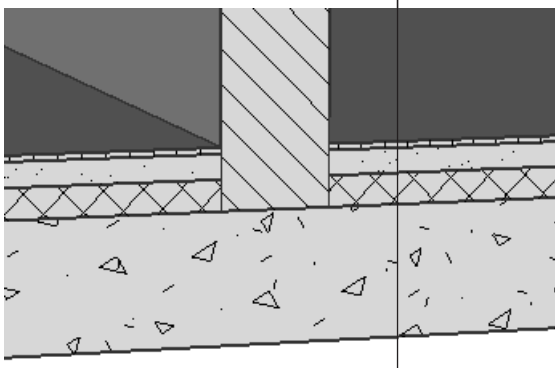
20.5.7.2 Setting the foundations

In this example the 4 exterior walls, 5 interior walls, and floor slab have already been assigned a composition.

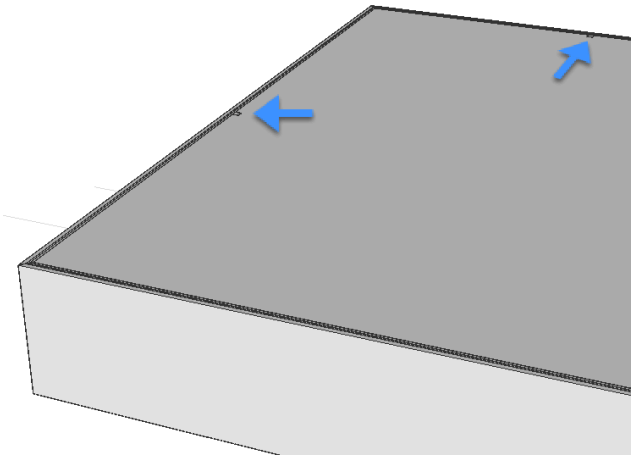


First, you must connect the 3 planar types of connections separately. For a detailed explanation of how to do this, read the article Propagate planar.

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.

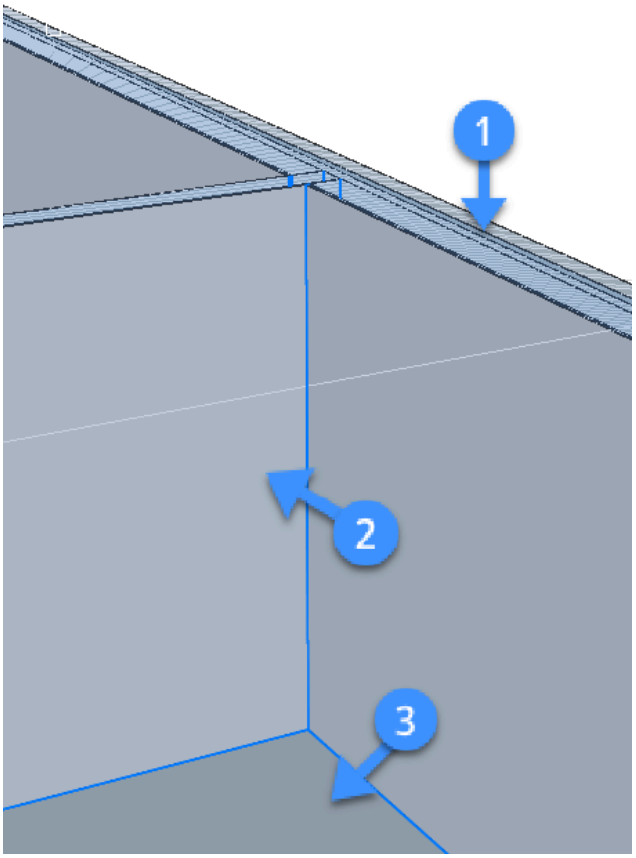
1 **You are prompted:**

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.

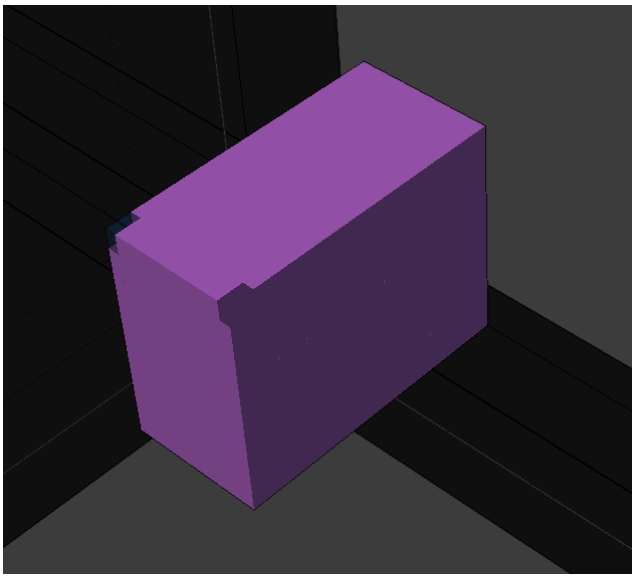
Uwaga: If you don't have any extra details, just press **Enter** to continue.

5 The volume that will be propagated is encased in a purple box.

6 **You are prompted:**

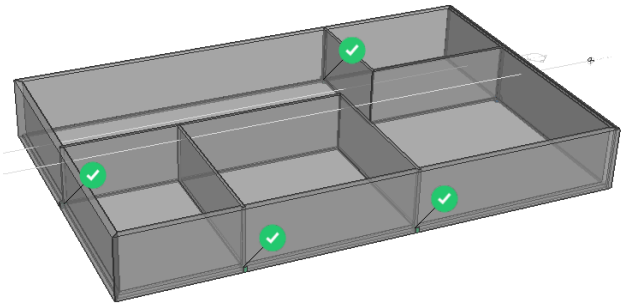
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

7 Press **Enter** to accept the detail.



Uwaga: 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

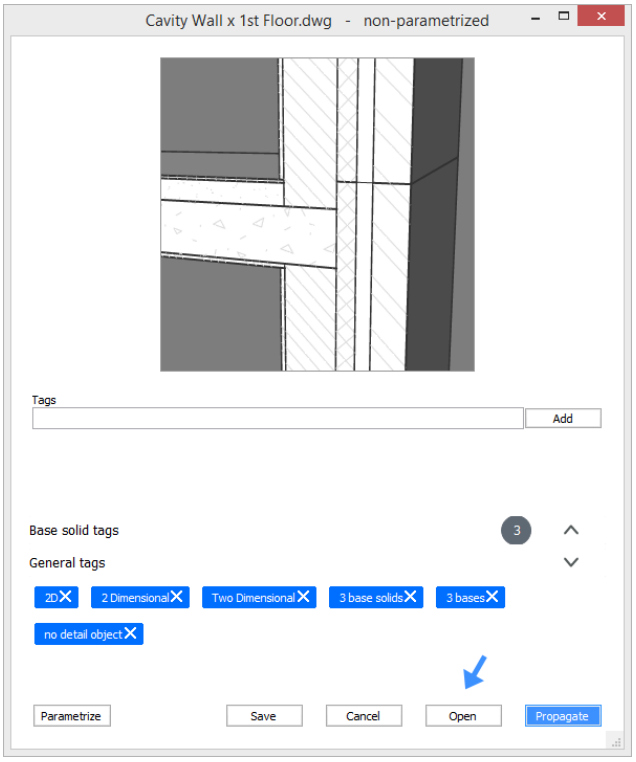
20.5.8.2 About BIMPARAMETRIZEDETAIL

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.

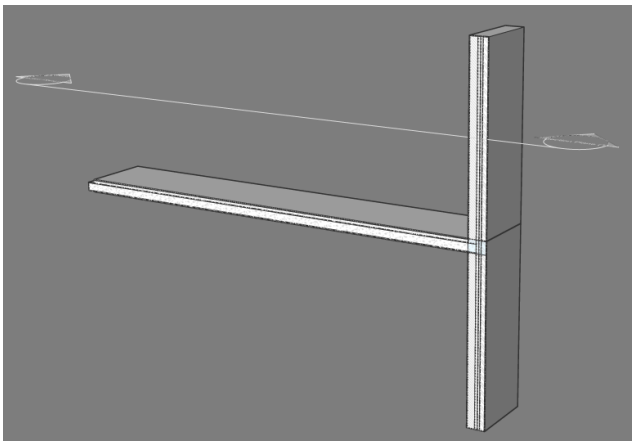
The automatically generated parameters form a framework that can control the position and shape of the reference solids. You can extend this framework adding custom constraints. You can evaluate the effect of these changes by animating the parameters from the **Mechanical Browser**. You can add custom 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**.

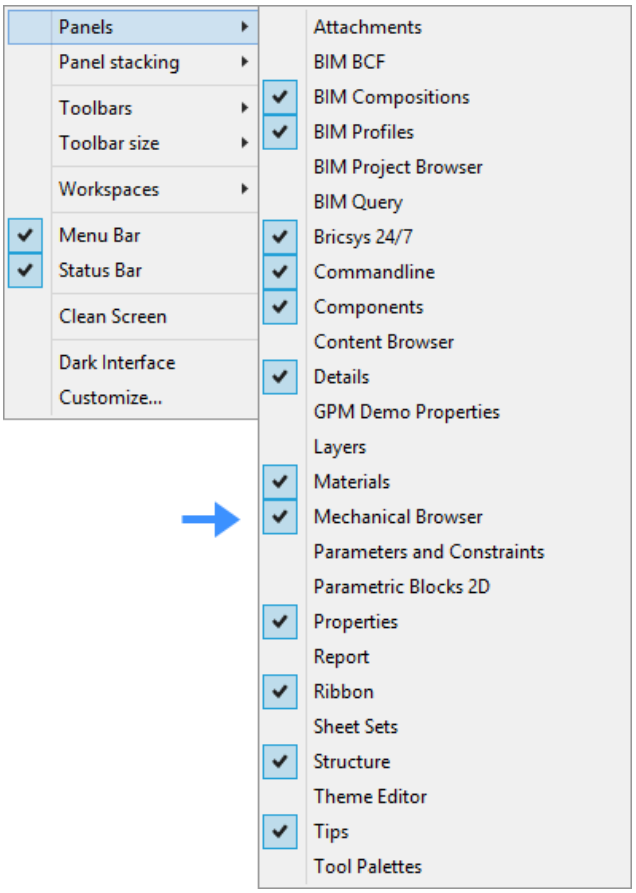
4 Click the **Mechanical Browser** button on the right of the screen.



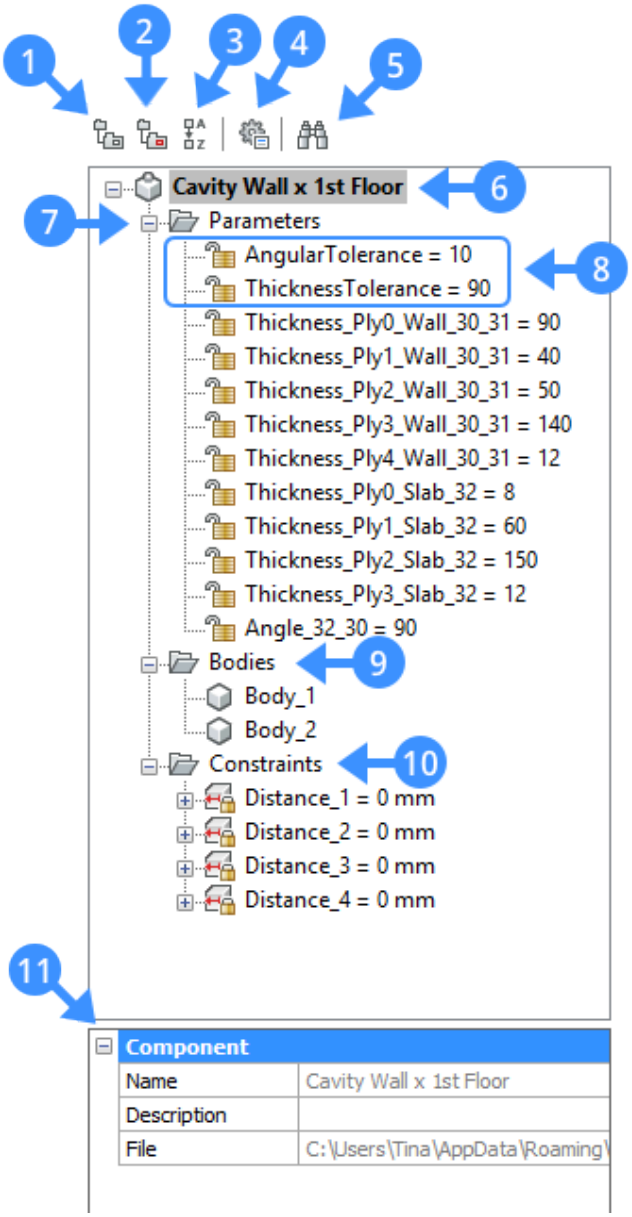
5 If the **Mechanical Browser** button isn't visible yet, move the cursor to the toolbar or ribbon panel and right-click. A context menu displays.

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:



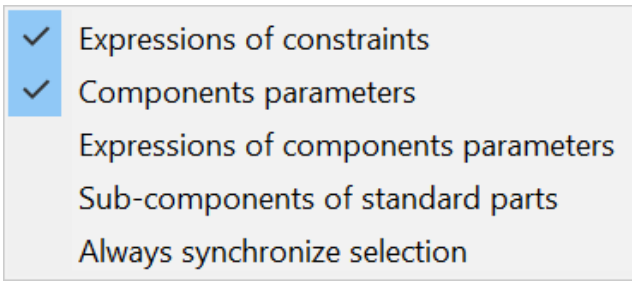
Uwaga: You can also use the MECHANICALBROWSEROPEN command.

(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:

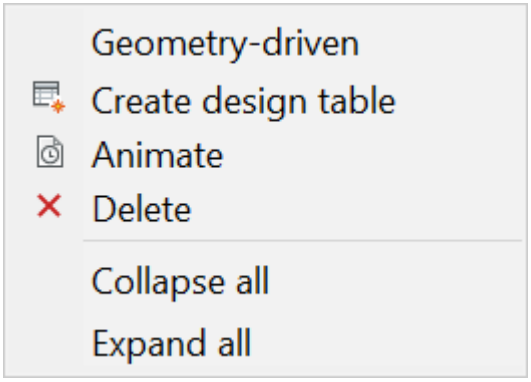


- **Expressions of constraints**:Controls whether the numeric value or the assigned parameter name displays.
- **Components parameters**:Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.
- **Expressions of components 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.

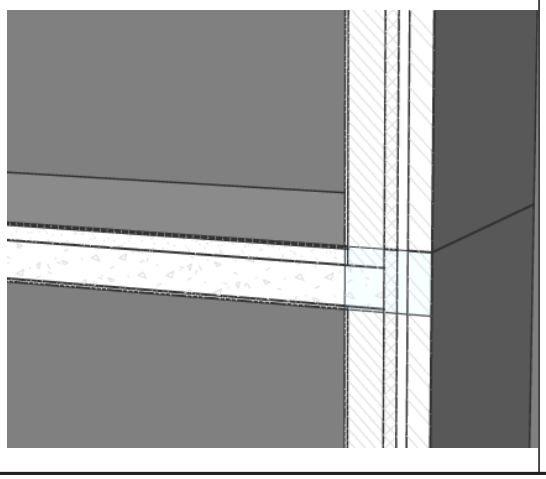
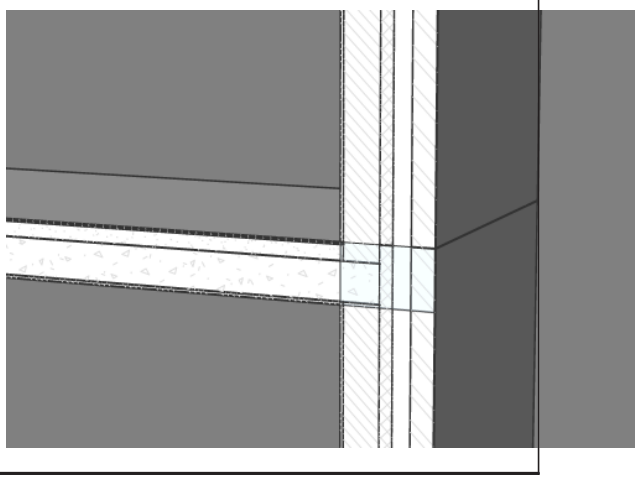
7 ***Uwaga:*** Some default constraints are generated that link the detail volume and the detail objects to the reference solids. You can delete and replace them with other constraints.

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.

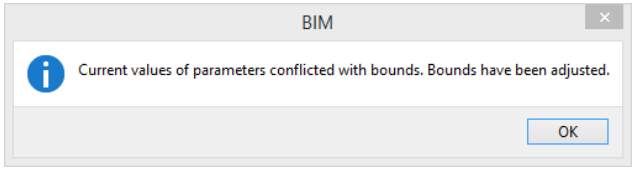
For example:

- 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
	

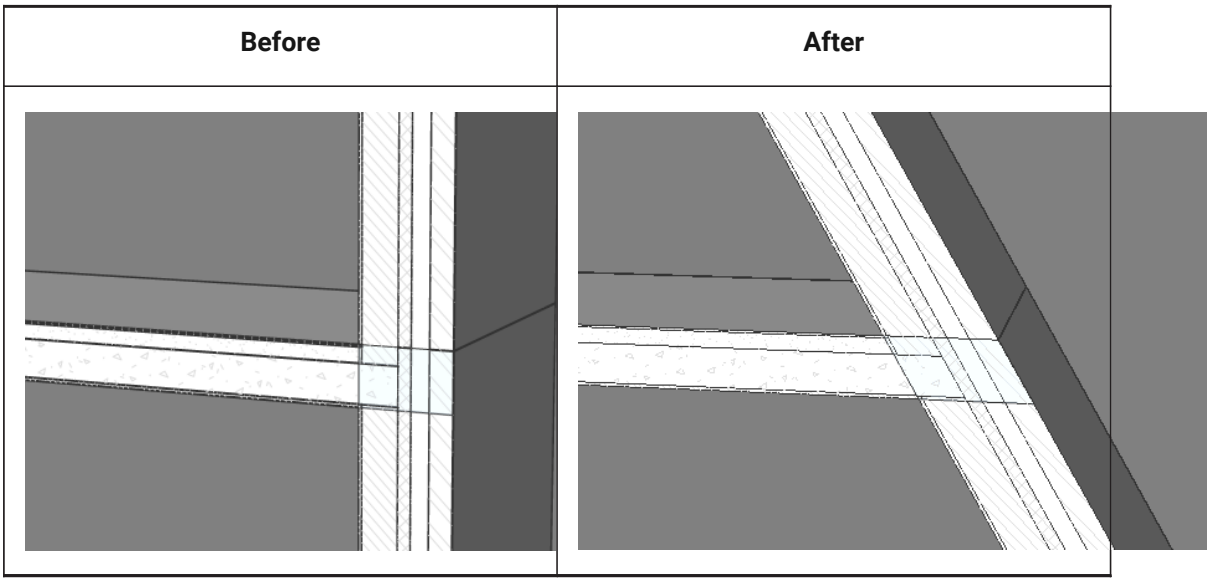
- Change the **Angle_32_30** parameter from 90 to 120. The new angle between the wall and the slab is different in the model space.

  A warning pops up:



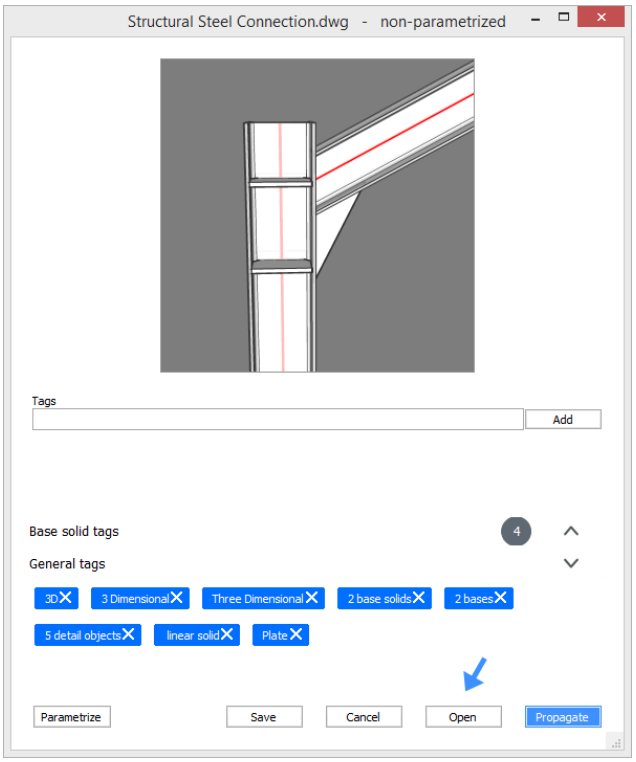
  This is because the **AngularTolerance** parameter was set to 10. The new **AngularTolerance** parameter automatically changes to 30 because you changed the **Angle_30_31** parameter by 30 degrees.

- Click **OK**



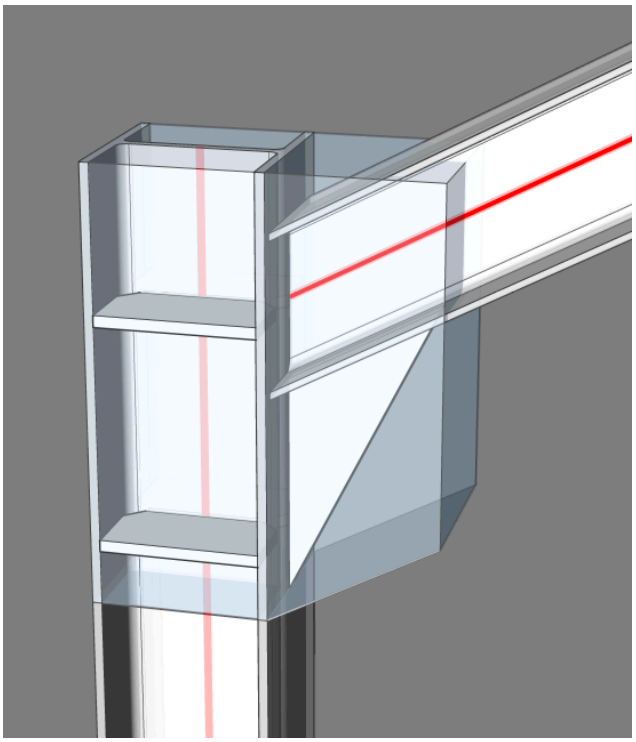
20.5.8.4 Parametrize and edit a structural steel connection

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:

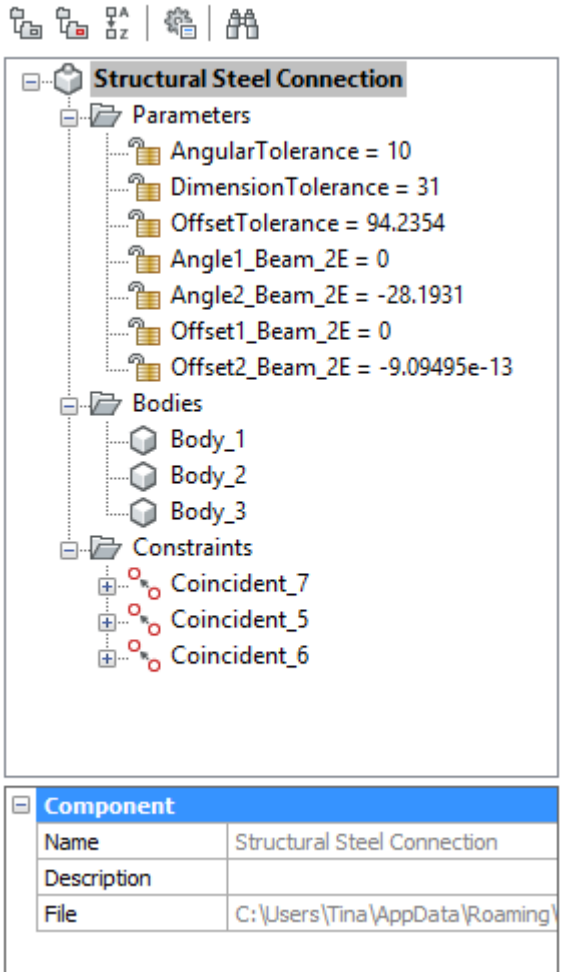


2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL.

The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**:



Uwaga: 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.

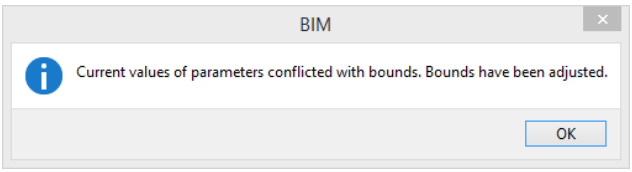
4 Edit parameters.

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.

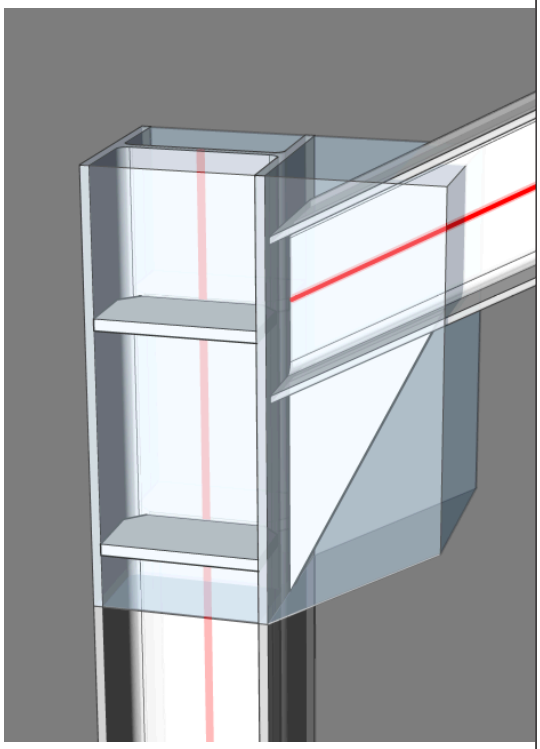
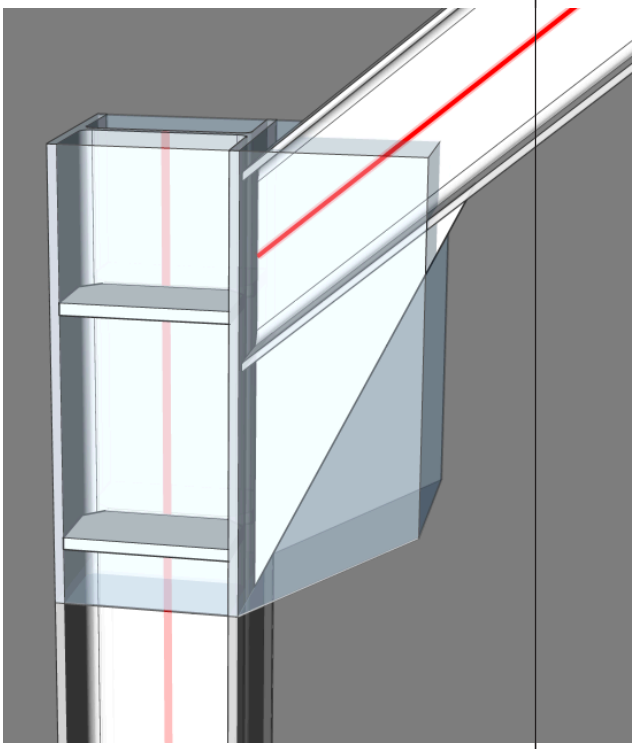
For example:

- Change the **Angle2_Beam_2E** from -28.1931013282 to -40. The column and the girder change in the model space.

  A warning pops up:

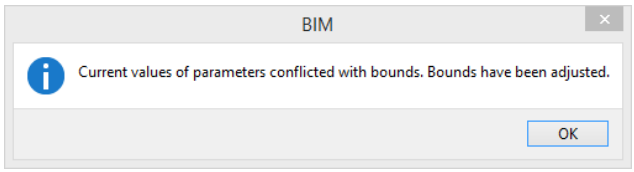


This is because the **AngularTolerance** parameter was set to 10. The **AngularTolerance** parameter automatically changed to 11.8069 because you have changed the **Angle2_Beam_2E** by 11.8069 degrees. Click **OK**.

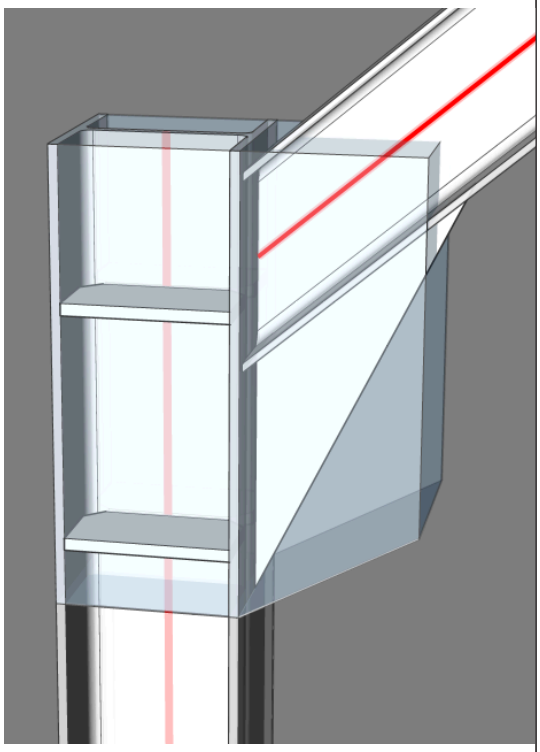
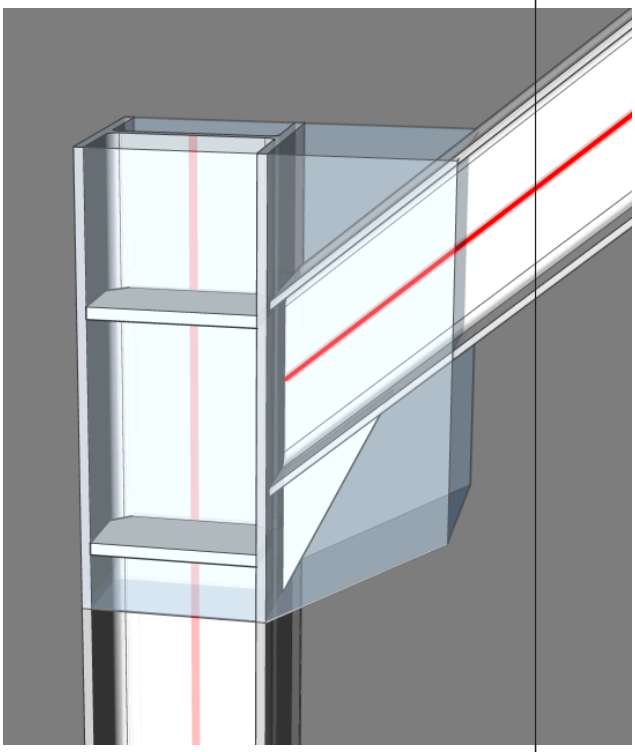
Before	After
	

- 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.

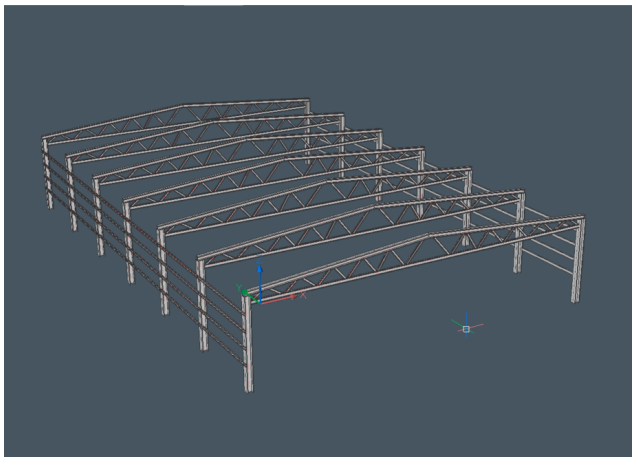
For example: if a non-parametrized detail of a wall-slab connection is propagated, BricsCAD BIM won't be able to propagate this detail to a wall-slab connection when the composition is different or a particular ply in the wall differs 10 mm from the corresponding ply in the wall of the original detail. 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.

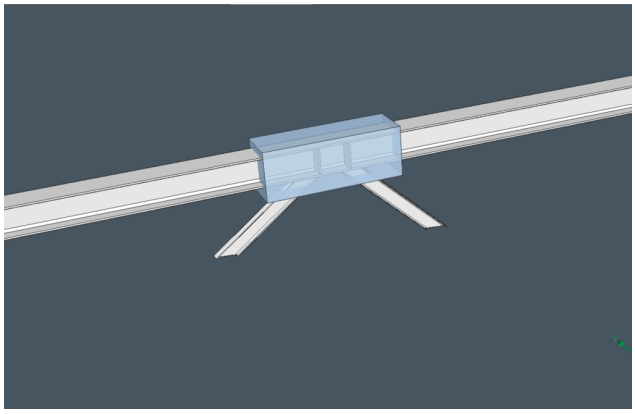
Different tolerance parameters:

- **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.

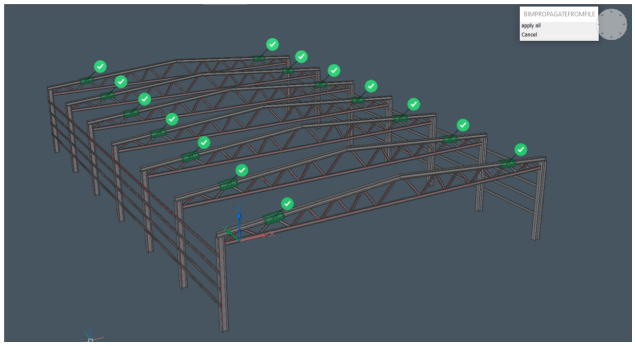
In the following project, there are a lot of similar connections present. However, the angles of the connections vary and the lower beam in the truss and has a different profile size than the upper beam.



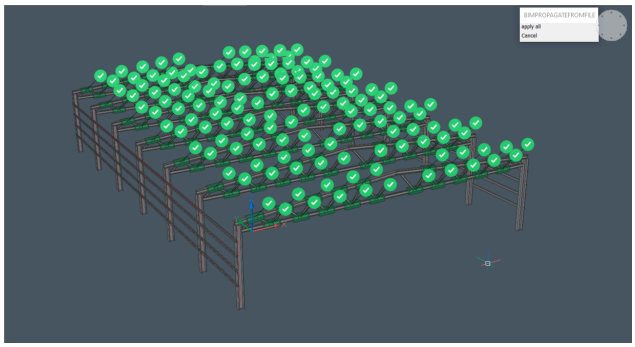
Propagate a detail in the project:

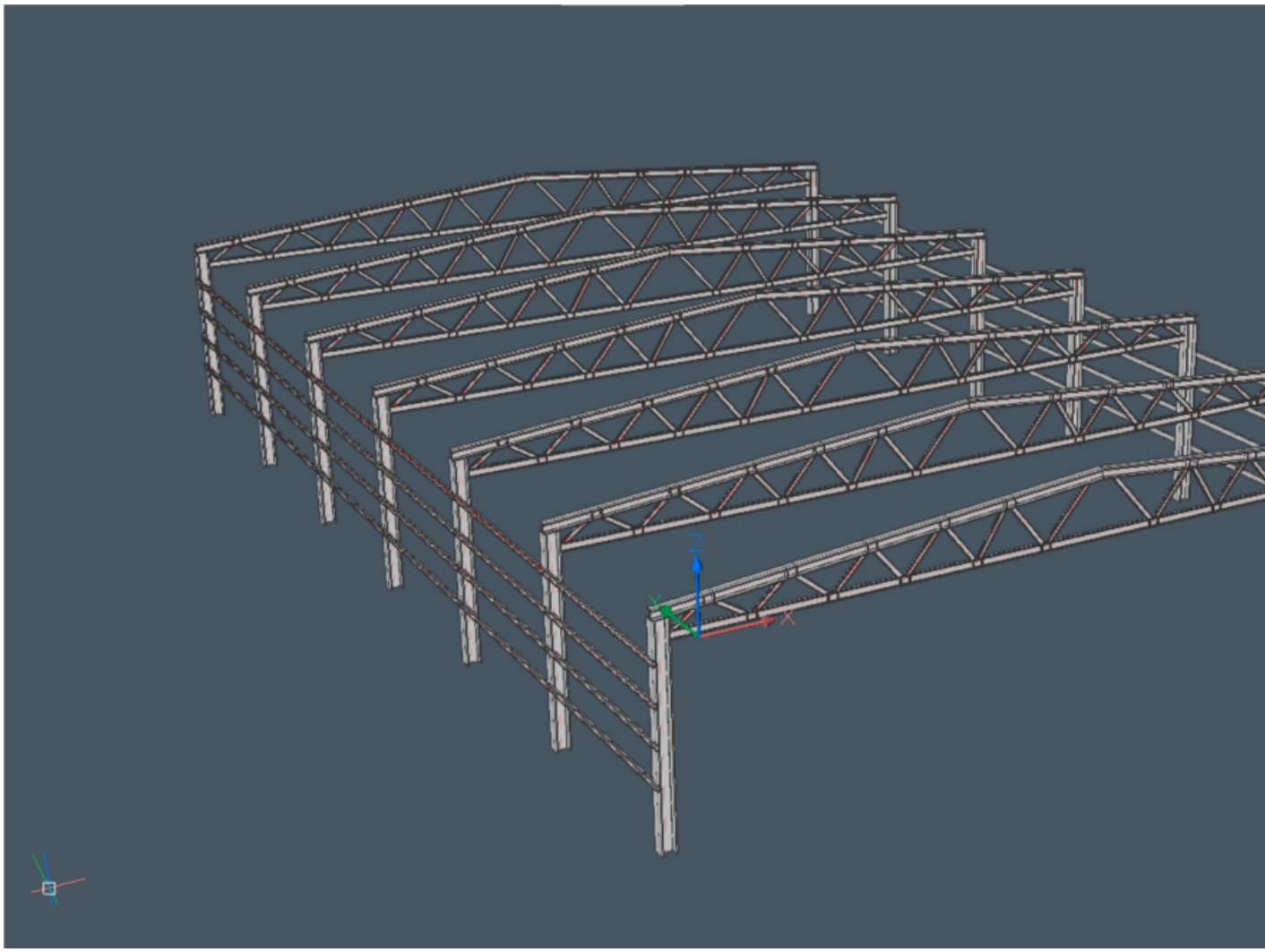


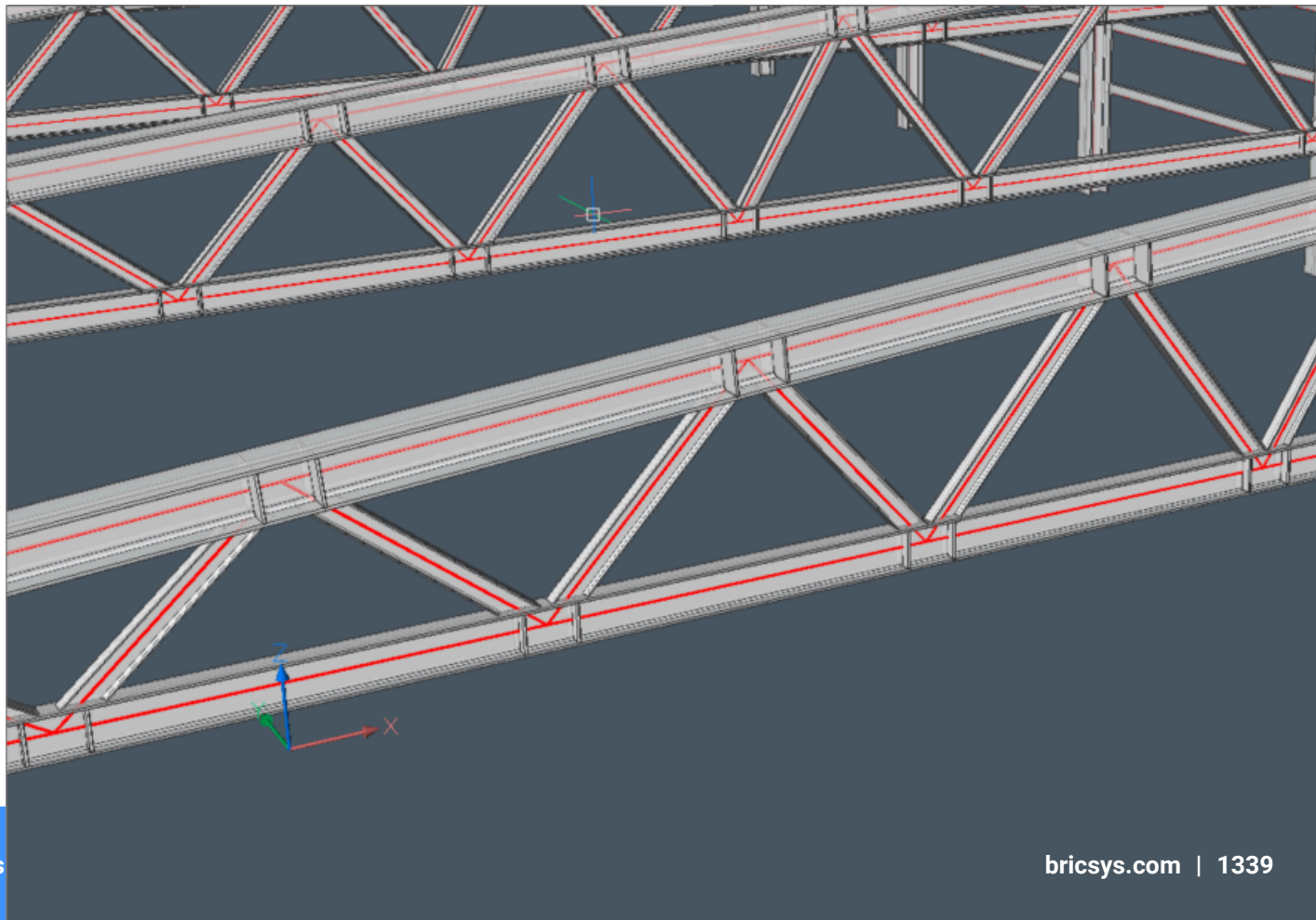
The **Mechanical Browser** displays the parameters. The **AngularTolerance** parameter is set to 0 degrees, the detail will only propagate to connections with exactly the same angle.



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.

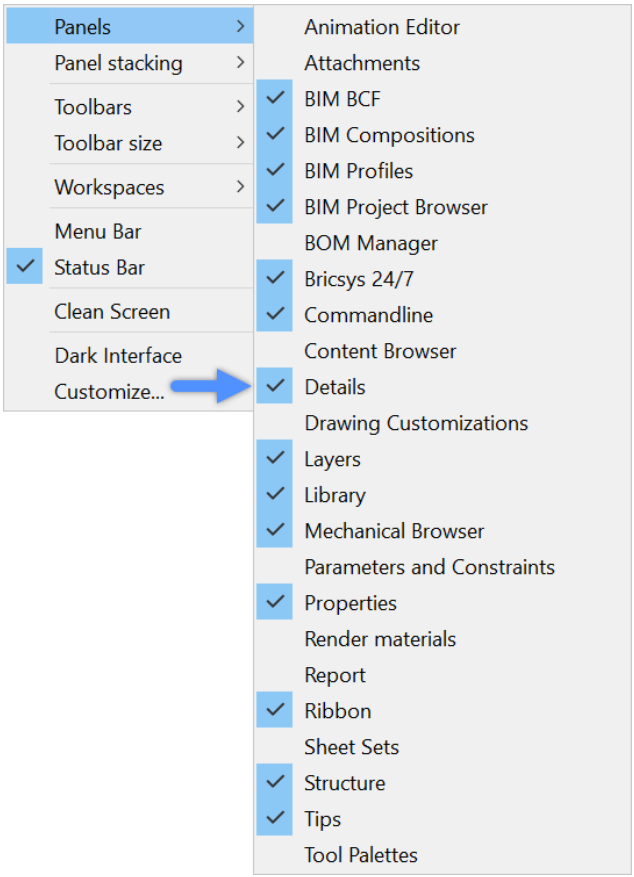
or

1 Place the cursor on a toolbar or ribbon panel and right-click.

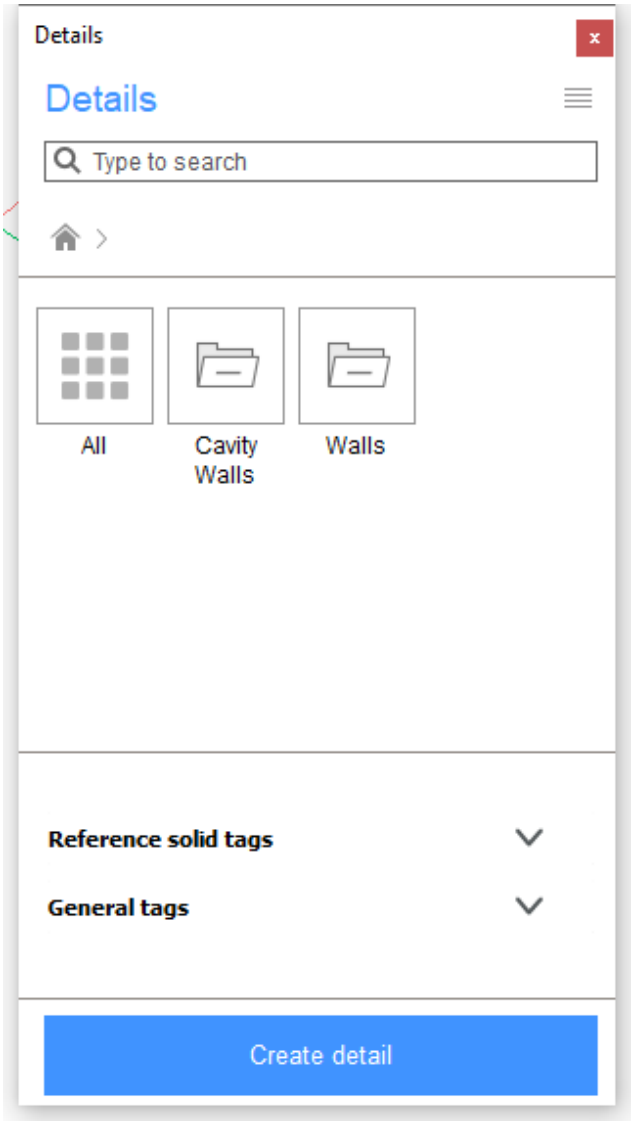
1 A context menu displays.

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.



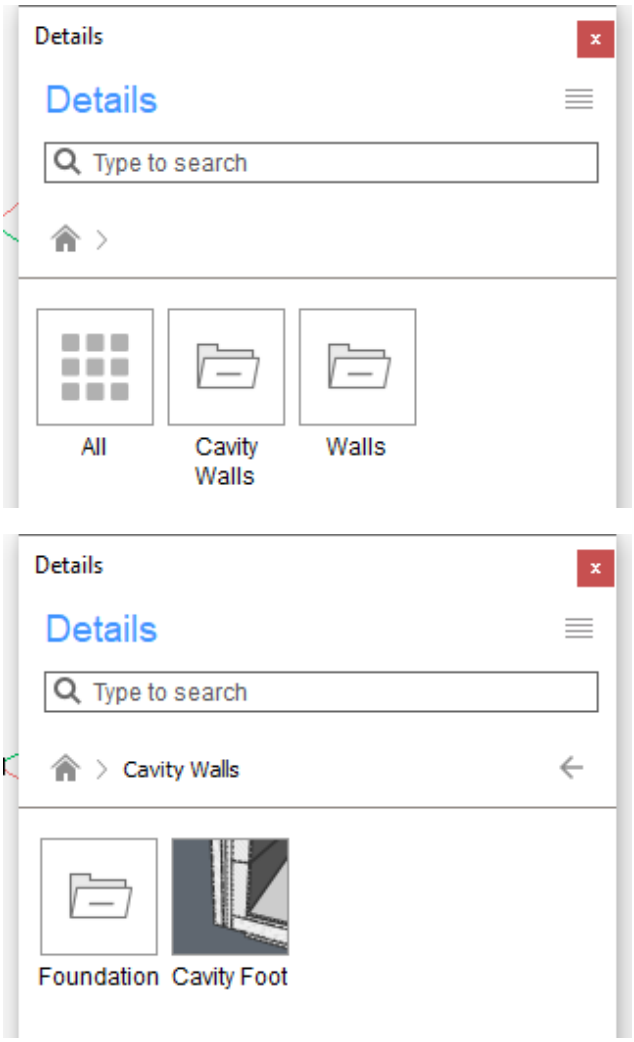
Click on the **Details** button ( ) to expand/collapse the **Details** panel.

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.

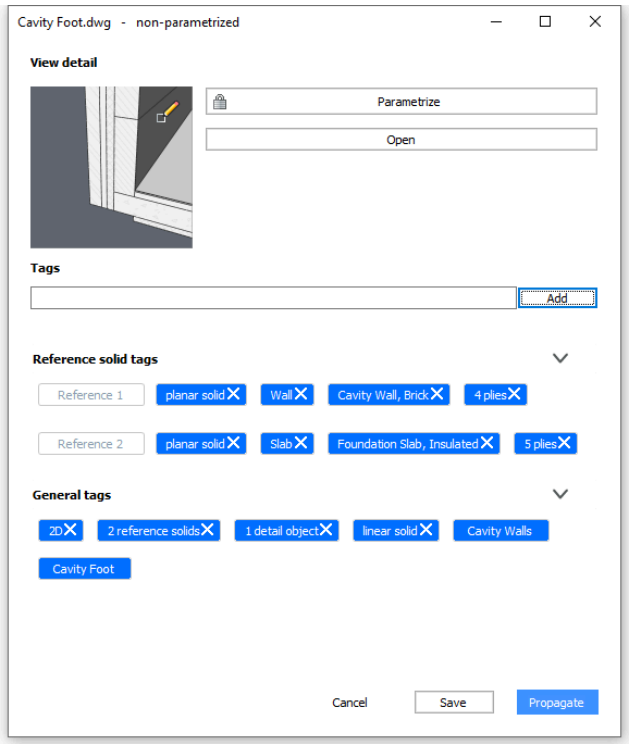
20.5.9.4 View a detail

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.

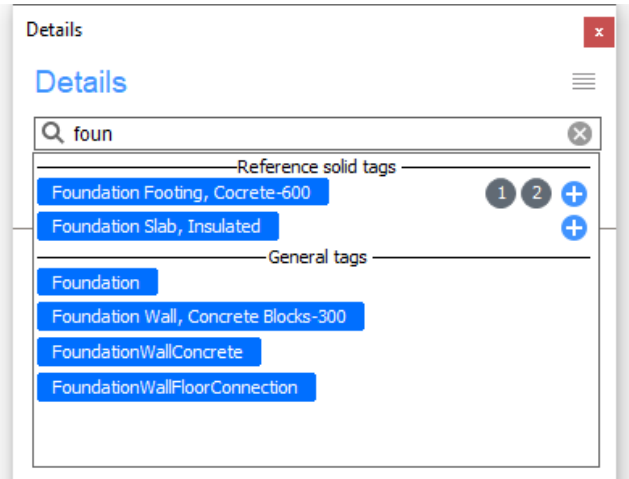
4 Click the **Save** button to save the changes and close the **Detail** dialog box.

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**.



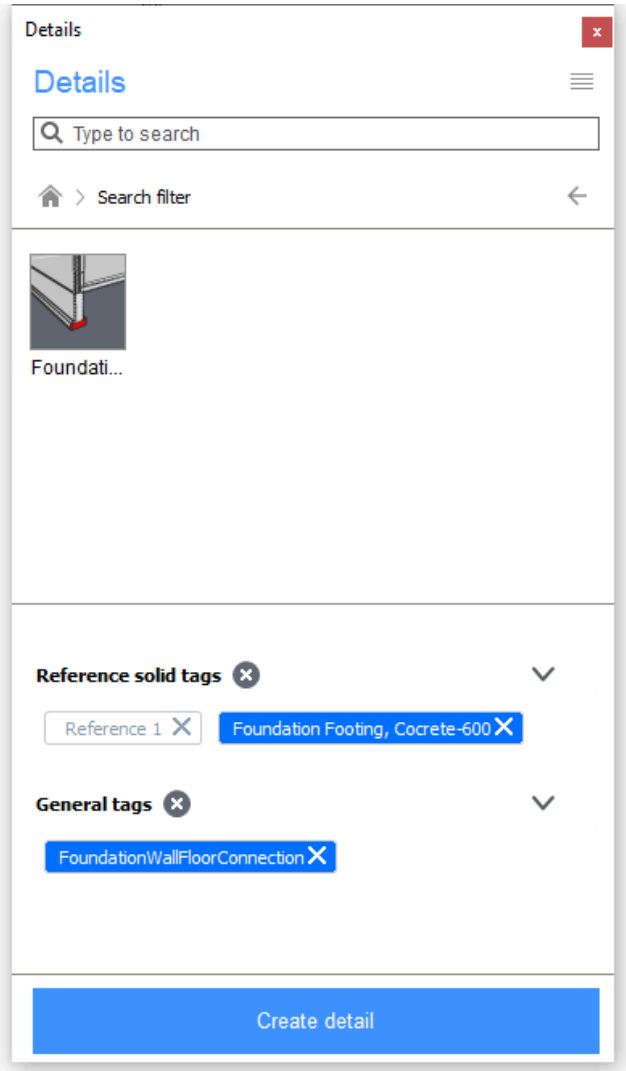
2 Do the following:

- **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.

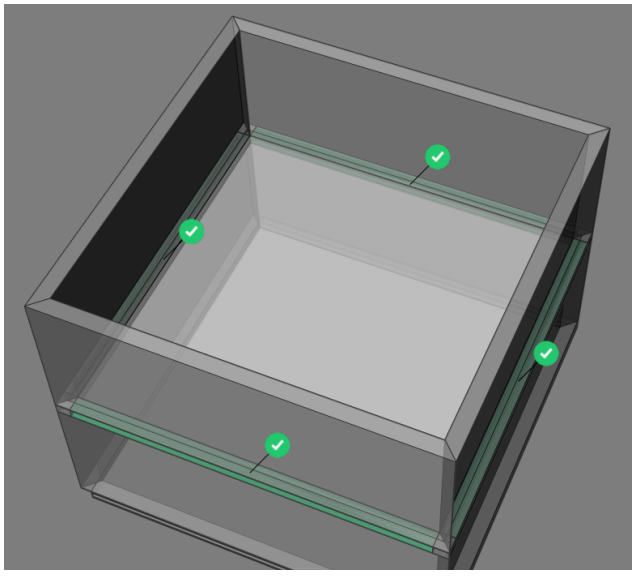
1 The **Detail** dialog box dispays.

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.

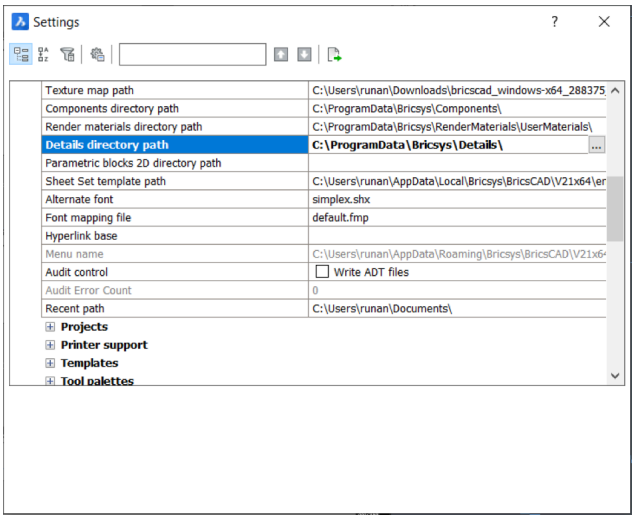
20.5.9.7 Sharing details

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.

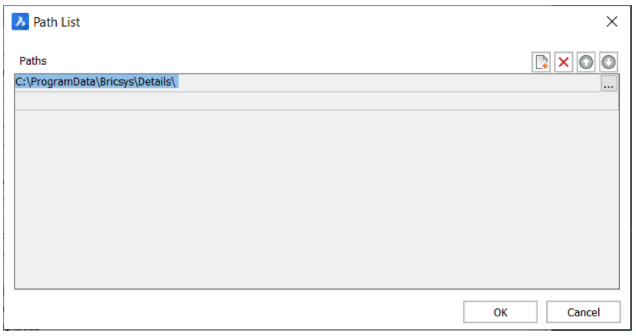
1 If you want to make a new detail to share, you can create a new detail first, as is explained above;

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.

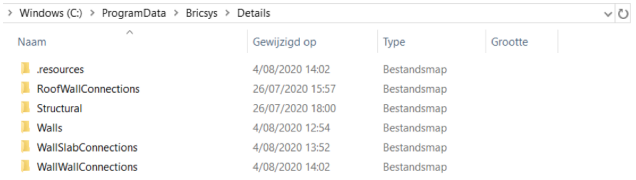


You can also create additional paths.

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.



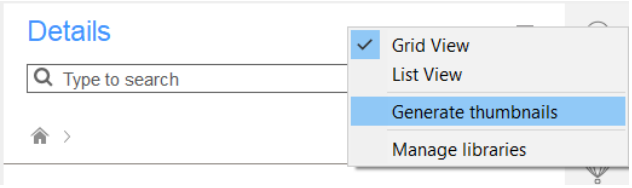
5 Copy the folder that you want to share and turn it into a zip file. This zip file can be shared with others.

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*.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

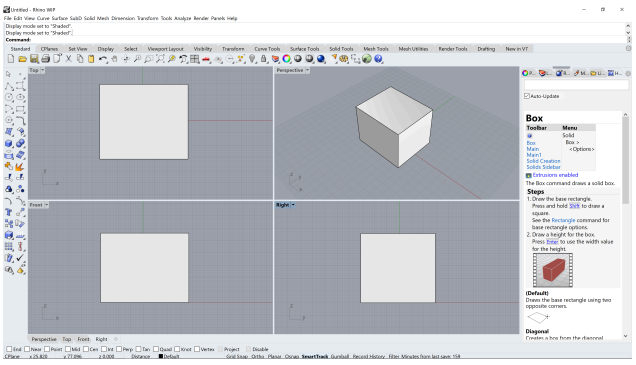
3 In BricsCAD, click on **Grasshopper** > **Rhino**.

4 Draw a Rhino box by typing BOX in the Rhino Command line. (This works in the same way as the BricsCAD BOX command).

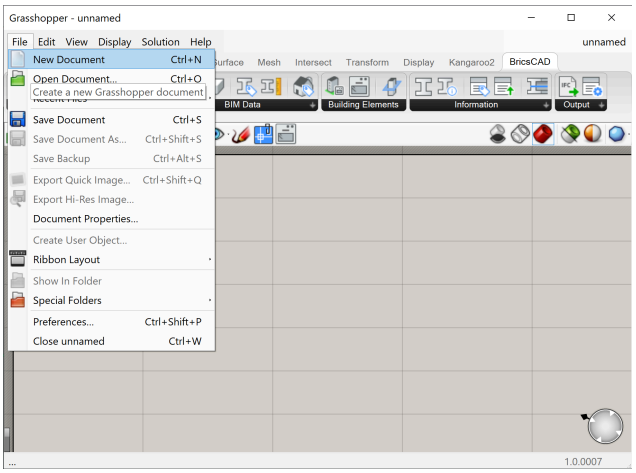
5 Click for the first corner of the base.

6 Click for the other corner of the base.

7 Click to define the height of the box.



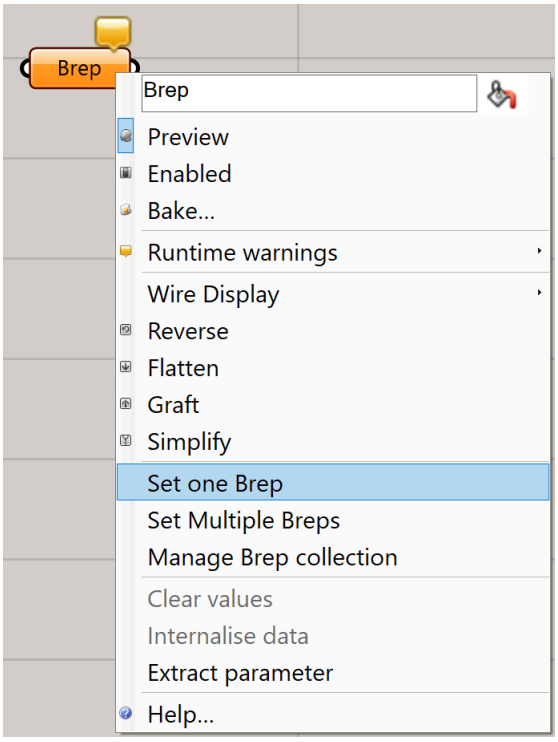
8 To open a new Grasshopper script, inside the Grasshopper environment, click on **GrasshopperGrasshopper** then click on **File** > **New Document**.



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**.



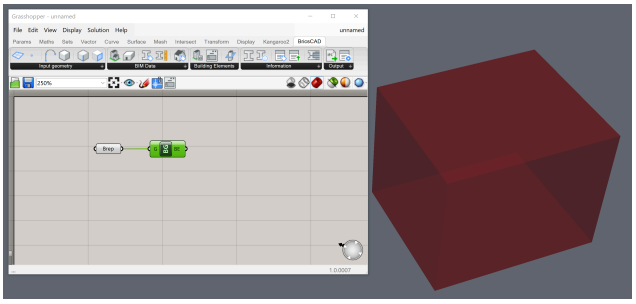
12 Go to the Rhino canvas and select your cube.

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.

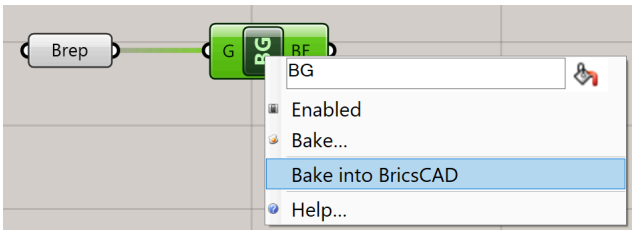
14 Drag and drop the **BricsCAD** > **Building Element** > **Bake Geometry component** onto the Grasshopper canvas.

15 To link the two components, click and drag from the Brep component's right dot to the BricsCAD Bake

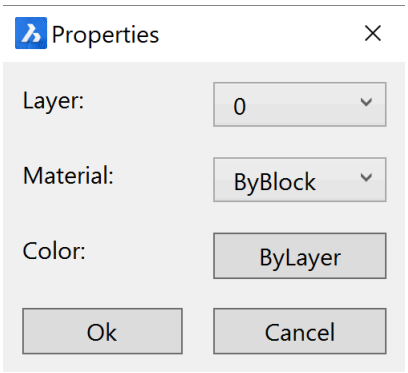
component's left dot.



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**:

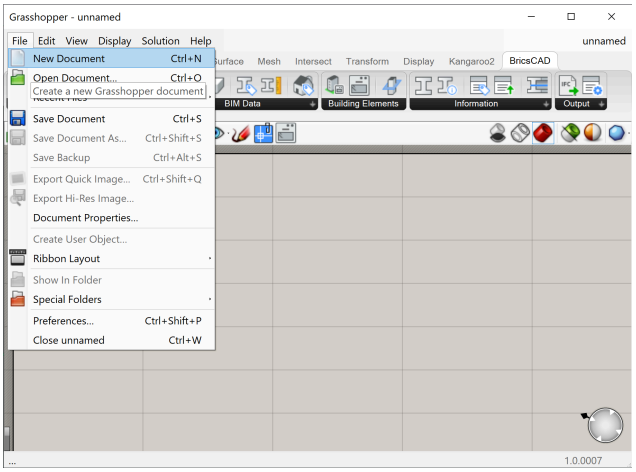


17 Click **OK**. The box is baked as a solid into BricsCAD.

20.5.10.7 How to draw a box in Grasshopper from a BricsCAD rectangle

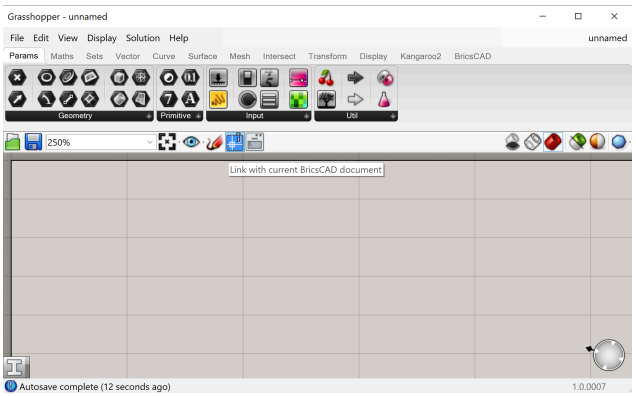
1 To open a new Grasshopper script, click on **Grasshopper** > **Grasshopper** and clicking on **File** > **New**

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.

Uwaga: 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.

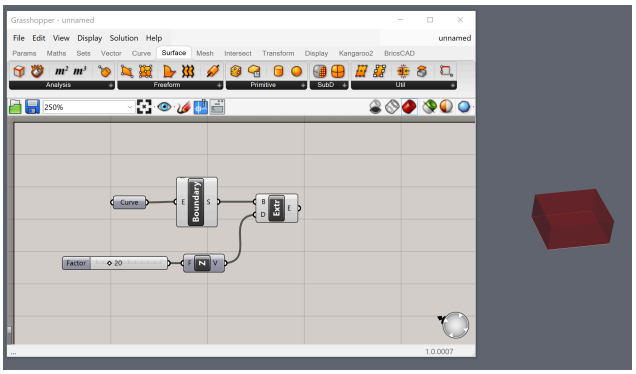
Uwaga: Save time with the Quad: select the curve in BricsCAD and then select **ToGrasshopper** under the **Grasshopper** tab of the Quad. A Grasshopper component appears in the upper left corner of your Grasshopper canvas.

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.

Uwaga: 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.

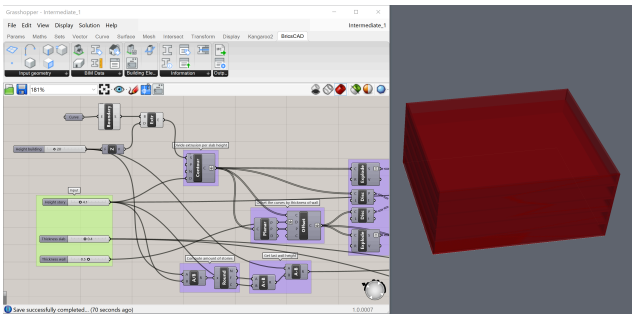
20.5.10.8 How to split up a box into different floors using Grasshopper

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.

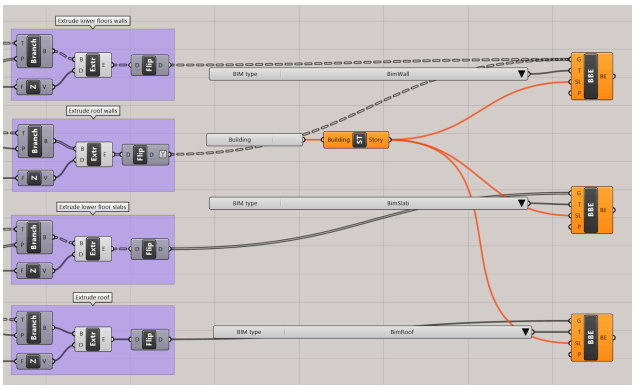
9 Attach the extruded elements (the Flip components) to the Geometry input nodes of the **Bake Building Element** components:
   - Connect the extruded lower floor walls and extruded roof walls to one **Bake Building Element** component. (Hold Shift while connecting multiple nodes to one input node).
   - 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.

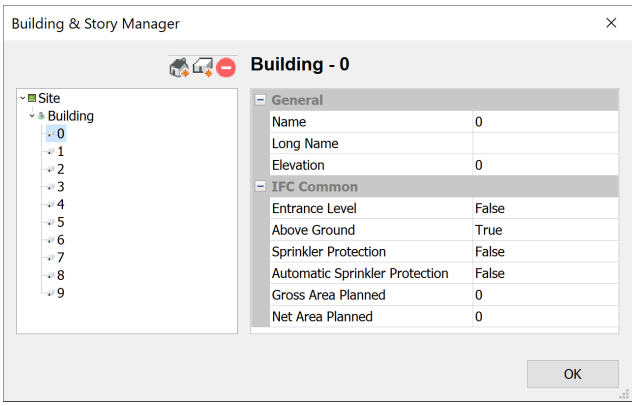
12 Attach BIM Types (set respectively to **BimWall**, **BimSlab**, and **BimRoof** in the selection menu) to the Element Type input of the **Bake Building Element** components of the extruded walls, the slabs, and the roof respectively.

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.

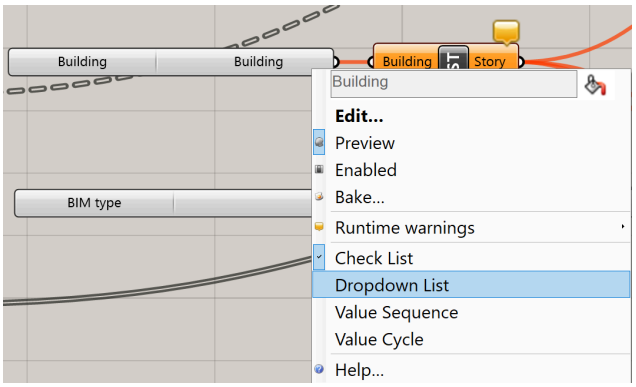


14 To make spatial locations, for the component to detect, go to BricsCAD and click on **Home** > **Classify** > **Spatial Locations**.

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**.

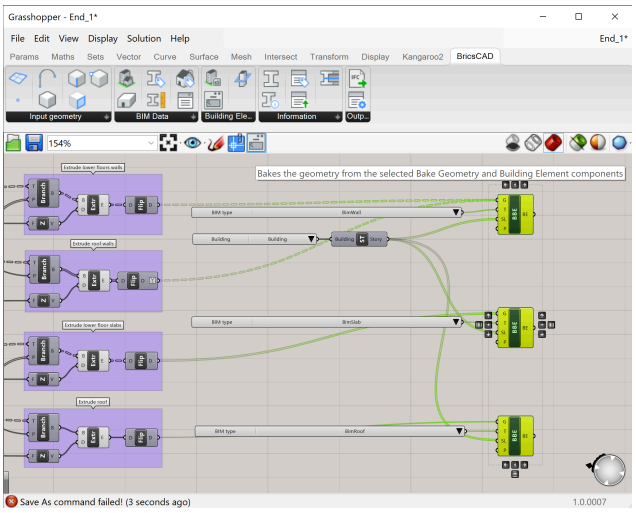


17 Select the newly made building in the drop-down list.

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.

Uwaga: 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.

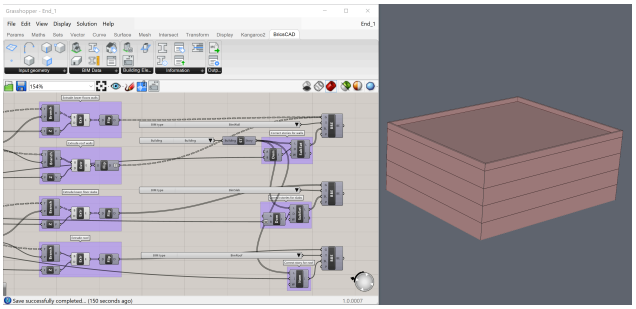


20 The last input geometry will be baked onto all remaining stories as there are now more stories than geometry. To correct this, select a correct subset of the stories to assign. Bake the geometry in BricsCAD and the stories will be correct. No 'duplicate' geometry will be baked.

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.



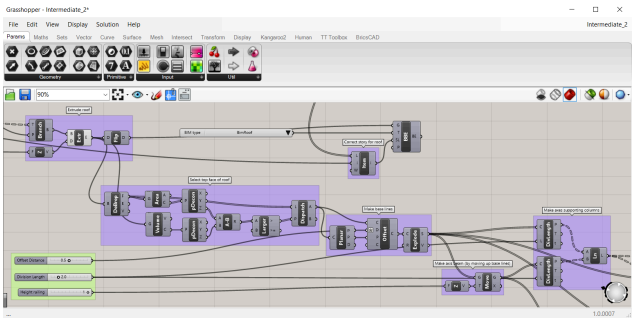
20.5.10.9 How to add a railing to the top of the building

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.

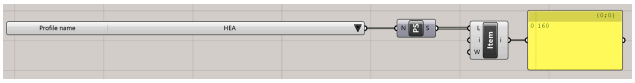
7 Select **HEA** from the drop-down list in **Profile Names**. This drop-down list displays all the names of the profiles in the **Profile Library** of BricsCAD.

8 Link the component to the input **ProfileName** of the **Profile Sizes** component.

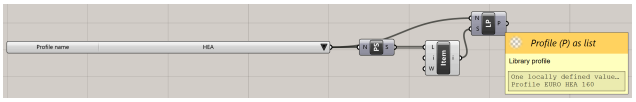


9 Link the **Profile Sizes** component to the input node **List** in a **Sets** > **List** > **List Item component**, where i is set to '4' (right-click on the **i** input and **Set integer** to **4**). This **List Item** lets you pick one size from the list with all available profile sizes for the **Profile Name** linked to it.

Uwaga: If you link a **Params** > **Input** > **Panel component** to the Profile Sizes output, you will see that the index 4 stands for size 160.

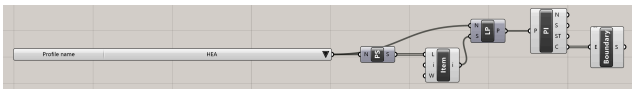


Link the **Profile Names** and **List Item** components to the **BricsCAD** > **Information** > **Library Profile component**, to get an HEA 160 profile as an output.



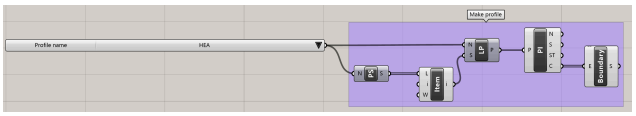
10 To get the curves that make up this profile, attach a **BricsCAD** > **Information** > **Profile Info component** to the **Library Profile**. One of the outputs created is the **Profile Curves** as a tree.

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.

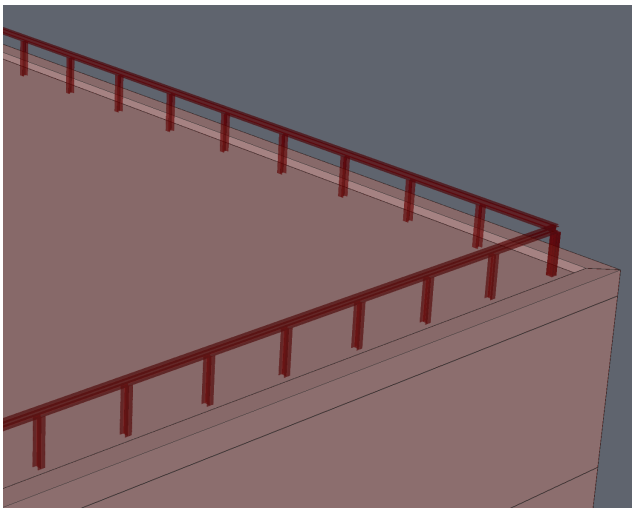


12 To group together the components just created, select them (with a selection box) and right-click on **Group**.

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.)

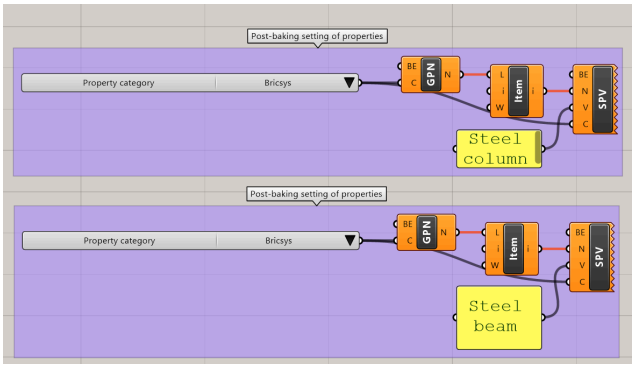
20.5.10.1 Procedure: add information to a baked model 0

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'.

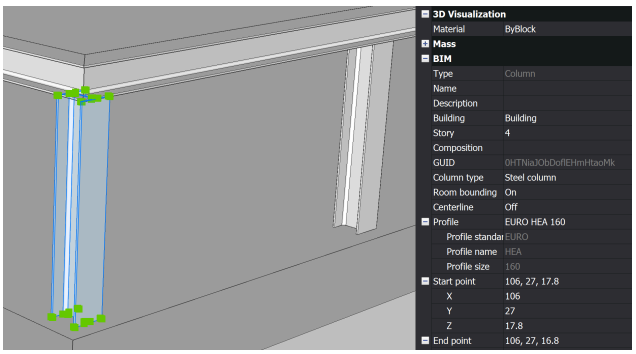
1 Add the 'Start_3.ghx' code to the canvas.
- You do that by downloading the zip file at the bottom of this page and extracting its content.
- Then open the 'Start_3.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
- You can now select all of the Grasshopper code 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 Współpraca nad projektem

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

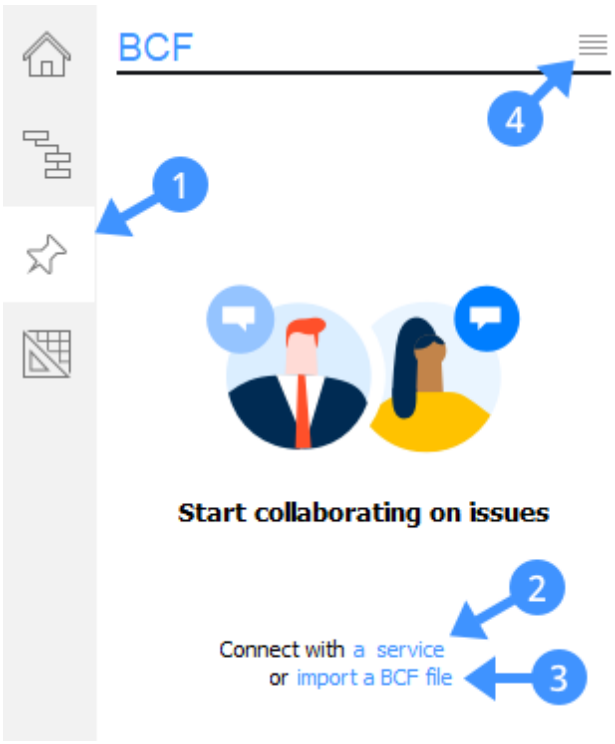
It is generally easier to use cloud services to manage BCF issues. Currently, BricsCAD BIM supports connection to three such services: **BIMcollab**, **BimSync**and **BIMtrack**. If you wish to connect to these services from within BricsCAD BIM you will need an account of these services first. These 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.

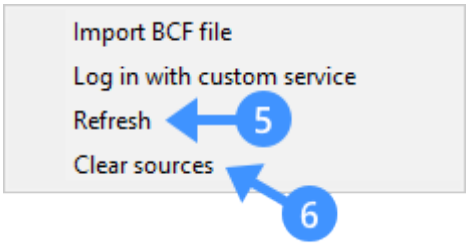
20.6.1.4 The BCF panel

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.



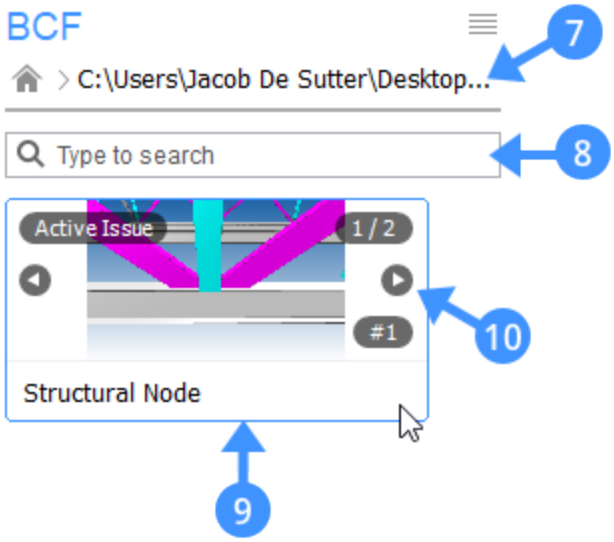
Refreshing the panel (5) will make sure you see the latest updates in case you are working on the cloud. **Clearing the sources** (6) will disconnect you with the cloud service in case you are online or close the .bcf file in case you are working on an imported .bcf file.



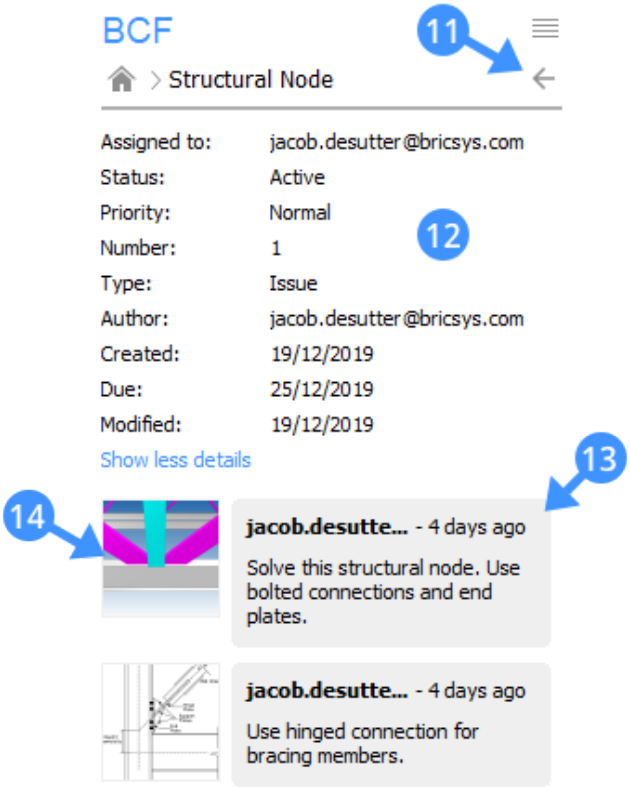
20.6.1.5 Procedure: working with the BCF panel

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.

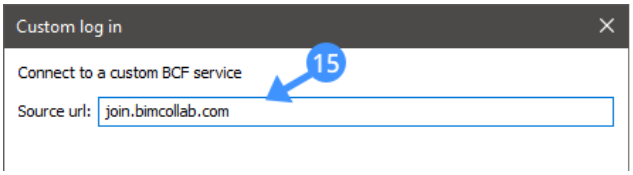


When using a cloud service, you are presented with some more options, such as setting statuses and adding comments and screenshots.

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

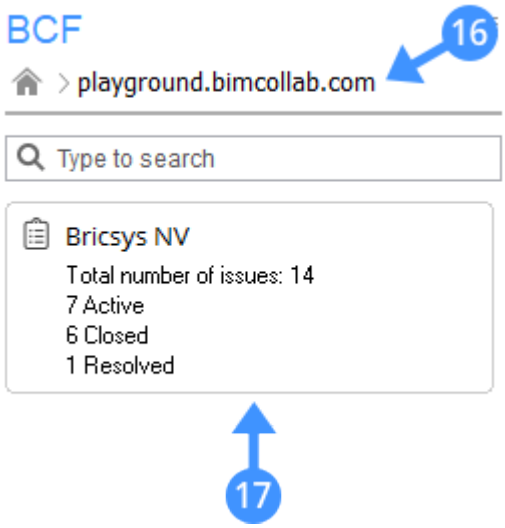
* Note that the Source URL for BIMcollab includes the name of the space you bought. If you are using a free account, use *join.bimcollab.com* instead.



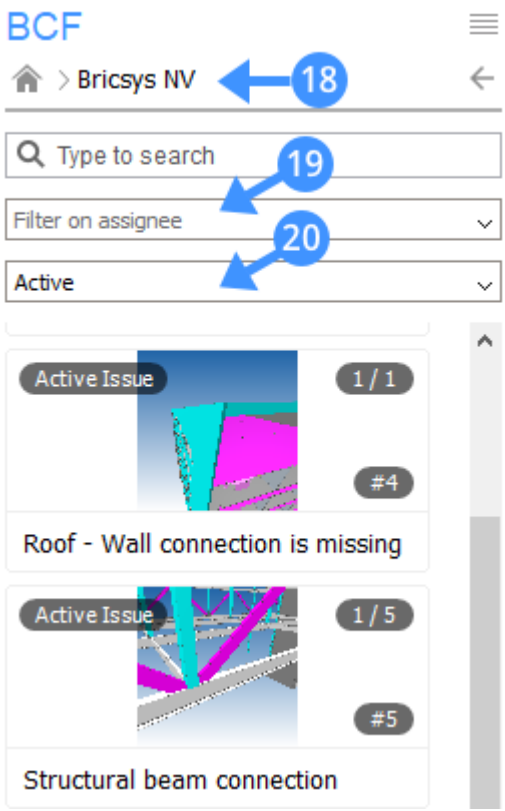
The following examples will be shown on an internal cloud space so it will not be possible to follow along with these steps exactly. It is possible however to create a free account on, for example, BIMcollab.

Clicking the **Log In** button should take you to a login page of the chosen service.

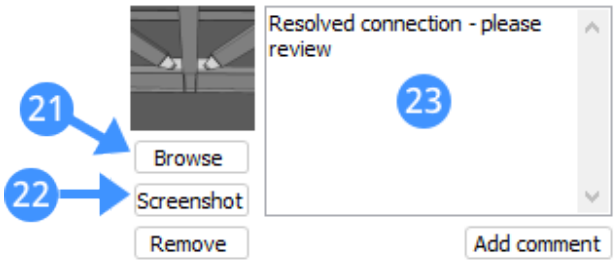
After logging in, the source URL should be shown (16), along with a list of projects that you are involved in (17). In this example, only one project is shown with 14 issues. Left-clicking the project will open it and show a list of the issues, similar to what we saw when importing the .bcf file.



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

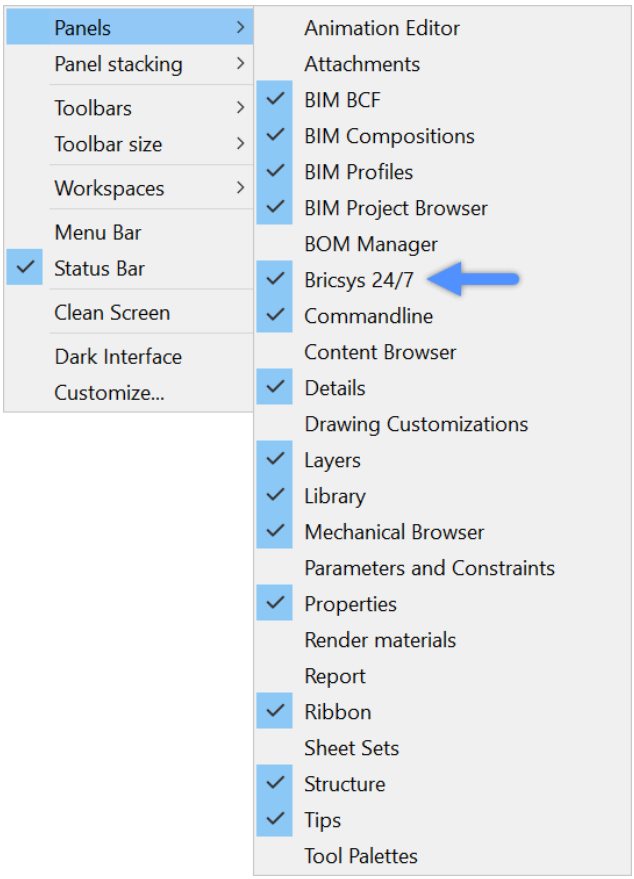
Bricsys 24/7 helps team members to find and share their drawings. Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

20.6.2.2 Use the Bricsys 24/7 panel in BricsCAD BIM

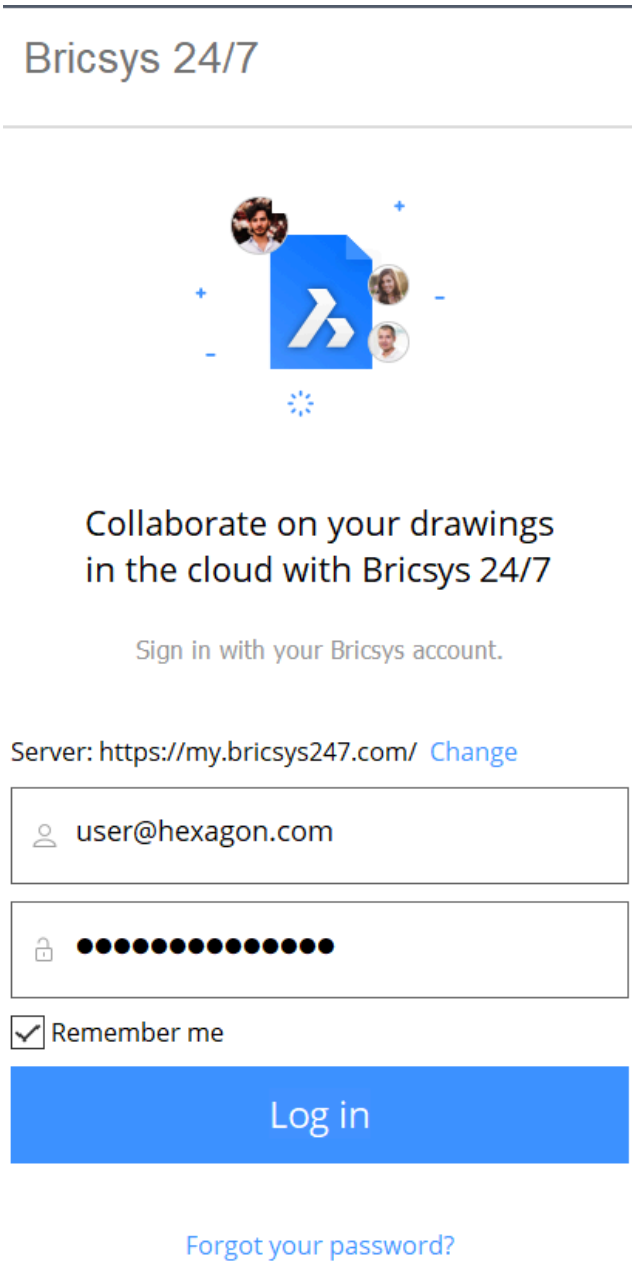
1 Open the **Bricsys 24/7** panel by clicking on the Bricsys 24/7 icon in the **Tool** panel on the right-hand side of the workspace.



Uwaga: 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.



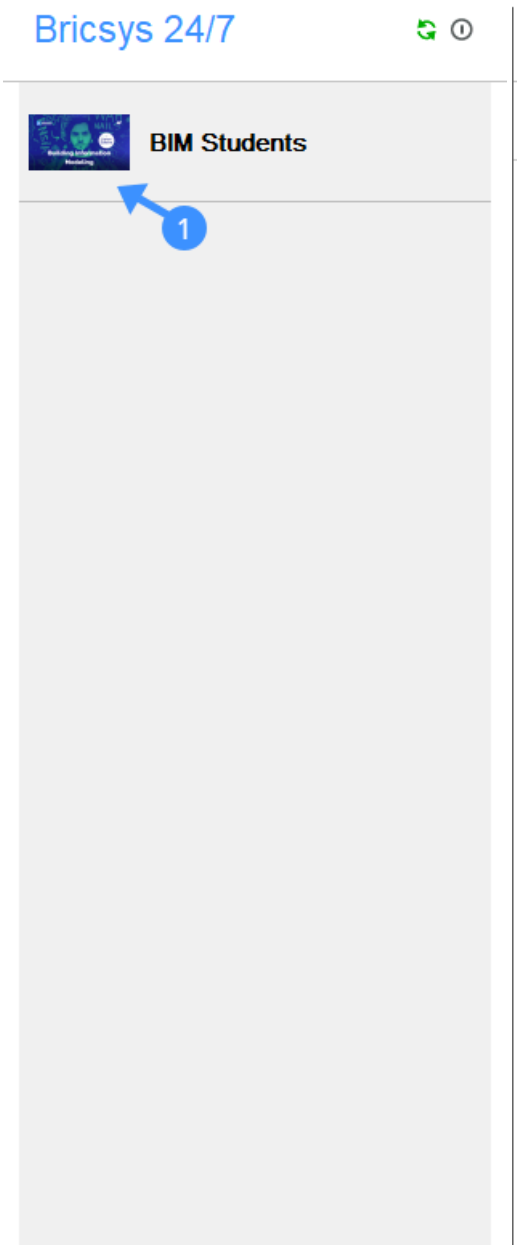
If you are logged out of your 24/7 account a screen will appear:

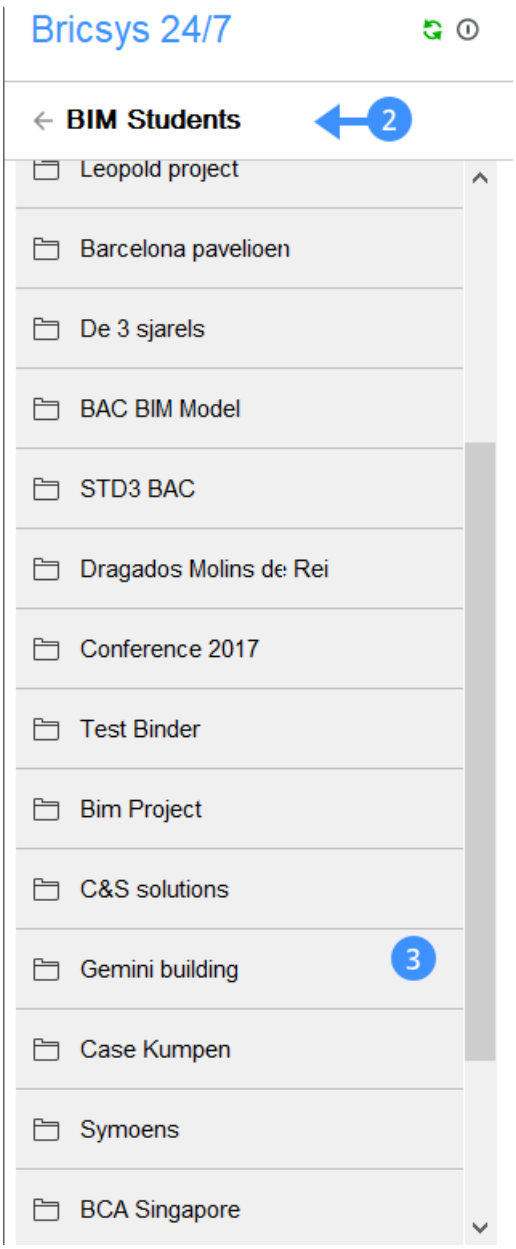


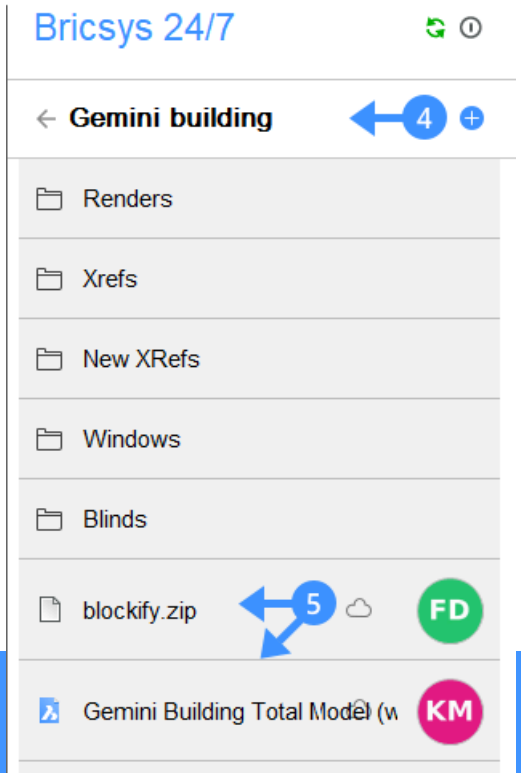
2 Enter your email address and password and press the **Log in** button.

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.

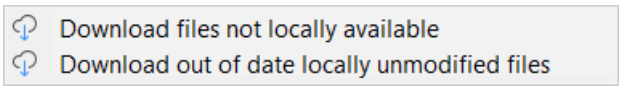

Bricsys 24/7
BIM Students
1


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

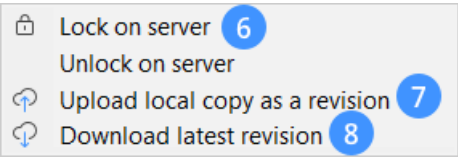
(1)	**project**	The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.
(2)		At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.
(3)	**folder**	When you open a project, a list of folders is shown.
(4)		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

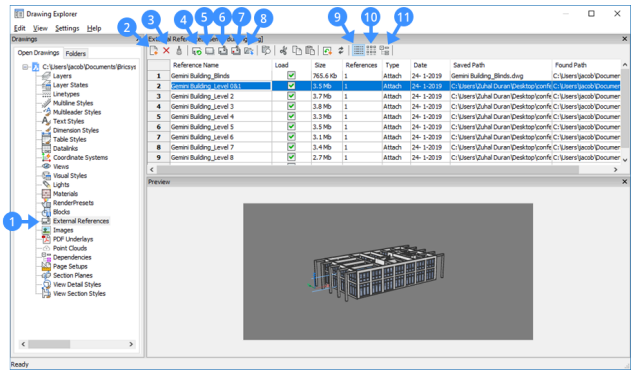
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. 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.

Uwaga: 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.

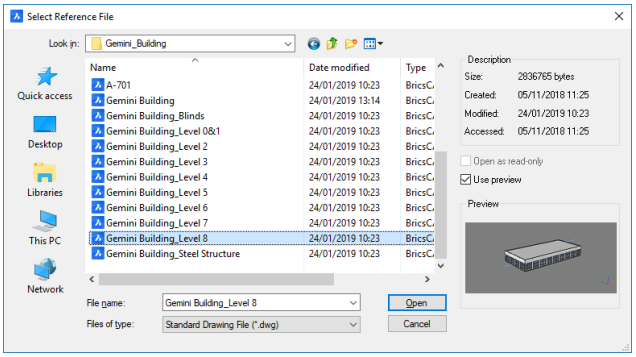


External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and 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.

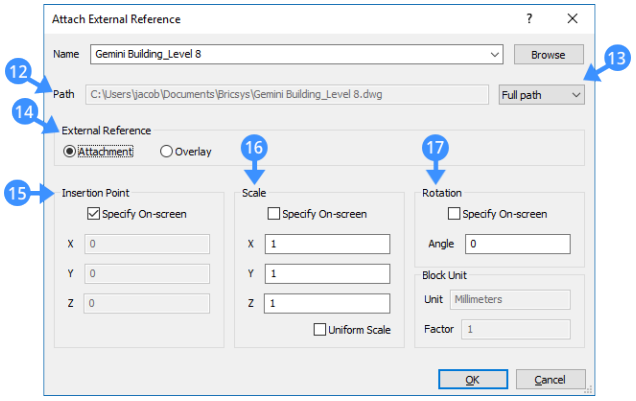
The **Select Reference File** dialog box displays.

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.

Uwaga: 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.

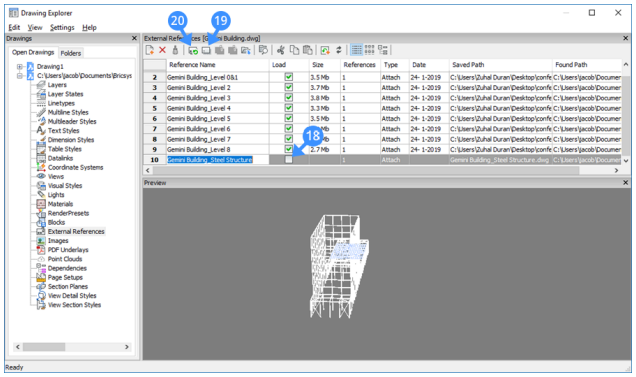
- Click the **Load** (18) column of a loaded XRef.

  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]:

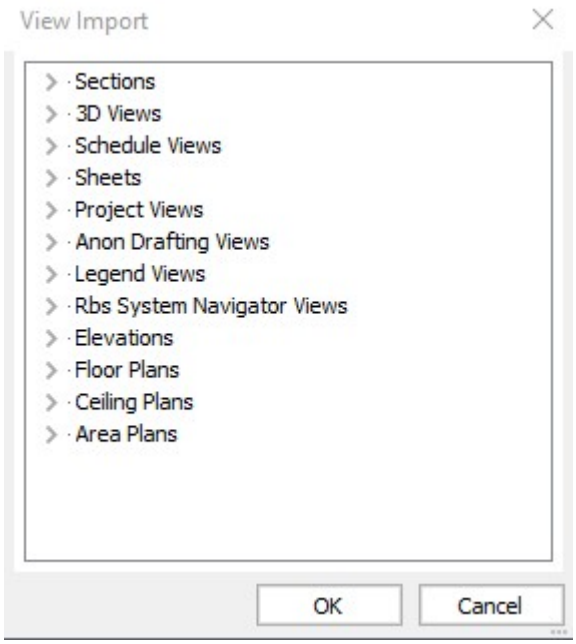
There are different filters to import a file. Select the filter you want to use by clicking into your desired option in the command-line or by hitting their respective letters in bold.

- **All**

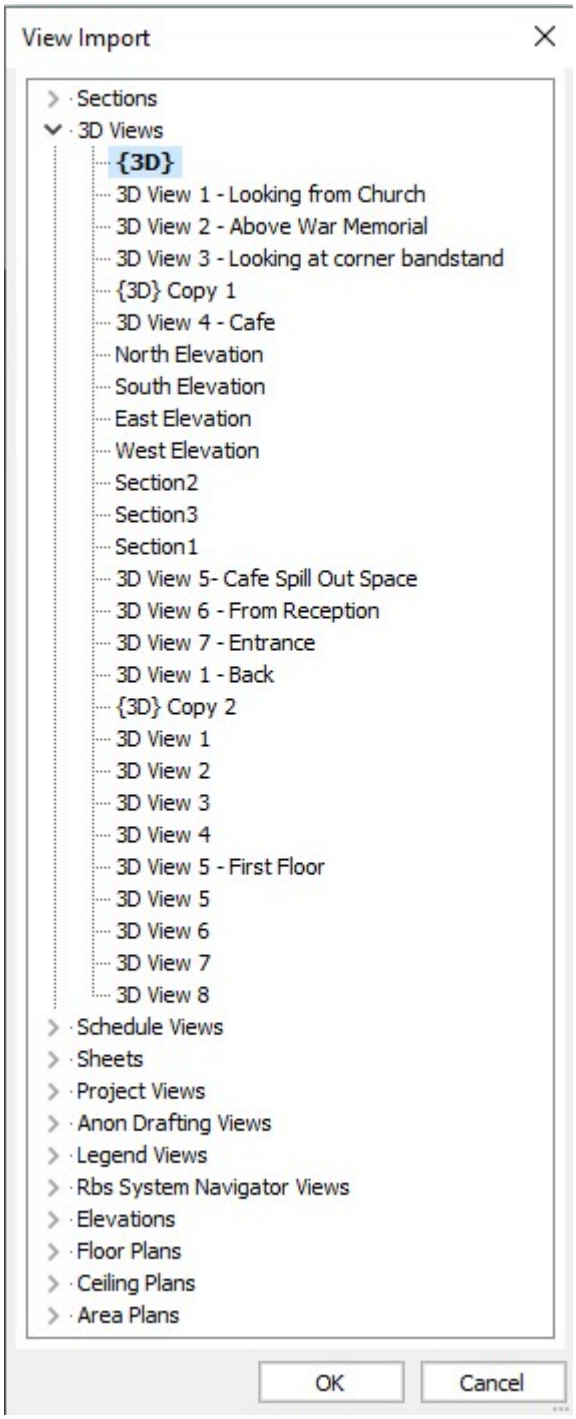
  All elements present within the RVT file will be imported.

- **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.



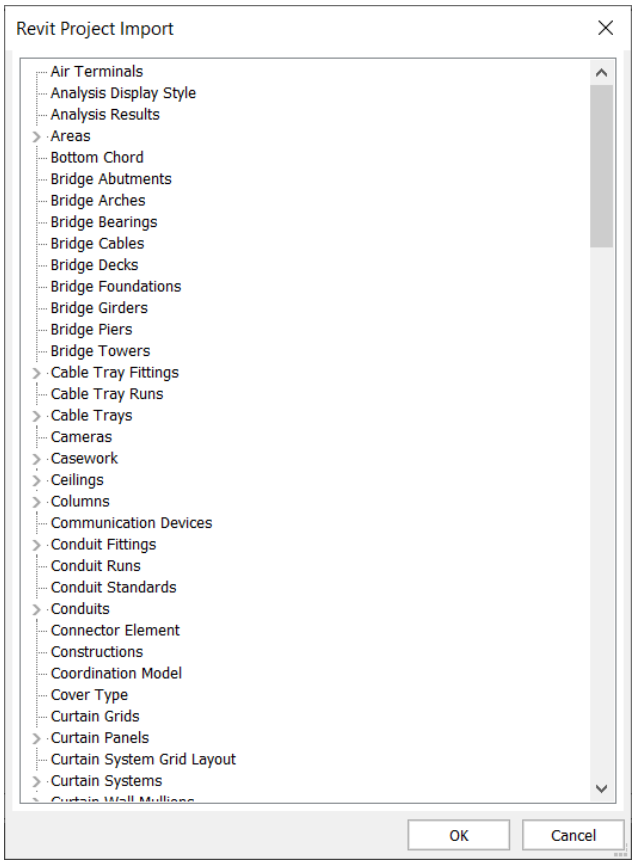
You can expand any relevant or applicable View Type category for the specific views. It could look like this, if the **3D Views** category is expanded:



Uwaga: 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.

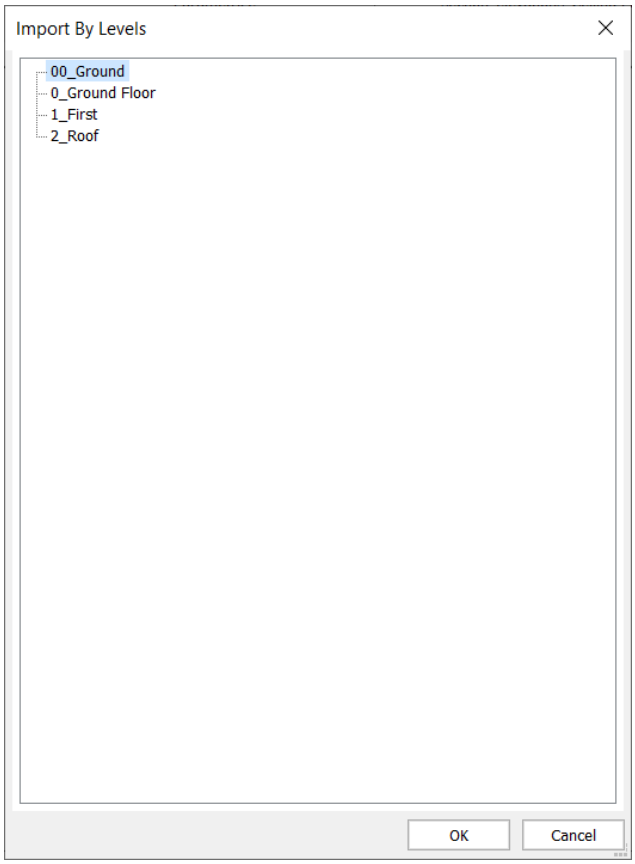
- **By Categories**

  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.

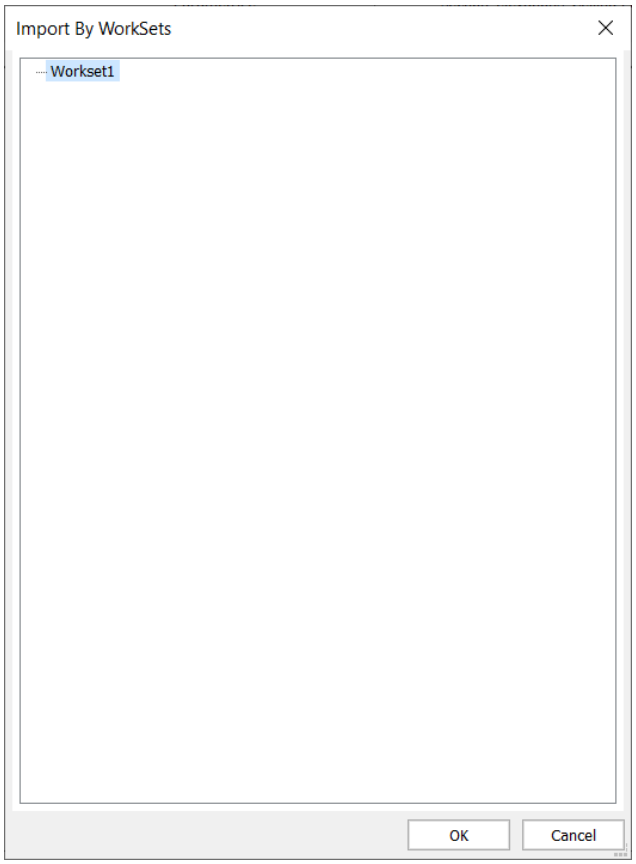


- **By WorkSets**

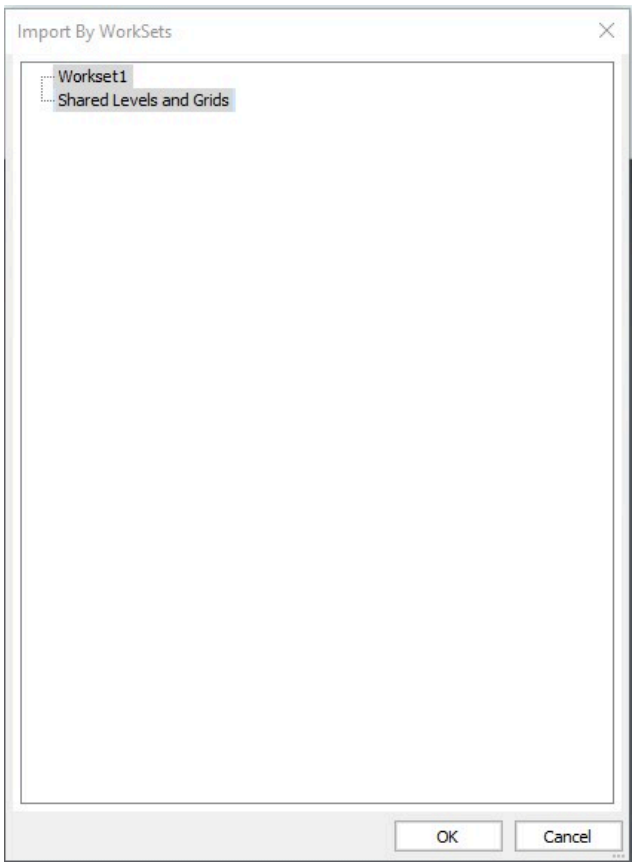
  You can select one or multiple worksets from the **Import By Worksets** dialog box.

  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.



Uwaga: 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.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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

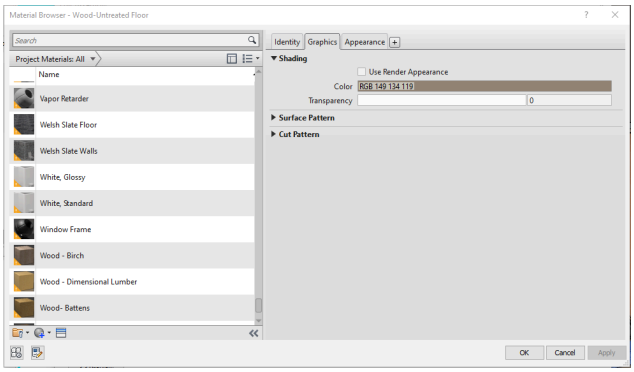
Please refer to the article **Importing Revit RFA Families in BricsCAD** for more information.

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.

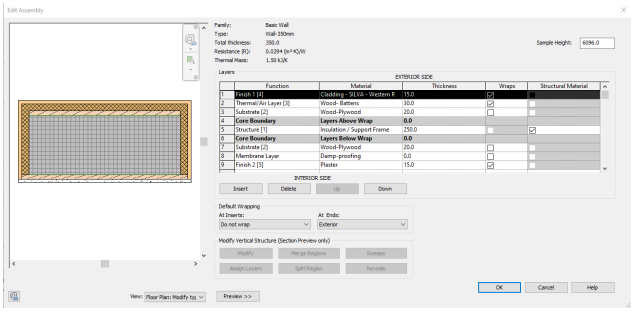
To retain the consistency of the graphical information during the import process, ensure your RGB color and transparency settings set in Revit's Material Browser are set to your desired values.

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.

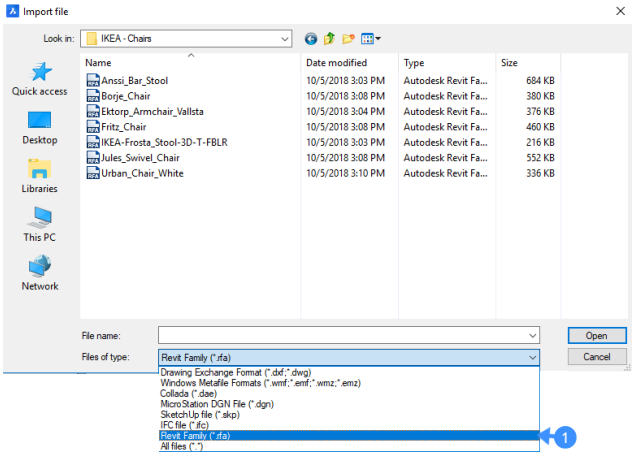
BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the **IFC import and export in BricsCAD** topic.

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

Uwaga:

- 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.

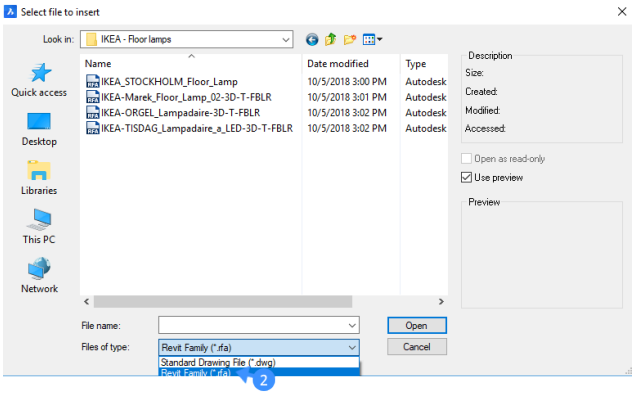
Using BmInsert

1 Launch the BMINSERT command.

1 A **File** dialog box appears.

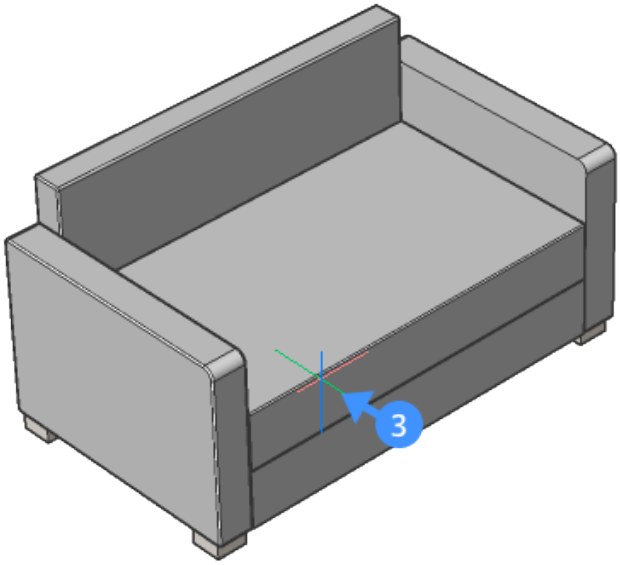
2 Browse to the folder where your .rfa family files are saved.

3 Select **Revit Family (*.rfa file)** (2) in the **Files of type** list.

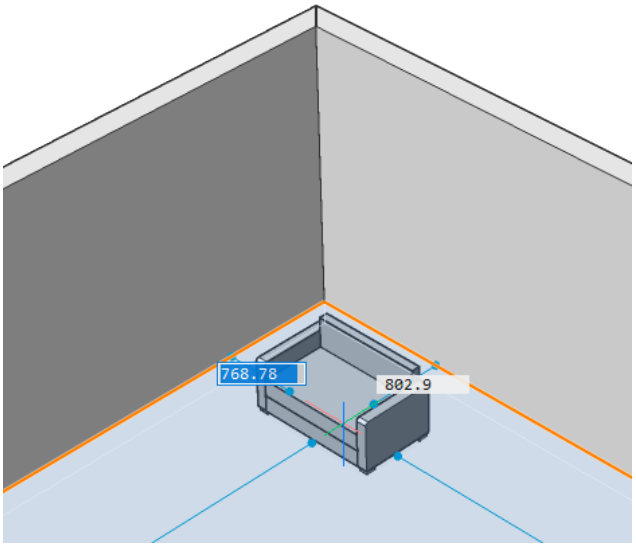


4 Select the .rfa file you want to import.

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.



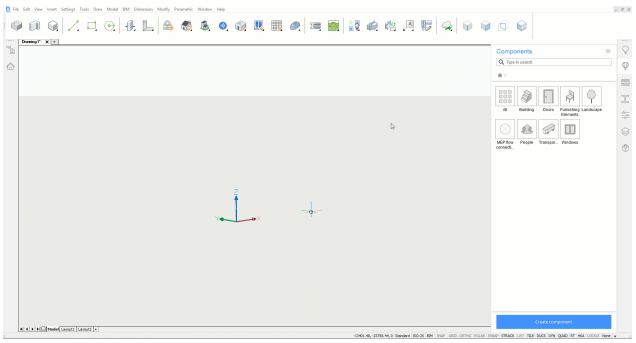
6 Do the following to place the selected RFA family file using the dynamic dimension fields:

- 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.

Uwaga: 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.

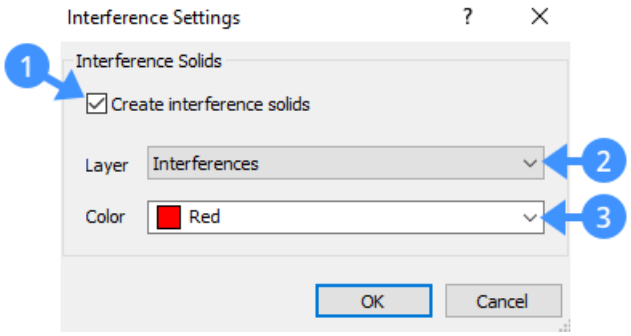
Uwaga: If you press the **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

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.

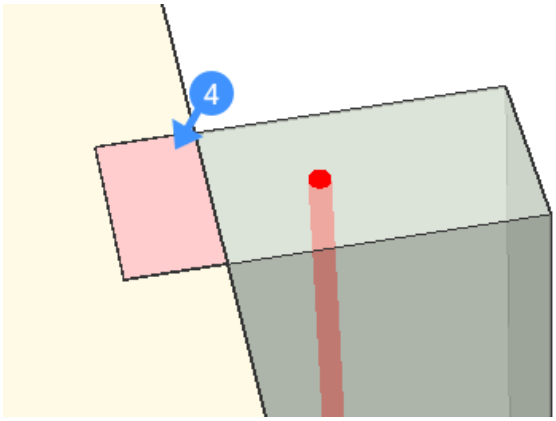
Uwaga: The solids of the interference volume are displayed in red since they are placed on layer **Interferences**. The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the Command line. 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.



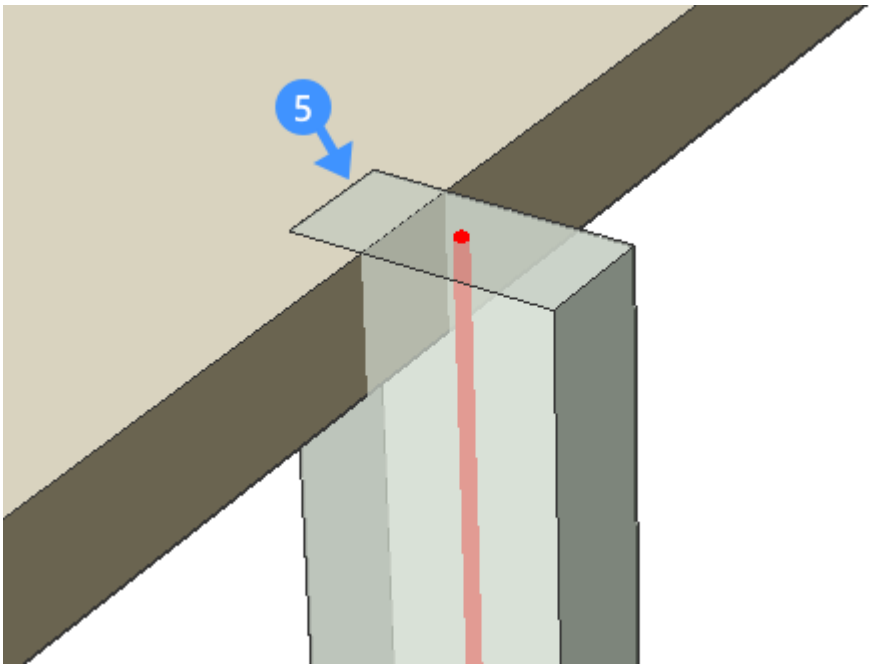
Create interference solids box (1)	Determines whether a solid or region of the interference is created. Check to create a solid of the interference volume or region of the interference area.
Layer (2)	Shows the current layer for interferences.
Color (3)	Shows the current layer color.

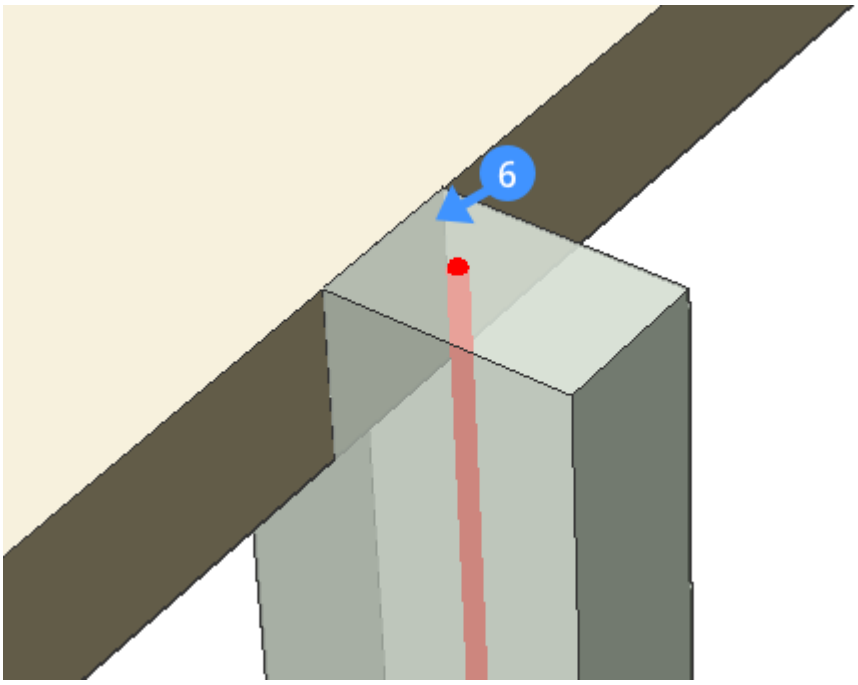
The following illustration shows the detected collision (4) between the floor slab and column.



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

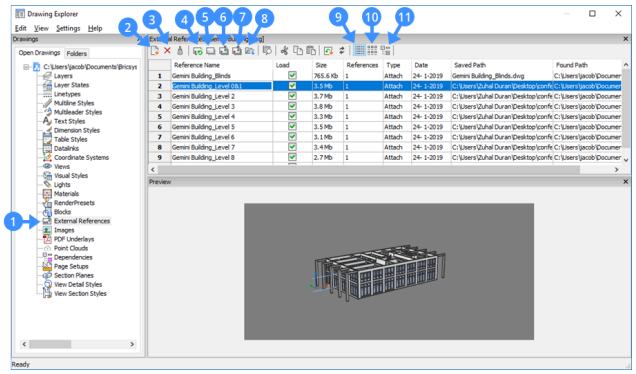
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. 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.

Uwaga: 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.



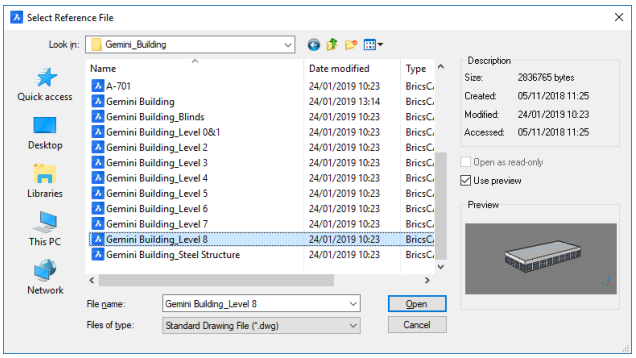
External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and 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.

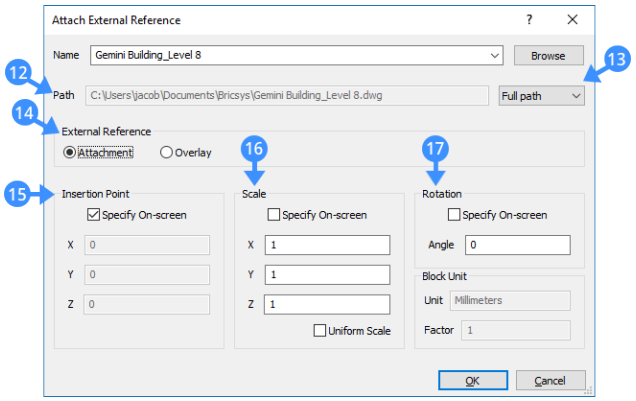
The **Select Reference File** dialog box displays.

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.

Uwaga: Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

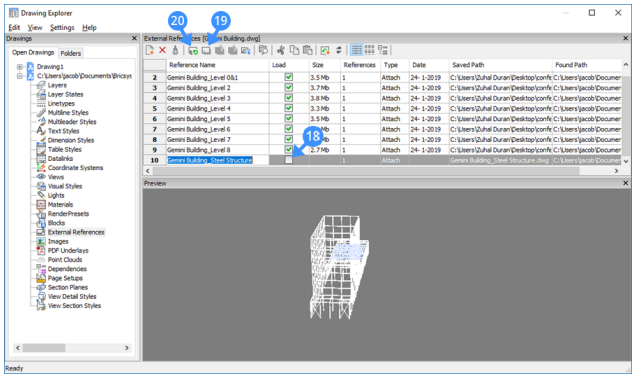
4 Do one of the following to unload and reload an existing XRef.

- 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 Design documentation

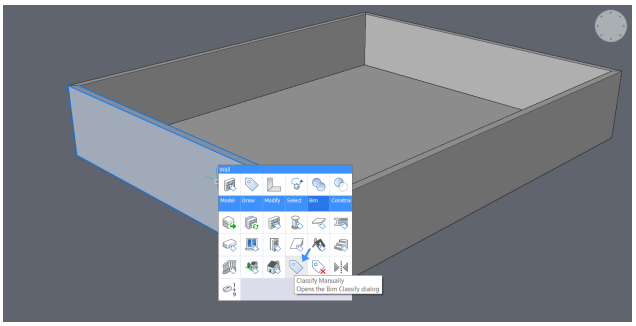
20.7.1 Annotations in 3D

20.7.1.1 Commands

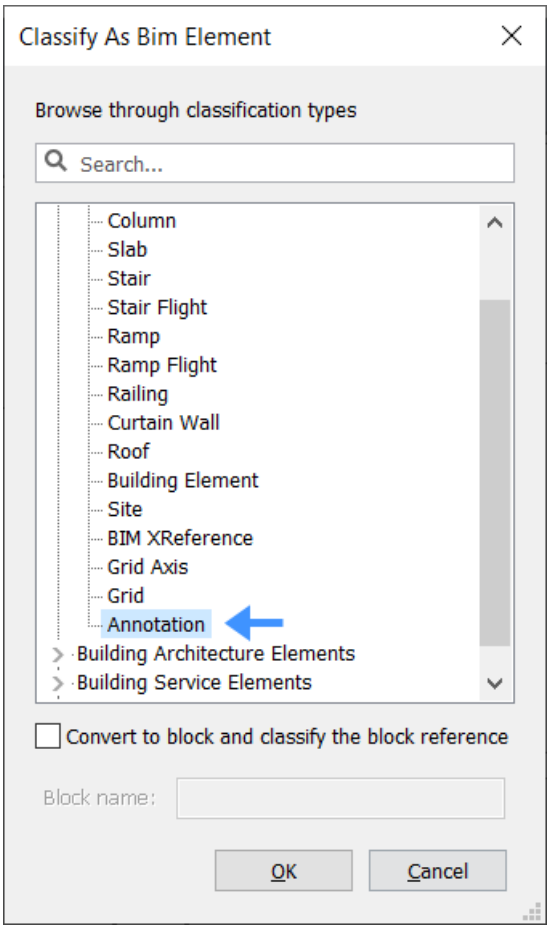
BIMTAG, BIMCLASSIFY

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:

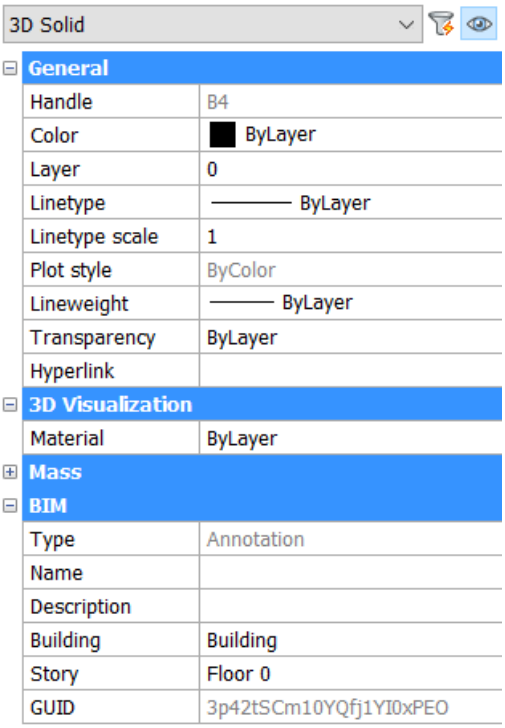


2 Choose to classify as **Annotation** in the folder of **Building Core Elements** and click **OK**:



3 When you open the Properties panel of the selected solid you can change the **Name**, **Description**, **Building** and **Story**.

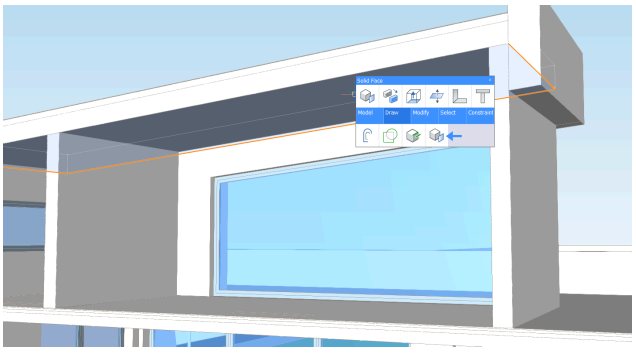
Uwaga: The property GUID cannot be changed because it has a unique, automatically generated string.



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):

Uwaga: By default, the BIMTag references the **Description** property.

Region	
General	
Handle	23E2
Color	White
Layer	1
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
3D Visualization	
Material	ByLayer
Geometry	
Perimeter	49.21292 m
Area	147.38 m²
UCS elevation	550 cm, 657.732 cm
Minimum	550 cm
Maximum	657.732 cm
Slope	174
BIM	
Type	Annotation
Name	
Description	Pigmented Concrete in wooc
Building	
Story	
GUID	3DkiSm2SvB$BySdAYHQqHw

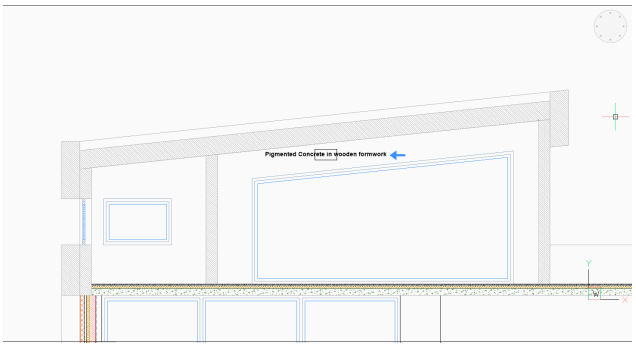
4 Make a section through the region (with BIMSECTION command) and select **Generate Section**( ) from the Quad:



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

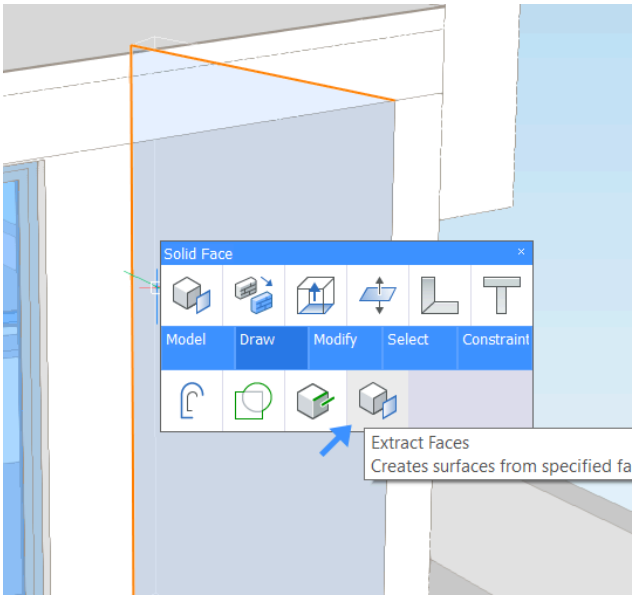
This procedure will teach you how to display other properties of an entity classified as an **Annotation** object.

Uwaga: 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.

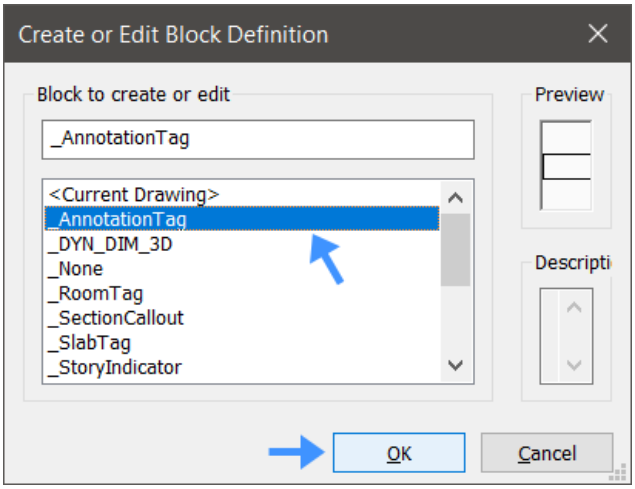
5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

Uwaga: The tag appears empty because it refers to the **Description** property by default.

7 Launch the BEDIT command.

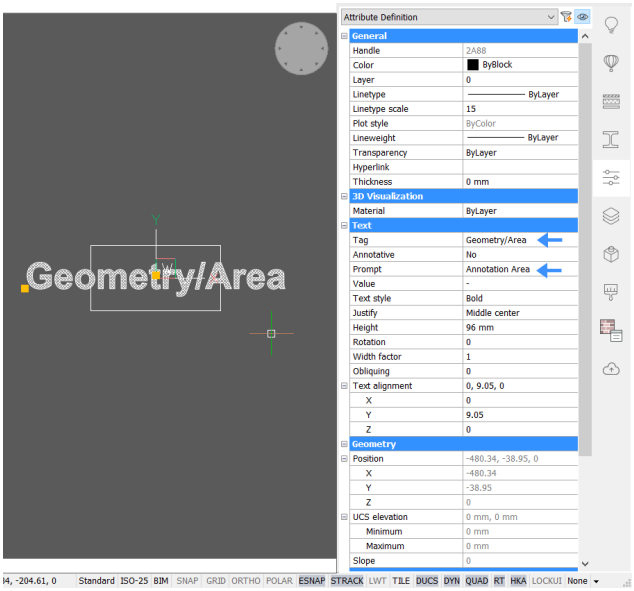
8 Select **_AnnotationTag** and click **OK**.



9 Select the attribute definition, in the properties panel.

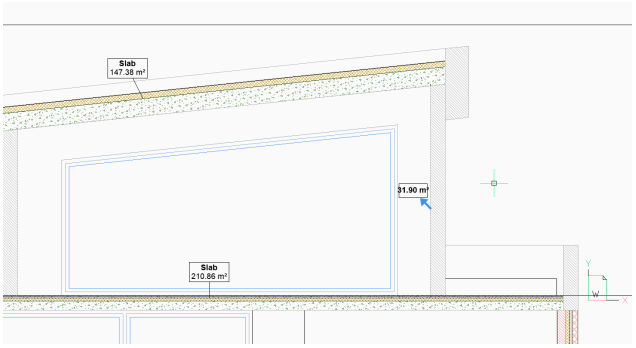
10 Edit the **Tag** property, change it to **Geometry/Area**.

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

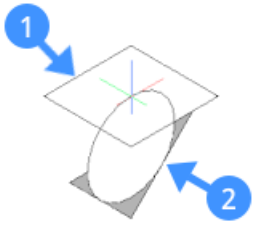
20.7.2.2 Defining a section

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.

Uwaga:

- 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**.

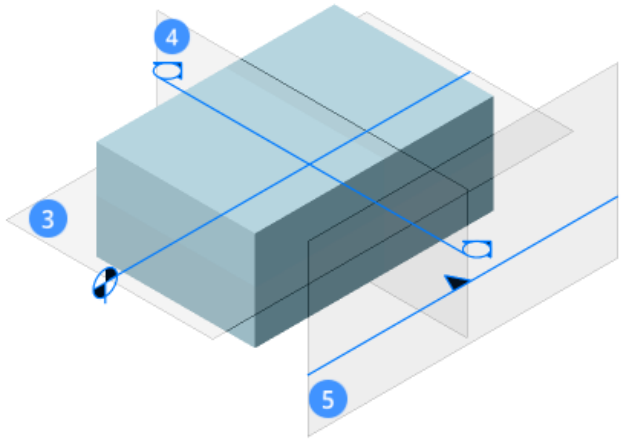
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

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.



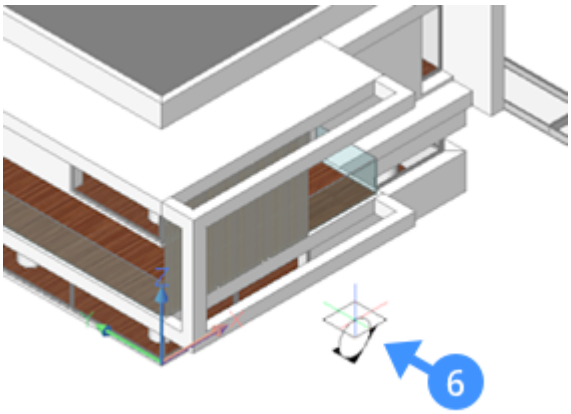
Plan (3)	Displays horizontal section plane.
Section (4)	Displays a vertical section that cuts through the model.
Elevation (5)	Displays the exterior elevation view.

Creating a plan section

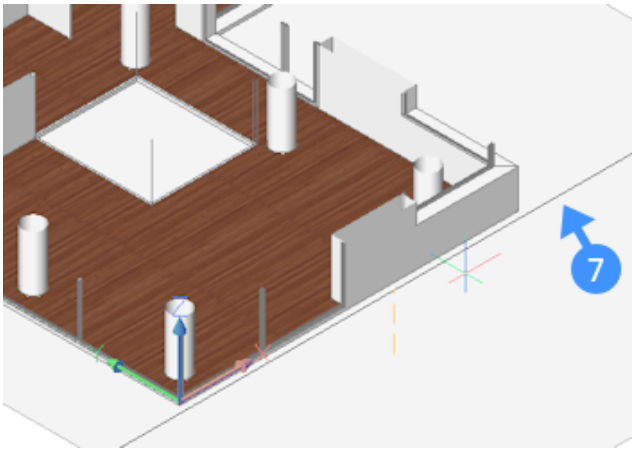
Uwaga: A plan section is best defined in the WCS (World Coordinate System).

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.

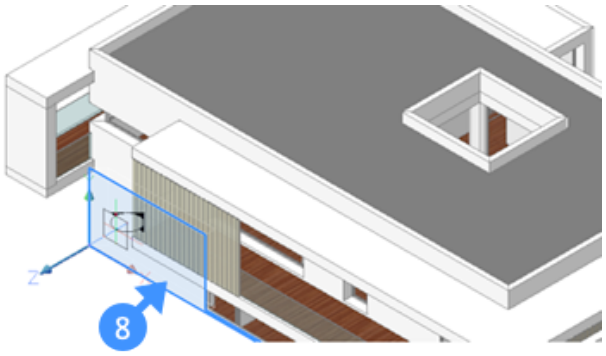
Creating a cross section

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

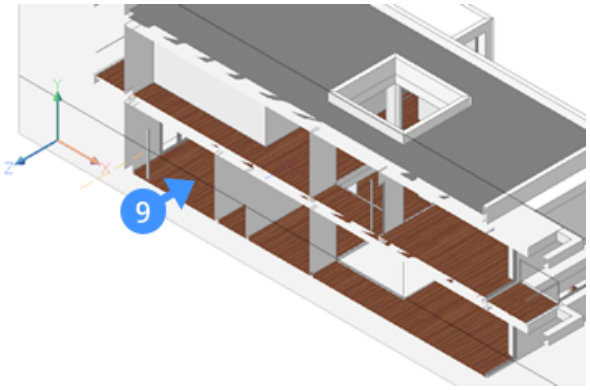
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

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.

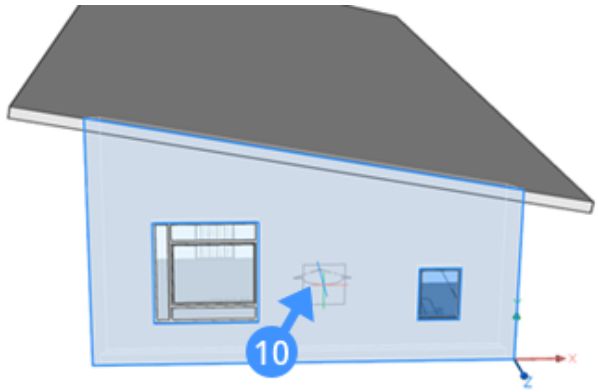
Creating an elevation

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

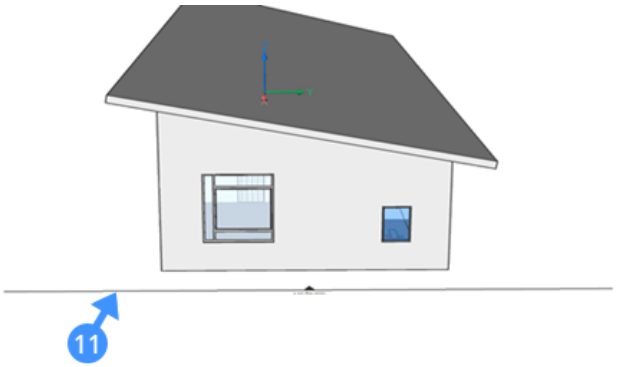
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

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:

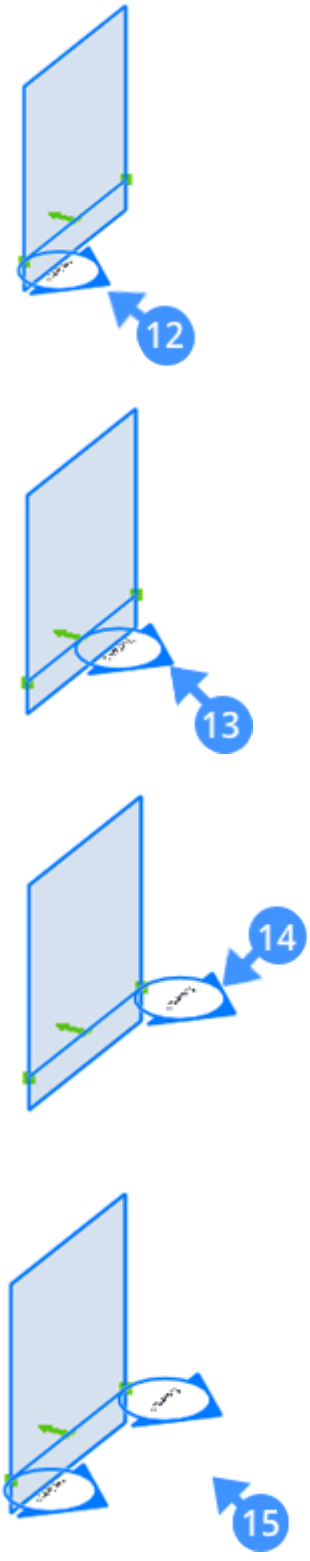
Name	Type a name.
State	Click the field and select a state. The selected state determines how the section result will be clipped. The options are: **Plane** (default), **Boundary** and **Volume**.
Live section	Sets the live section property of the section plane. Only one section entity can have the **Live 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.

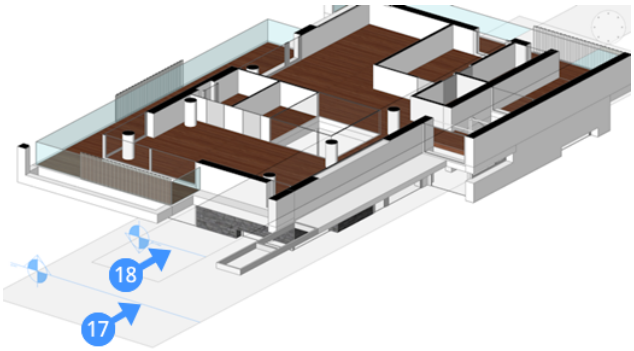


To clip the 3D model, do one of the following:

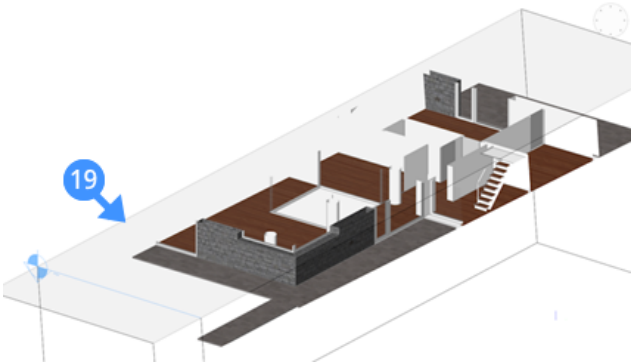
- Double click the section entity.
- Select **Clip display** from the Quad.
- Select the section plane and set the **Clip display** property to **Yes** in the properties panel.
- When **RT** (Rollover Tips) is selected in the **Status** Bar, place the cursor over a section entity to set the **Clip display** property.

Uwaga: **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

Uwaga: 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.



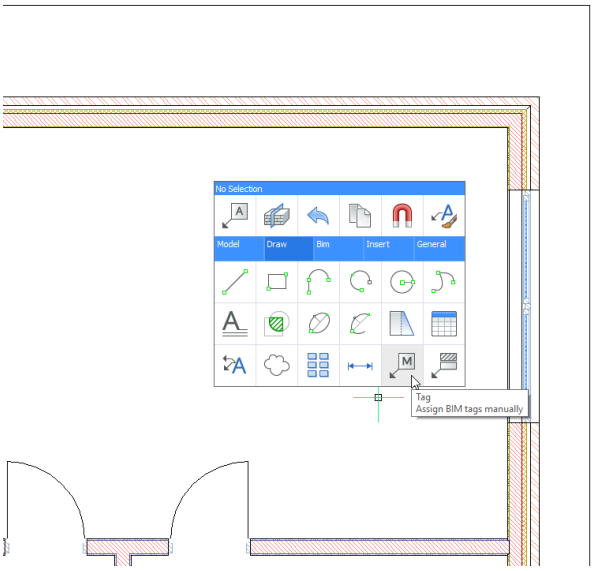
Manually create a BIM tag

1 Open the section drawing for which you wish to create BIM tags.

Uwaga: For more information about generating section drawings, please read the BIM article **Generating Drawings** first.

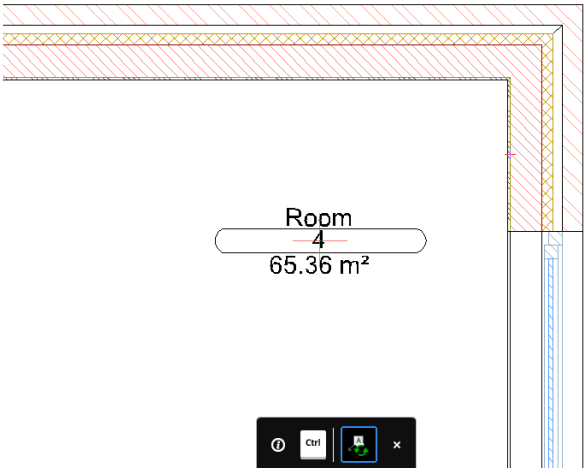
2 With no selection, select **Tag Assign BIM tags** manually from the Quad or enter BIMTAG followed by M

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.



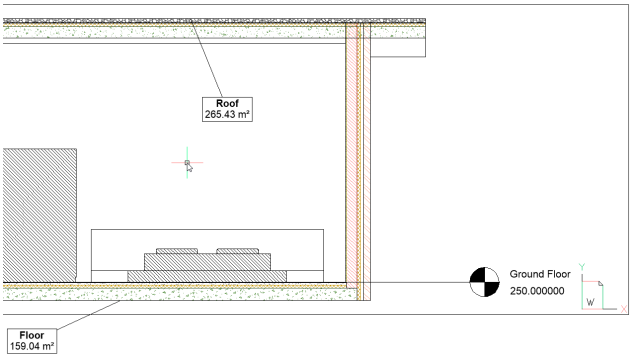
You are prompted: Place the tag [Next element]:

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

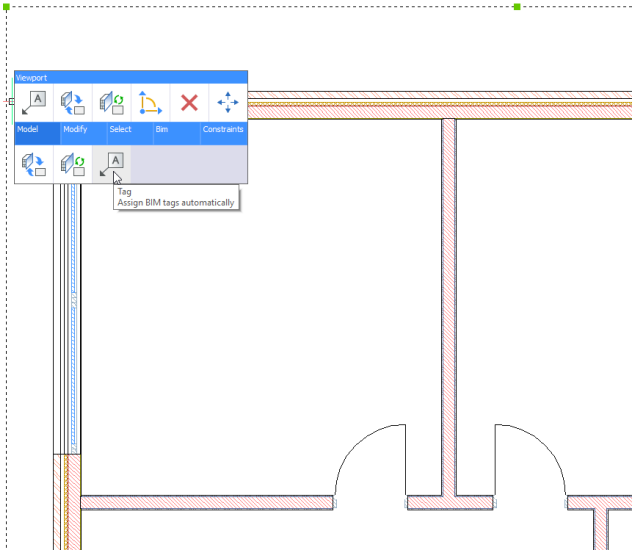
If you want all BIM elements in a section drawing to be tagged it is easiest to use the BIMTAG Select all entities command. This command automatically assigns BIM tags to all BIM entities inside a viewport. If the model is altered or new elements are added, the BIM tags can be updated and new tags added.

For more information about this command, visit the Command Reference article BIMTAG.

Automatically create BIM tags

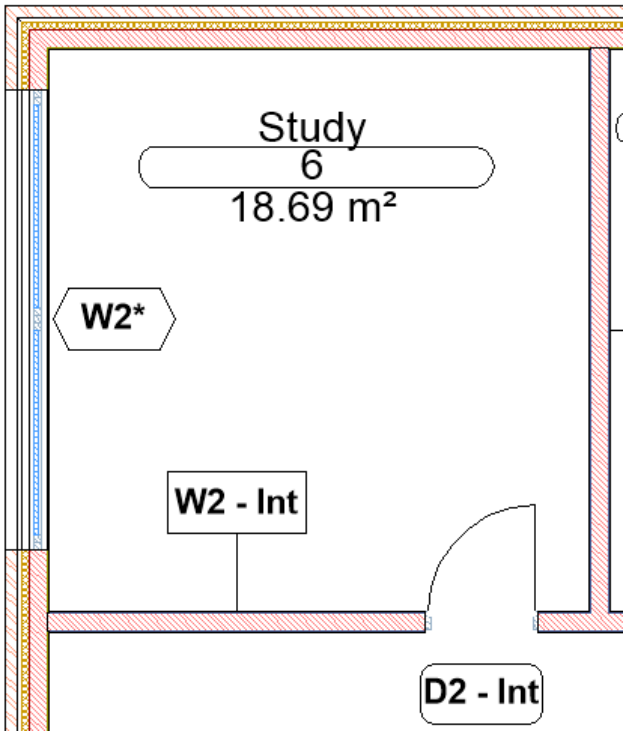
1 Open the section drawing on which you wish to create BIM tags.

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.



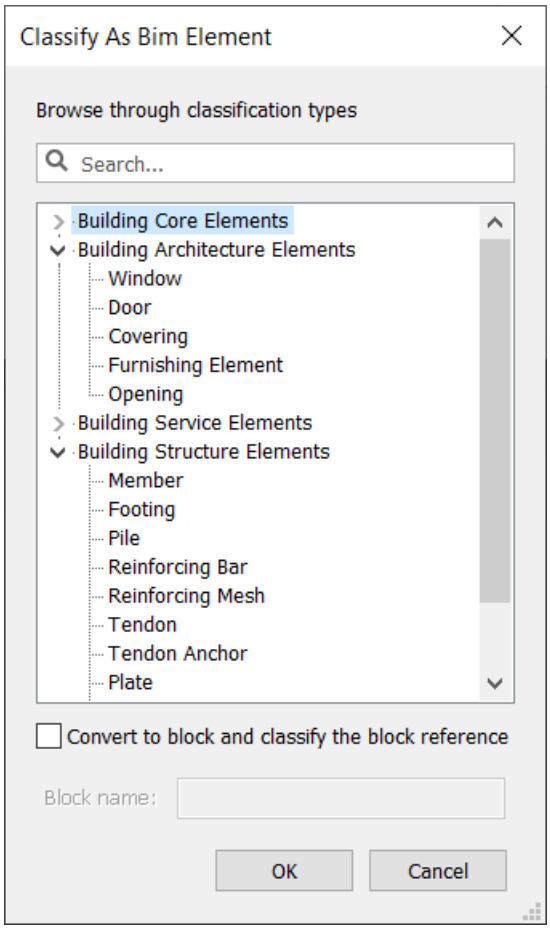
You are prompted: Select BIM section viewport [Manual]:

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.

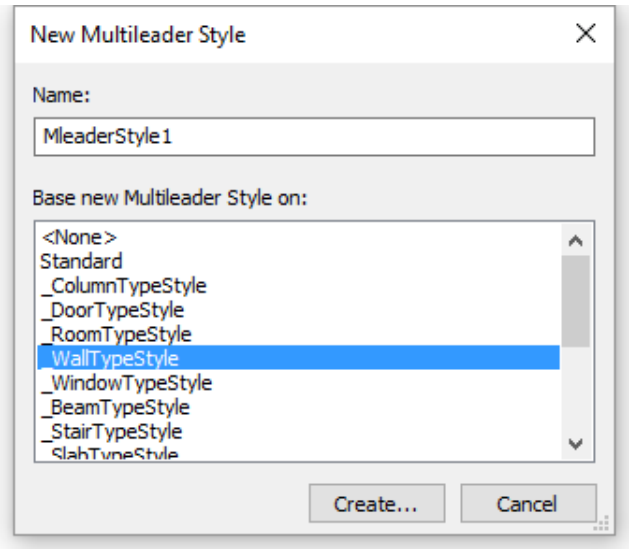
8 Click **Copy**, but keep both on the **Copy/Paste** dialog box.

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

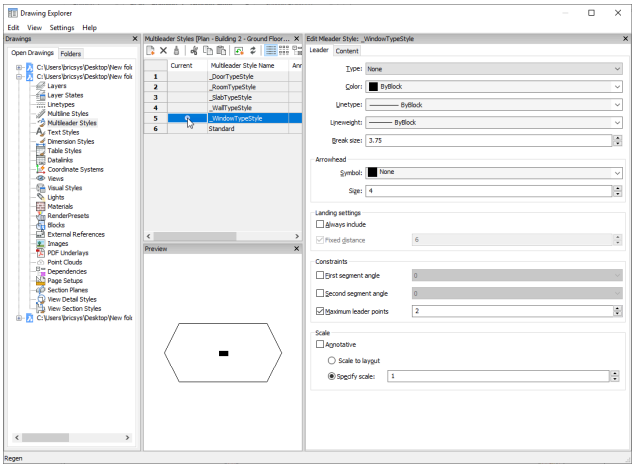
The content of a Bim Tag block consists of:

- Layout entities Linework, fixed texts, etc.
- Attributes Bim entity properties.

Uwaga: 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

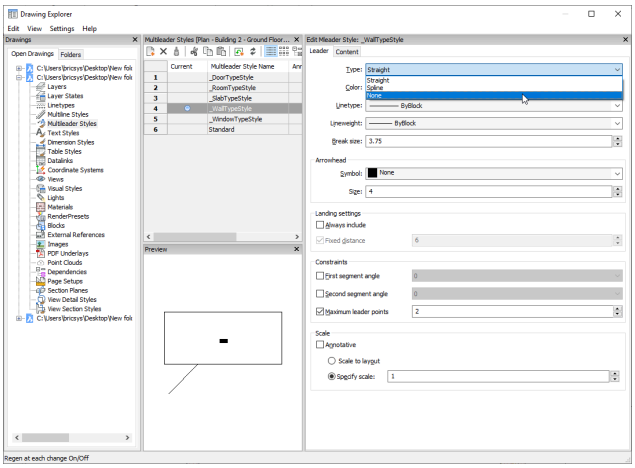
1 In the section drawing for which you wish to customize the BIM Tags, type MLEADERSTYLE in the Command line. The drawing explorer will open in the **Multileader Styles** tab.



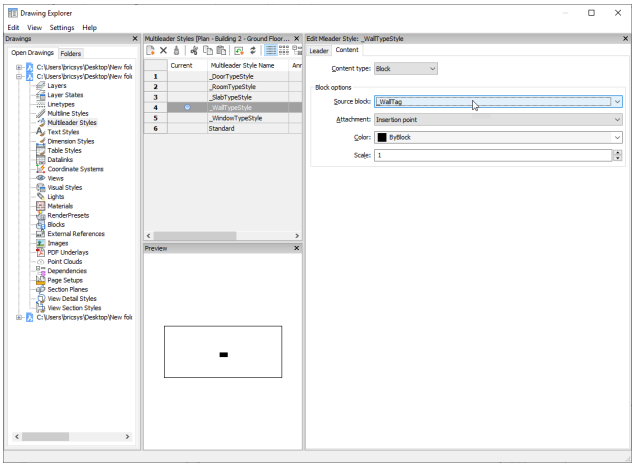
Each BIM tag is a multileader. The multileader style depends on the type of BIM element being tagged. All the multileader styles in the drawing are listed in the drawing explorer.

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.



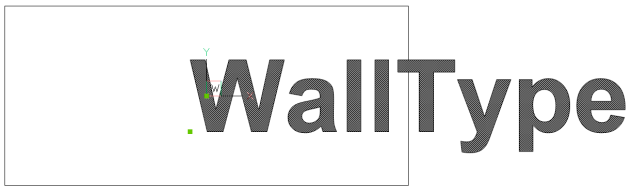
4 In the **Content** tab, the **Source block** is called **_WallTag**. Remember this and now close the drawing explorer.

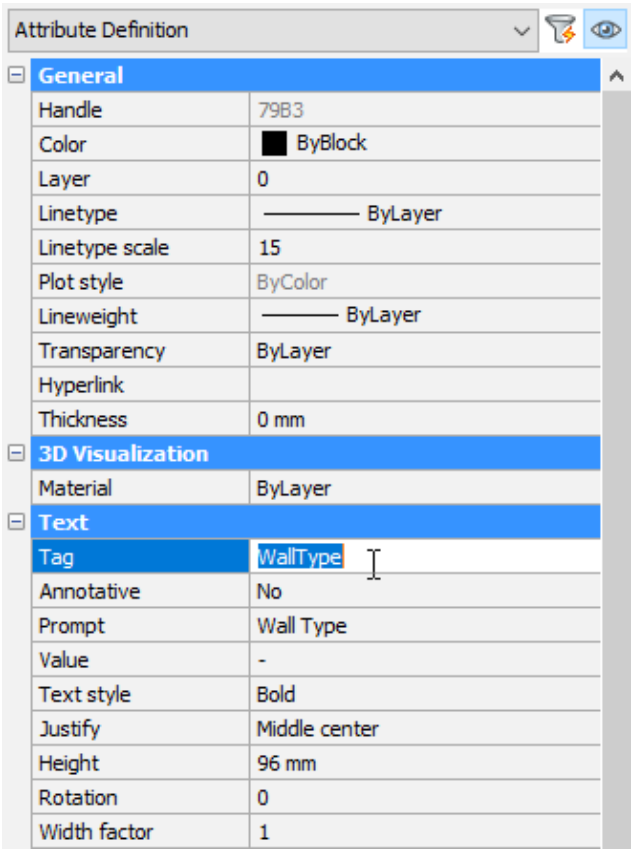


5 In the Command line type BEDIT and select **_WallTag** from the list. Click **OK**.

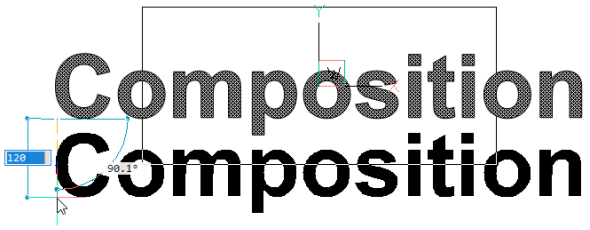
6 Select the **WallType** text and **Tag** value of the properties panel.

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.



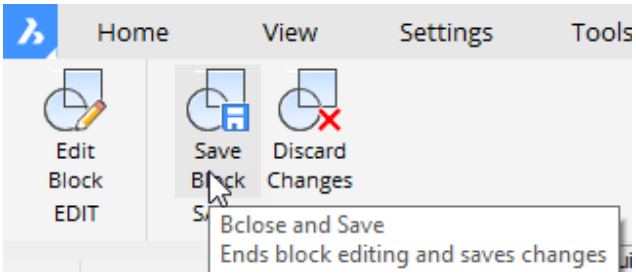
9 Repeat steps 5 - 7 to change the new attribute **Tag** value (**Quantity/Thickness**).

10 Delete the rectangle.

Composition Quantity/Thickness

11 Select the **Save Block** icon from the **Block Editor** tab of the ribbon.

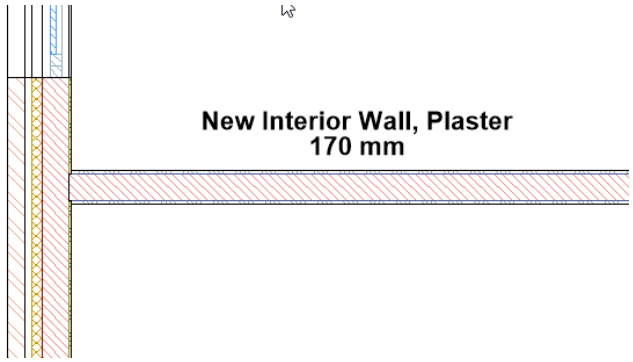
Your section viewport will now display.



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

Uwaga: 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

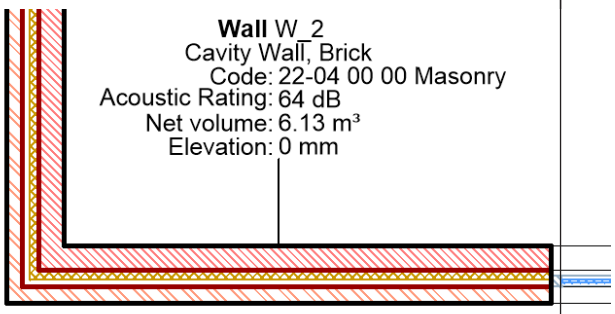
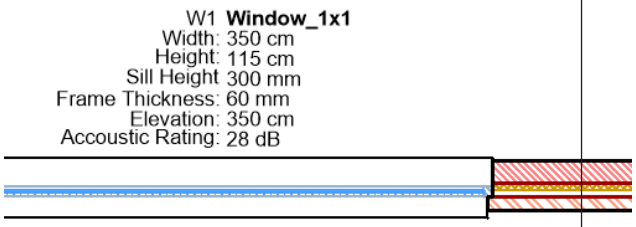
6 For each new attribute do the following:

- Copy an existing attribute or create a new attribute (ATTDEF command).
- Select the new attribute.

  Uwaga: 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.

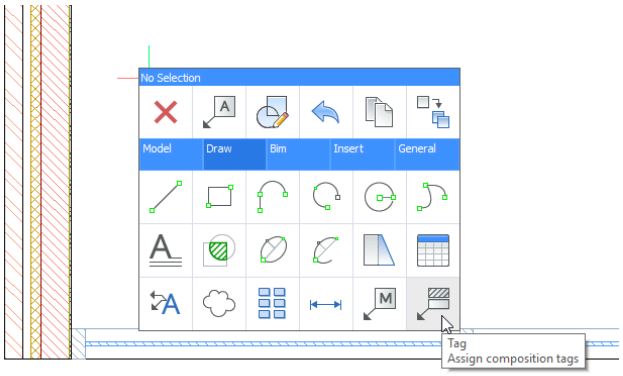
For more information about this command, visit the Command Reference article BIMTAG.



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.

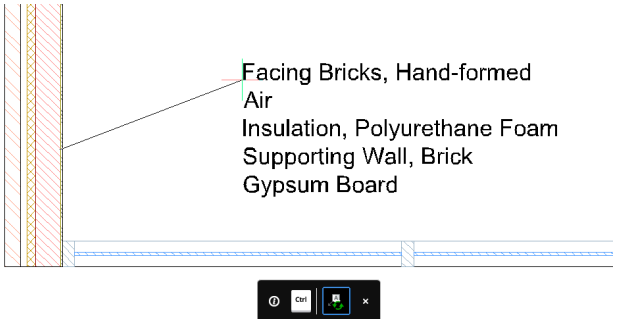


Or type BIMTAG in the Command line, then do one of the following:

- 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.

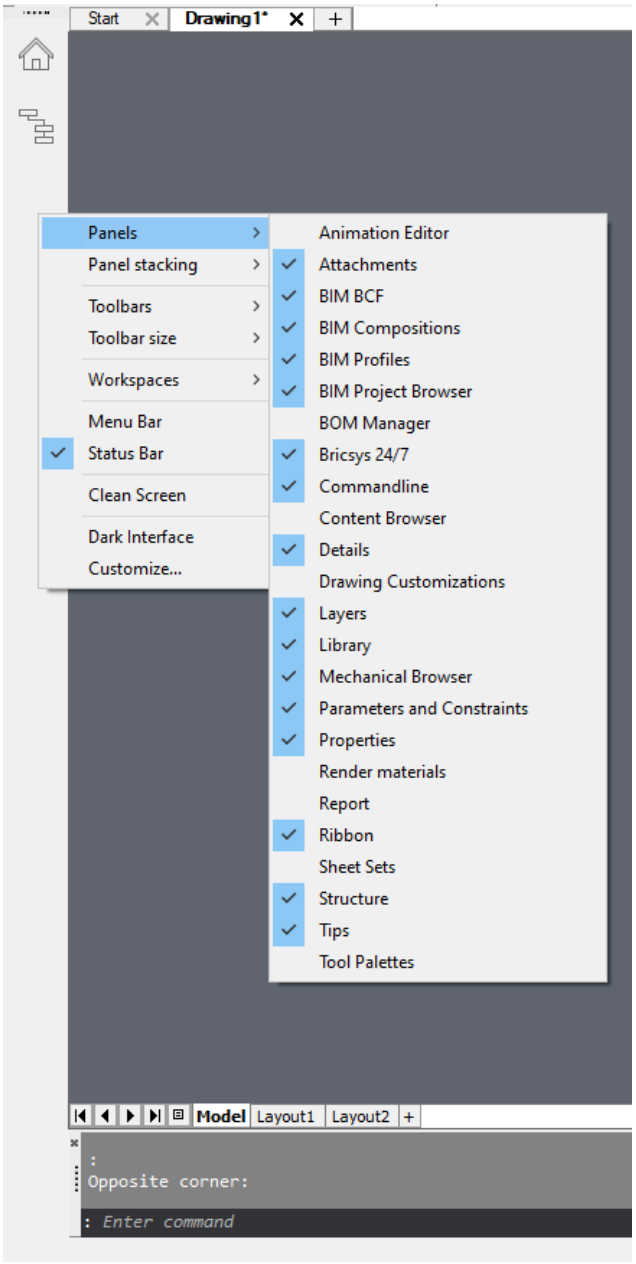
In V21, **Drawing Customization** allows you to create bespoke templates for drawing types based on a set filter parameter rule against an entity type's value or property within the 3D BIM model. This is especially helpful for customizing graphics of model entities independent of their assigned layers 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.

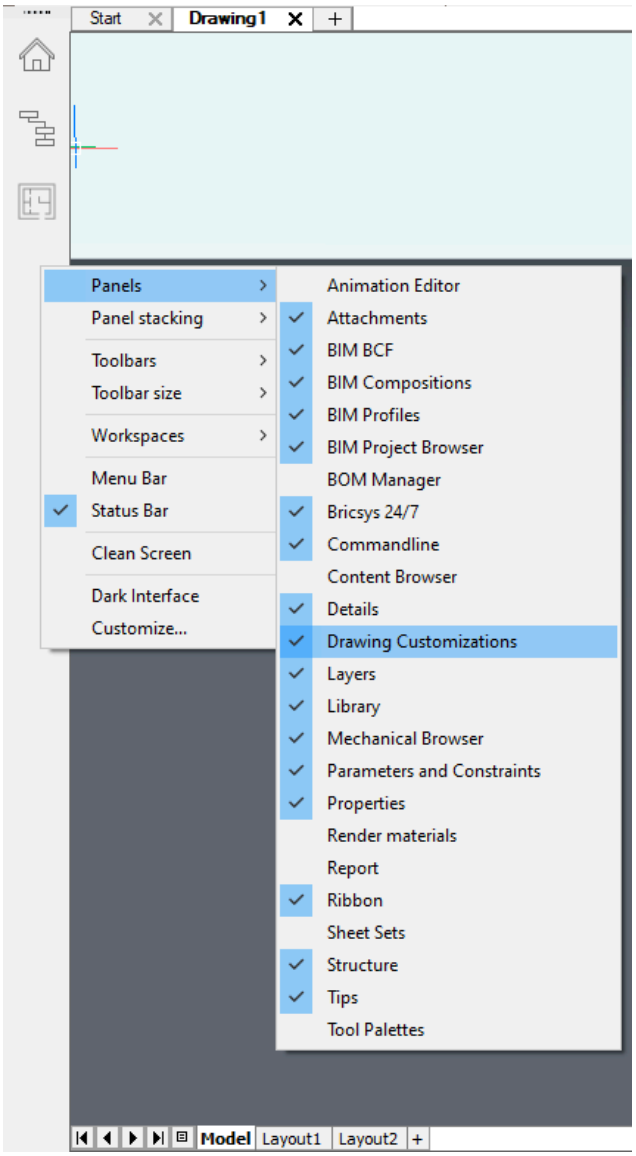
> ***Uwaga:*** Version Control for **Drawing Customization** will be implemented from V21.1.06. As features and parameters are constantly improved and added in future V21 updates, using older versions of BricsCAD to open **Drawing Customization** templates may affect their behavior and performance. Therefore, 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

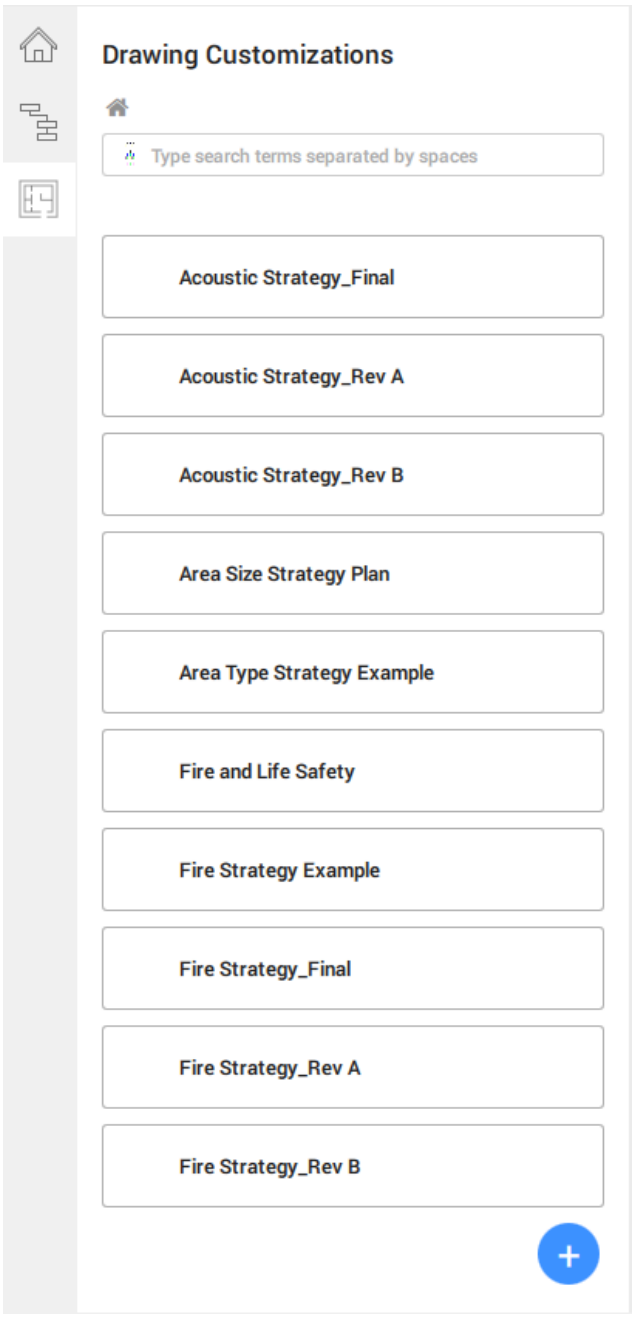
1 Right-click the panel and hover to **Panels**.



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.



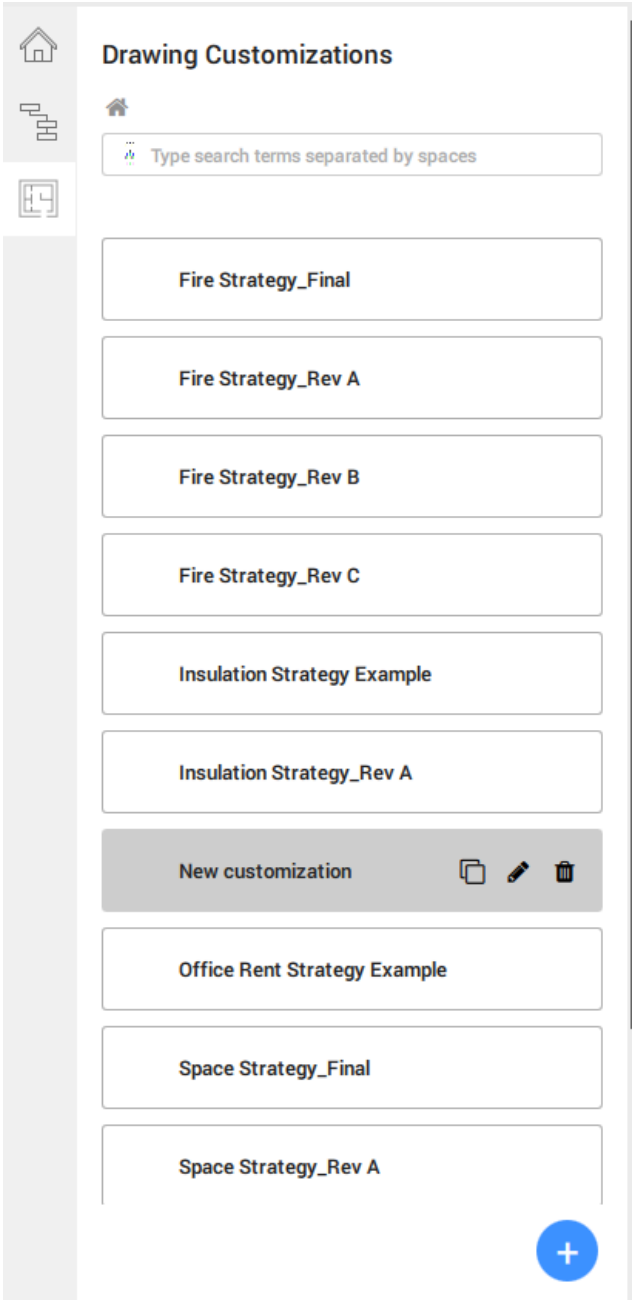
4 Click to create a new template. By default, the template is named **New Customization** with the text highlighted.



Uwaga: 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.

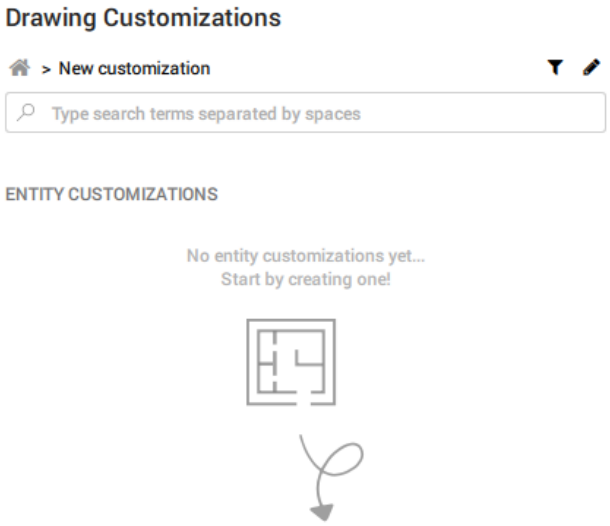


Uwaga: You can duplicate, rename or delete your customization templates.

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.

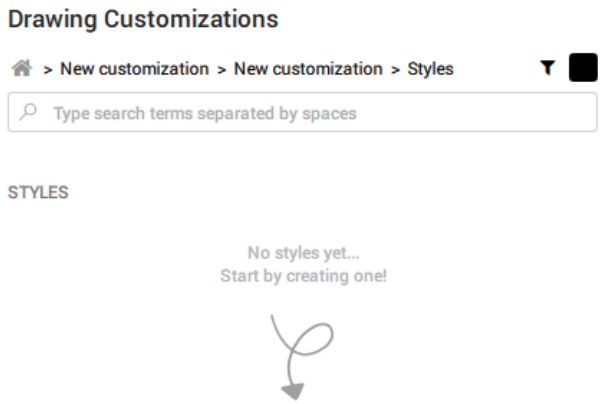


You will also notice a funnel icon and a pencil icon on the top right part of the panel. They are the **Filter Rules** and **Styles** tabs respectively.
The 3 principles – **Entity Customizations, Filter Rules & Styles** – which make up the **Drawing Customization** tool as described earlier in the article, are now accessible on this main page.

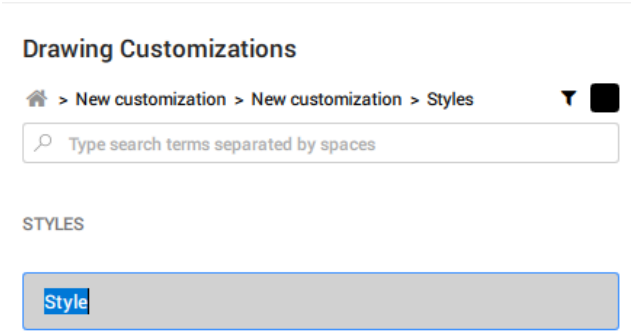
20.7.4.3 Drawing customizations styles

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.



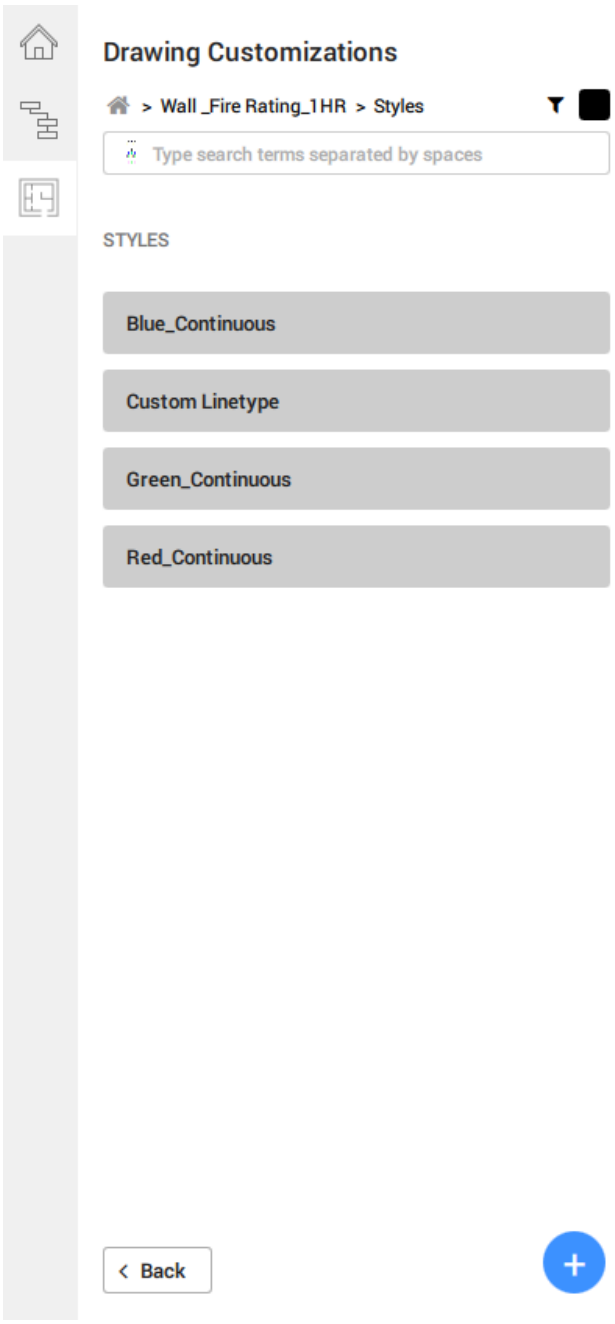
2 To create a new style, click + at the bottom right of the panel and a new **Style** template tab will appear.



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.

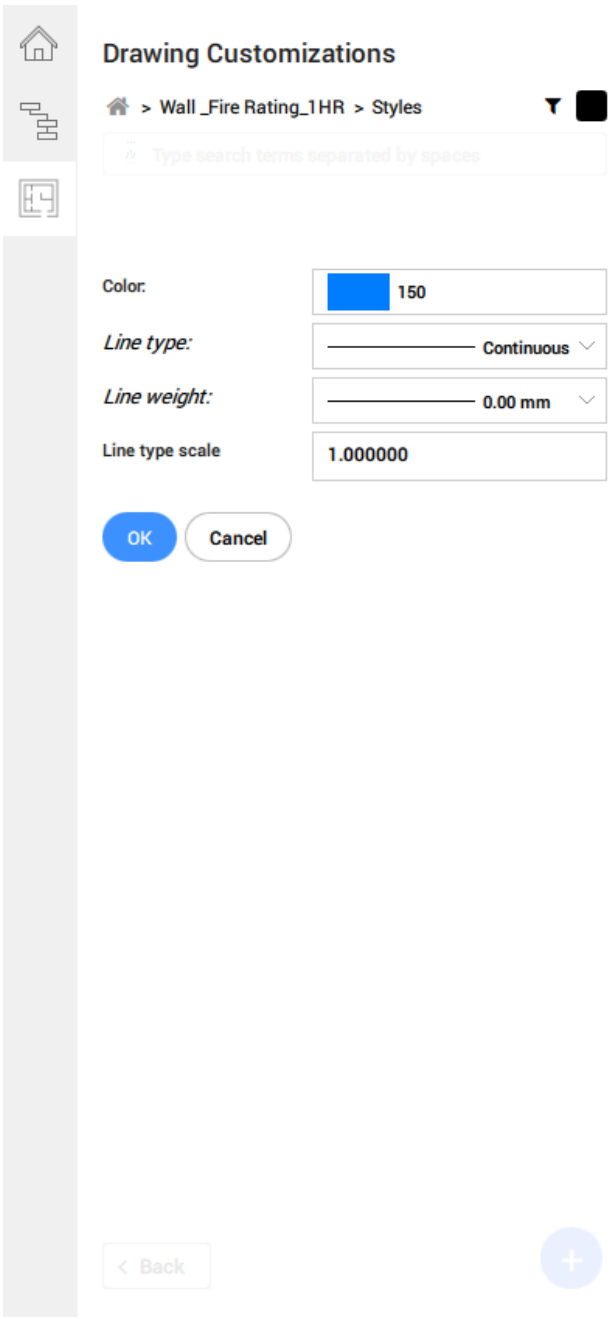
An example of the style pages a list of templates:



5 To edit any of the styles template available, click on its relevant tab to access its settings.

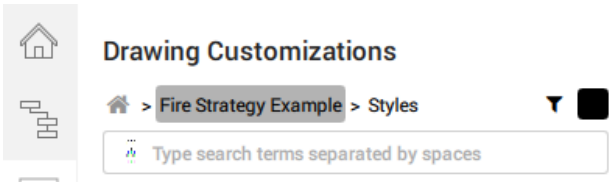
Uwaga: Similar to the **Drawing Customization template** tabs, hover over the styles template tabs to reveal the function icons to duplicate, rename or delete.

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.

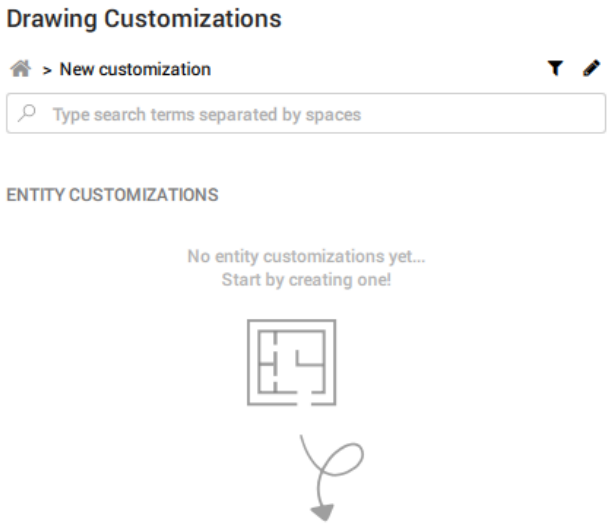
Or click on your desired branch in the tree structure map above the search bar to navigate to your desired page.



Uwaga: 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.

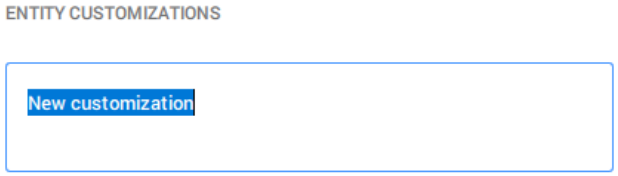
20.7.4.4 Entity customizations

Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

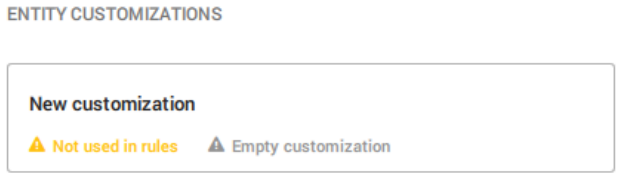


1 Click .

2 To create a new **Entity customization** template. By default, the template is named **New Customization** with the text highlighted.



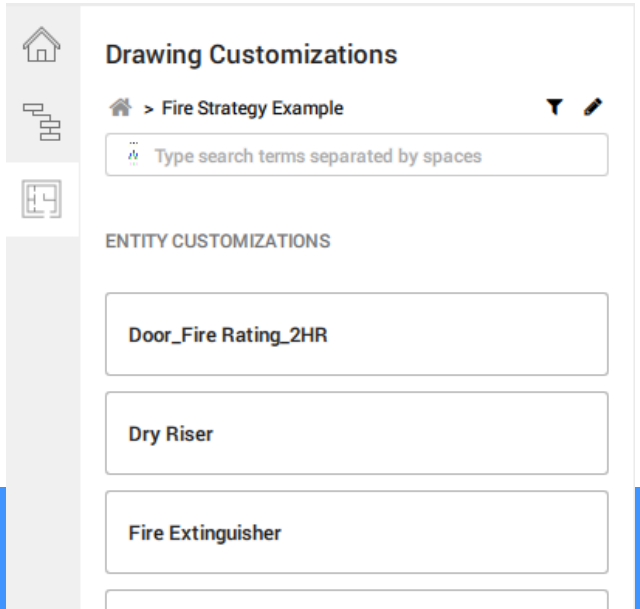
3 Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.



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.

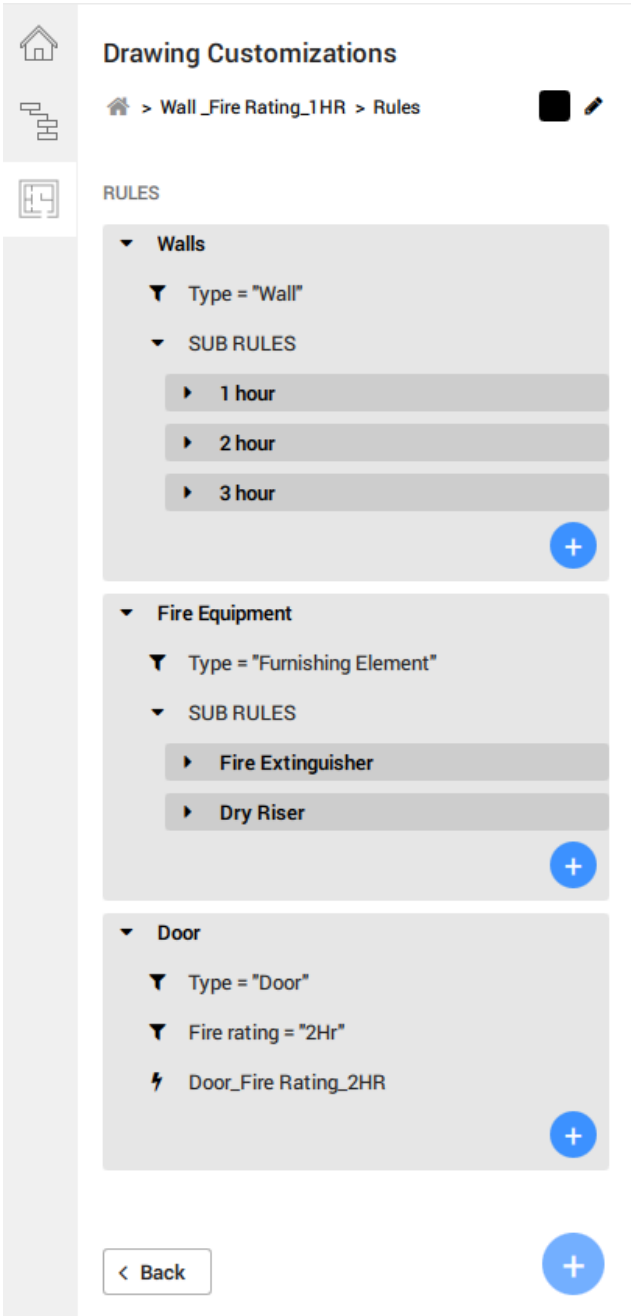
5 A list of **Entity Customizations** templates could look like this:



20.7.4.5 Filter rules

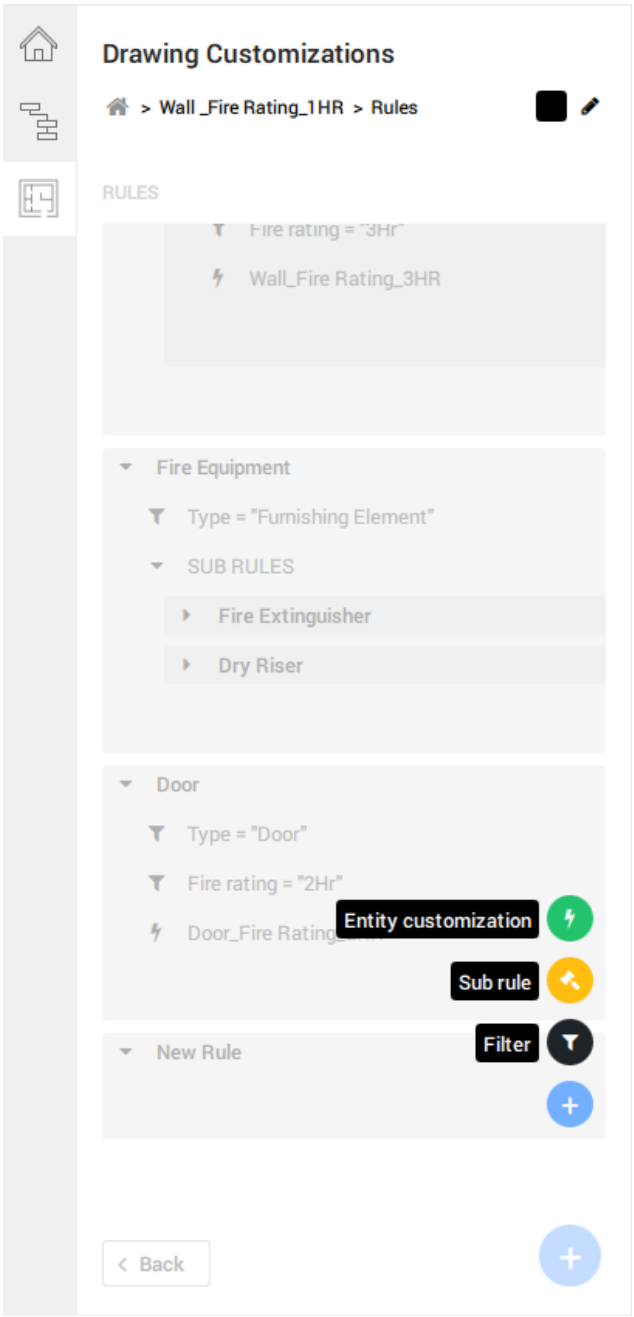
To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the **Filter Rules** tab located as a funnel icon on the top right-hand part of the panel next to the **Styles** icon.

Uwaga: 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.

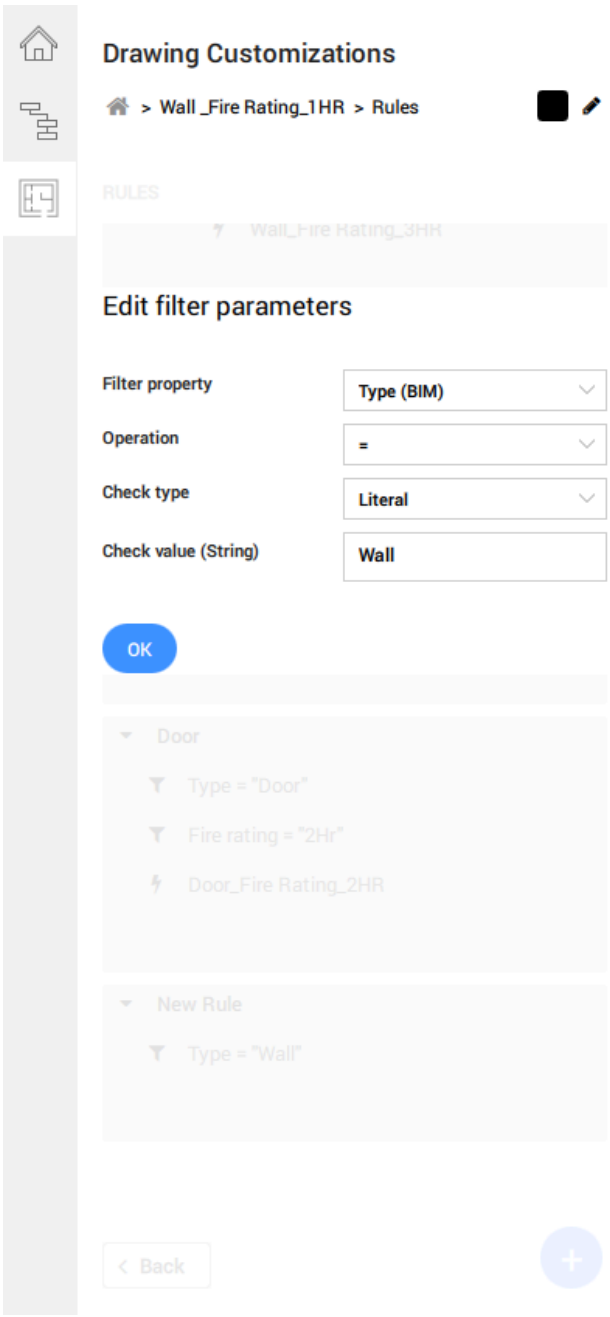


1 **Filter rules** can be further classed into **sub rules**, which falls as a sub-set of the parent rule, e.g. **Fire Equipment** as a **Filter Rule**, whereas **Fire Extinguisher** is classed as a sub rule.

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**.

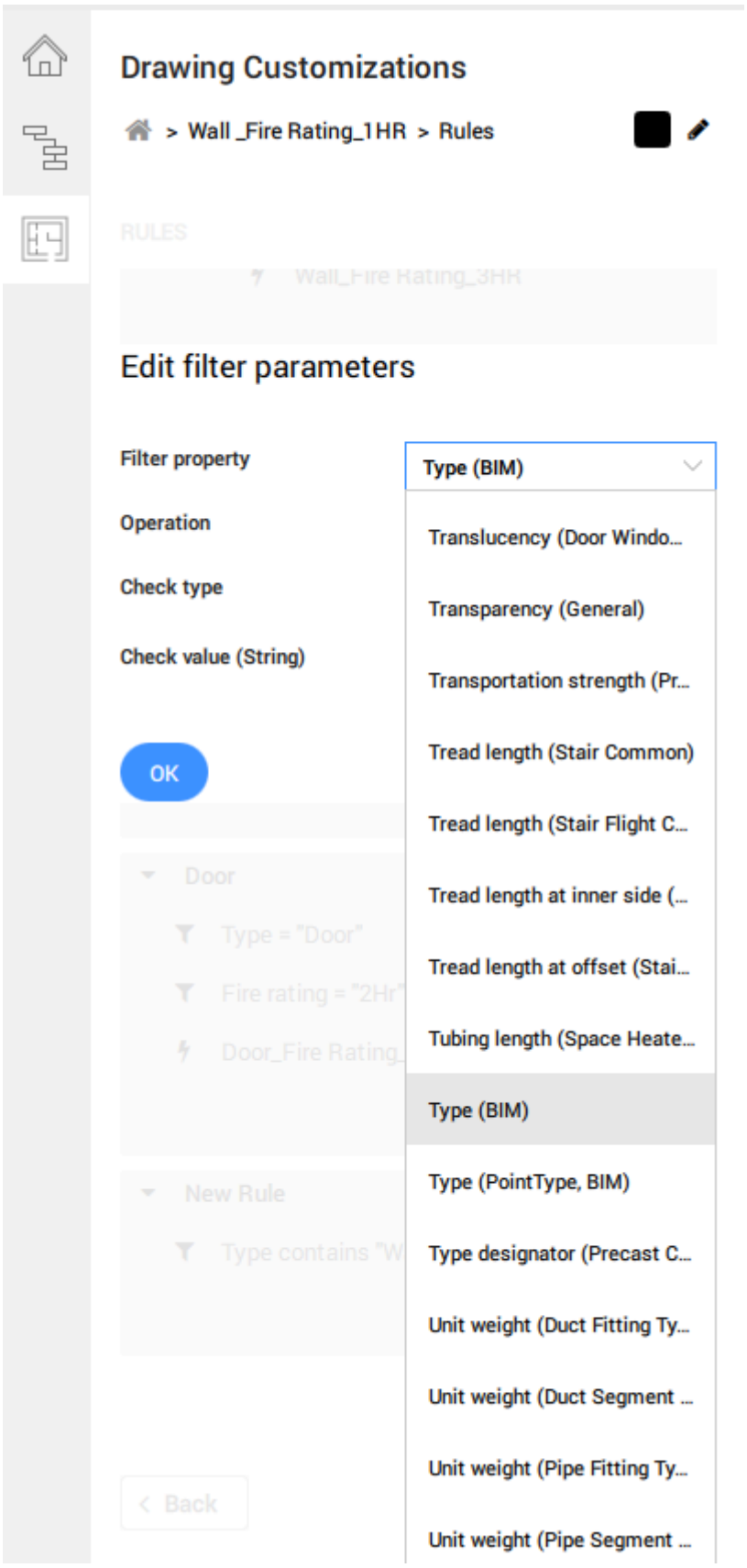


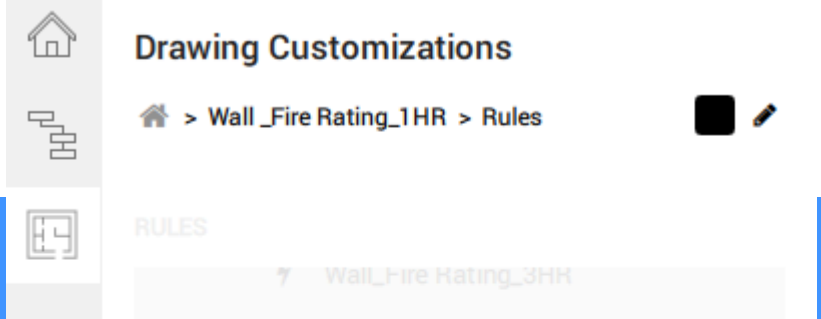
2 Adding and editing a filter parameter will lead you to a prompt, where the options of filter property are determined by the BIMPROPERTIES imported in the **namespace** settings.



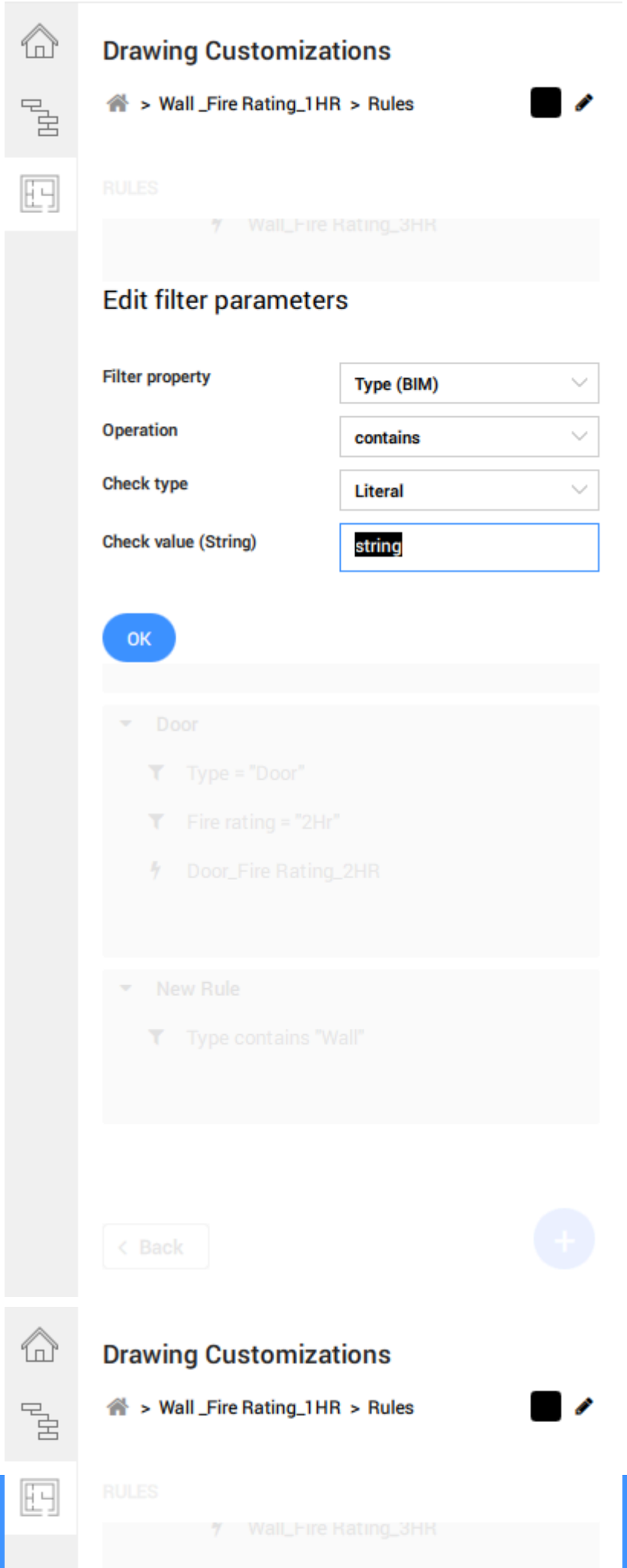
3 The filter **Operation** values are also available as a drop-down list.

Uwaga: You can also start typing a certain keyword to search a specific property that you want to filter on.

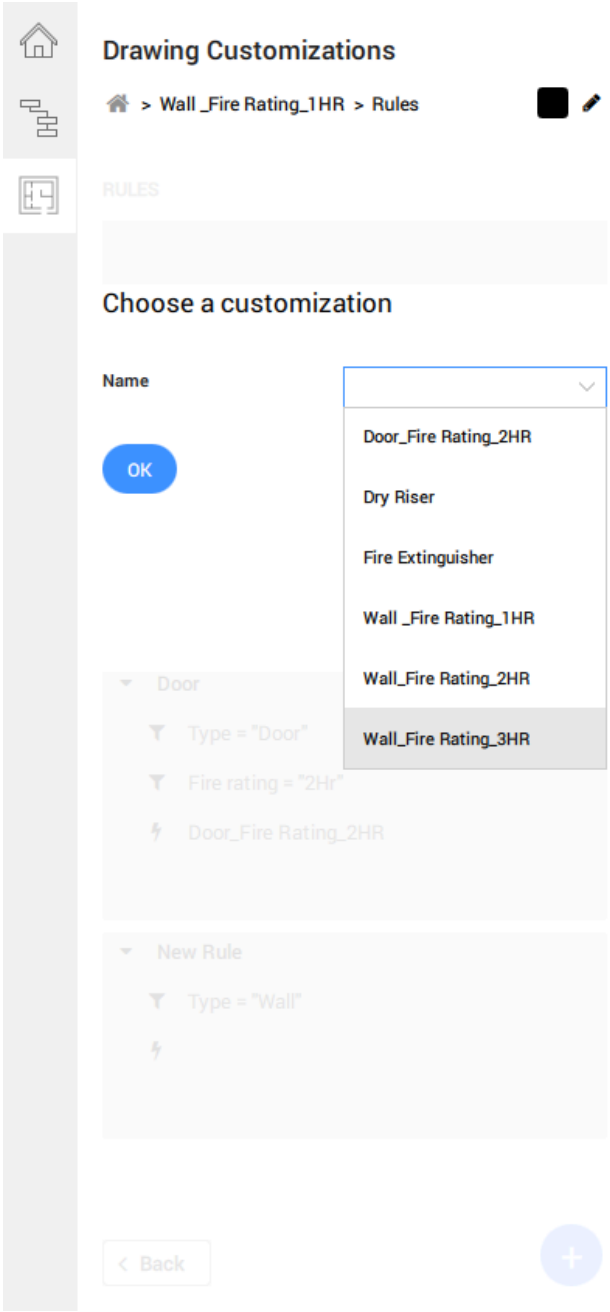

Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Edit filter parameters
Filter property
Type (BIM)
Operation
Check type
Check value (String)
OK
Translucency (Door Windo...
Transparency (General)
Transportation strength (Pr...
Tread length (Stair Common)
Tread length (Stair Flight C...
Tread length at inner side (...
Tread length at offset (Stai...
Tubing length (Space Heate...
Type (BIM)
Type (PointType, BIM)
Type designator (Precast C...
Unit weight (Duct Fitting Ty...
Unit weight (Duct Segment ...
Unit weight (Pipe Fitting Ty...
Unit weight (Pipe Segment ...
Door
Type = "Door"
Fire rating = "2Hr"
New Rule
Back


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
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

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.

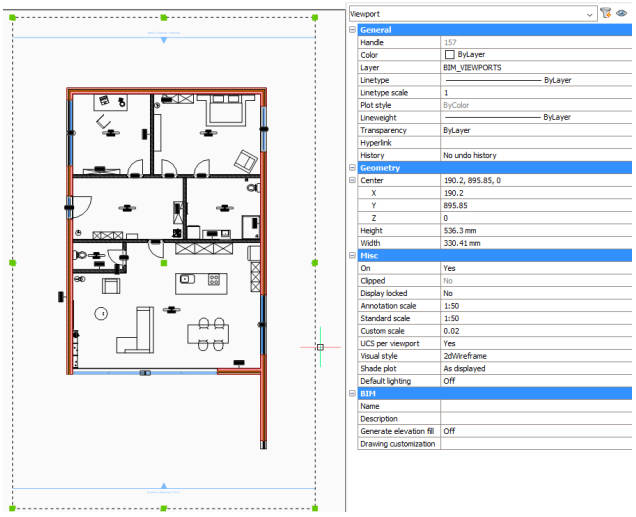


20.7.4.6 Applying templates to viewports on sheets

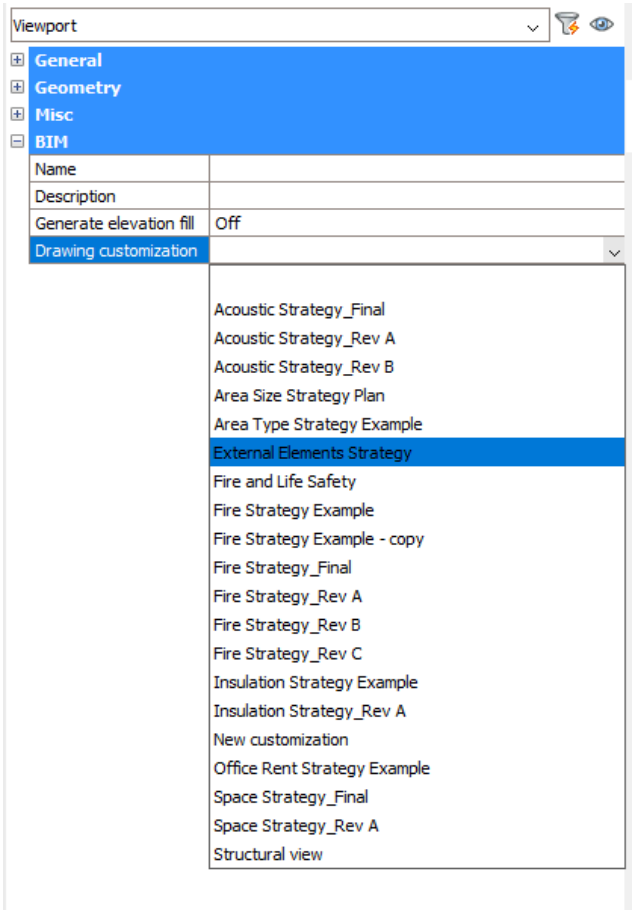
With your **Entity Customizations**, **Filter Rules** and **Styles** all set, your **Drawing Customization** template is now ready.

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.

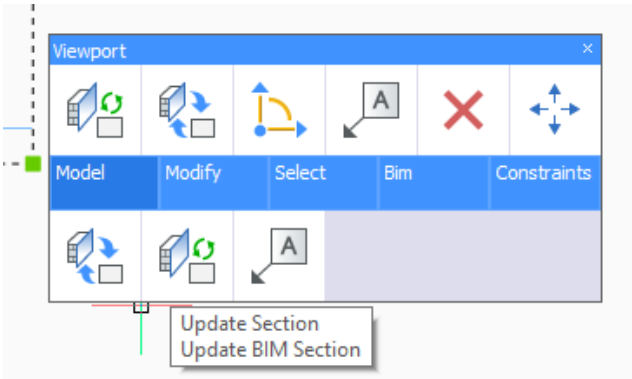


3 Navigate to the **Drawing Customization** property and click on the drop-down selection to the right.



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.

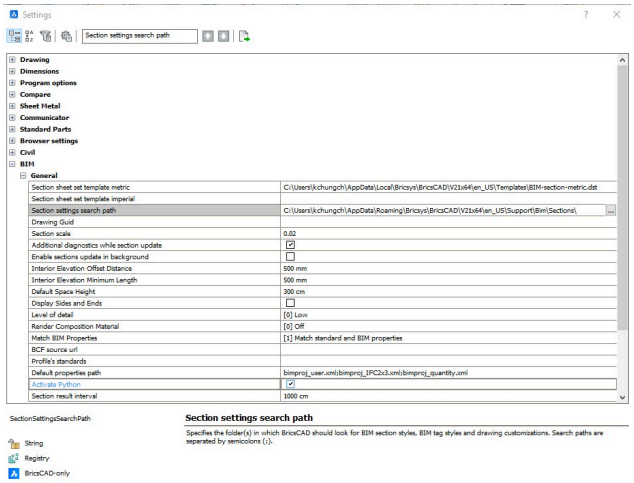
20.7.4.7 Accessing customization templates

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

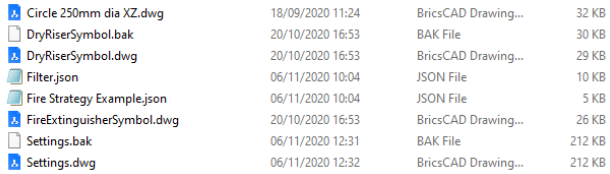
The **AppData** folder is hidden by default, so you may have to unhide it first. This path can also be accessed by entering SUPPORTFOLDER in the Command line.

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.



Uwaga: 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:

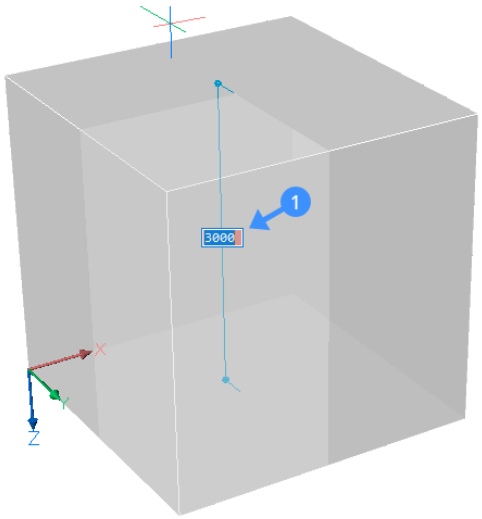
4 Select the opposite corner snap to a point or enter the distances. Use the TAB key to switch between dynamic dimension fields.

5 **You are prompted**: Select the height of volume box <4000>:

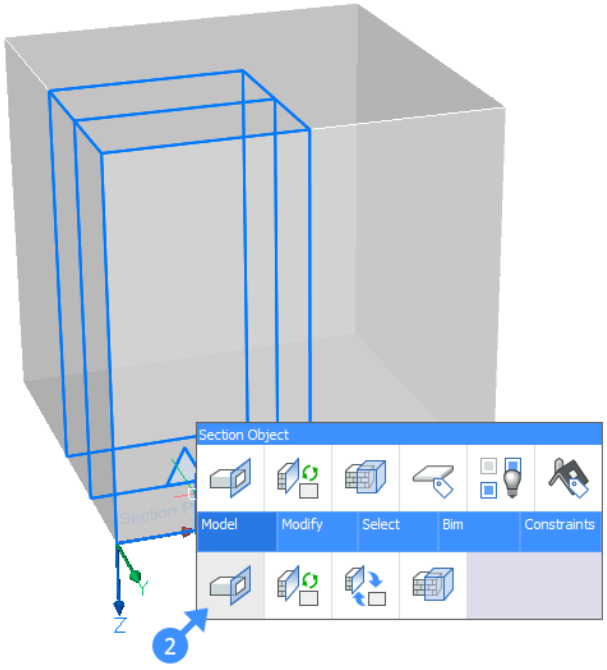
In this case, the default height is 4000mm.

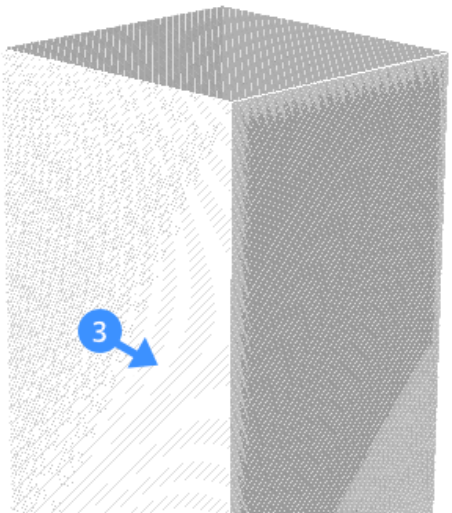
6 To complete the volume, do one of the following:

- 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).

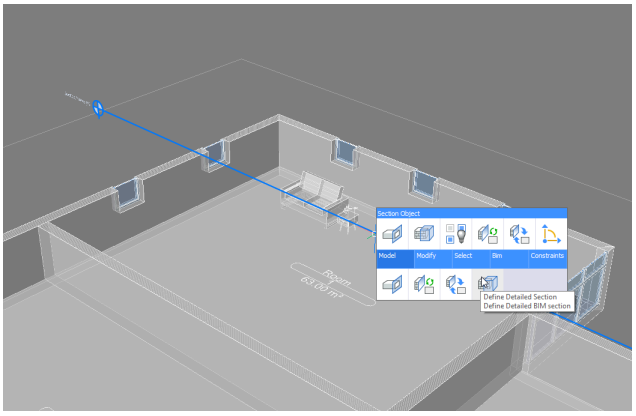




20.7.5.3 Define a detailed section based on an existing section

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.

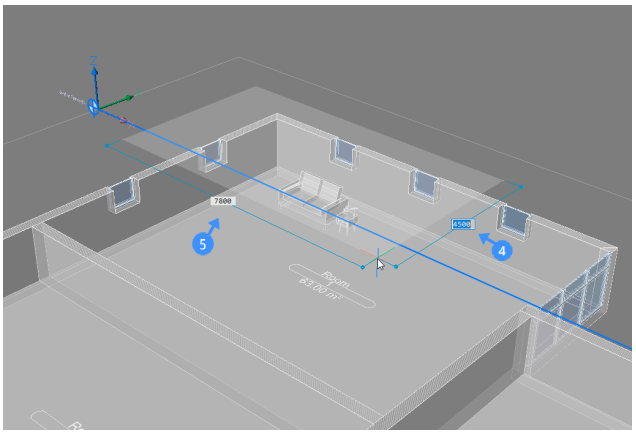


You are prompted: Select first corner of volume box base rectangle:

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.

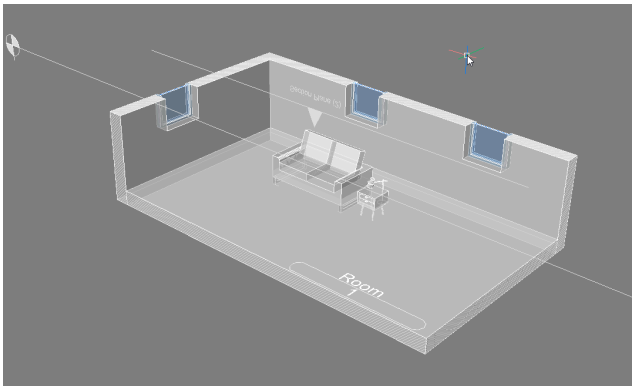
6 Specify a point.

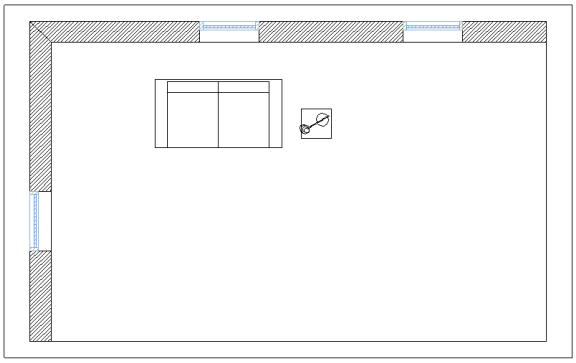
7 Type a value in the dynamic dimension field.

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.

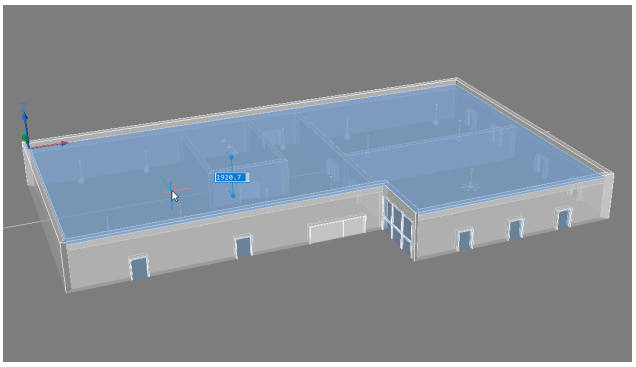
20.7.6.3 Create a reflected ceiling plan

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.

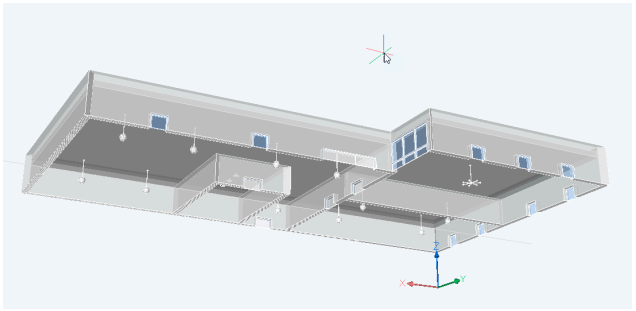


You are prompted: Specify distance:

3 Do one of the following:

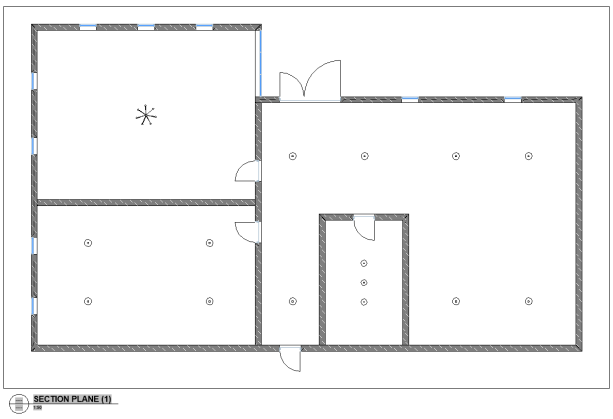
- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want.

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

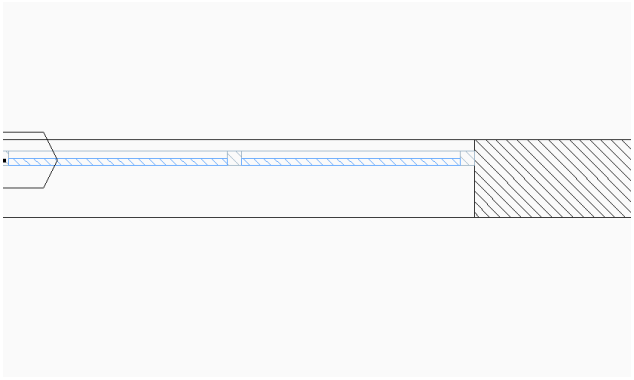
Uwaga: BIMPATCH cuts out part of geometry and replaces it by a block reference. In this cutting process, entity data linked to the 3D model are lost. Therefore, starting from V21.2, this command is no longer available.

20.7.7.1 Command

BIMPATCH

20.7.7.2 About

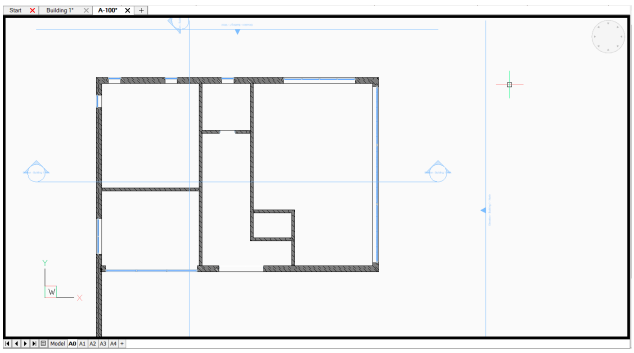
In this procedure, we will add fixtures and a vapor barrier to our window-wall connection. We will detail this window-wall connection.



20.7.7.3 Procedure: detailing a window connection

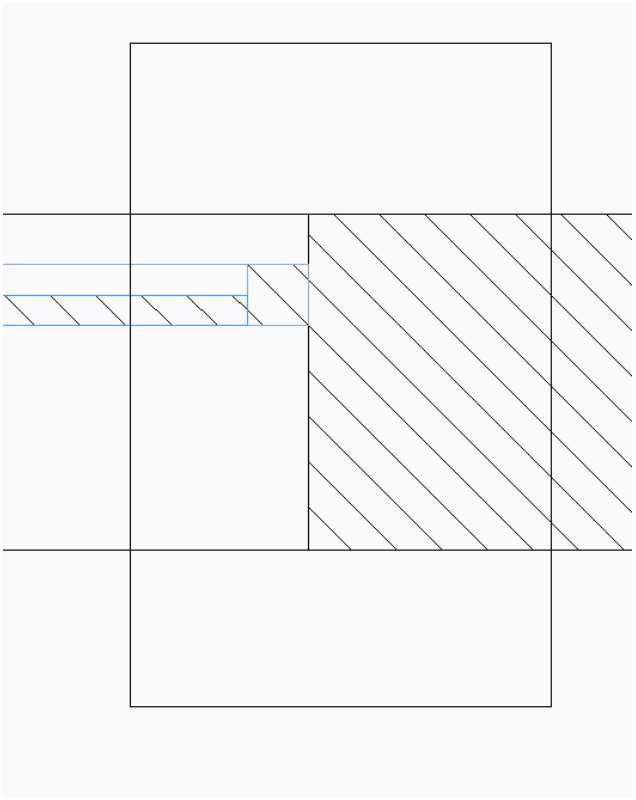
1 Open the section.

2 Go to the model space by double-clicking in the viewport **or**toggle the viewport mode in the **Status** Bar.

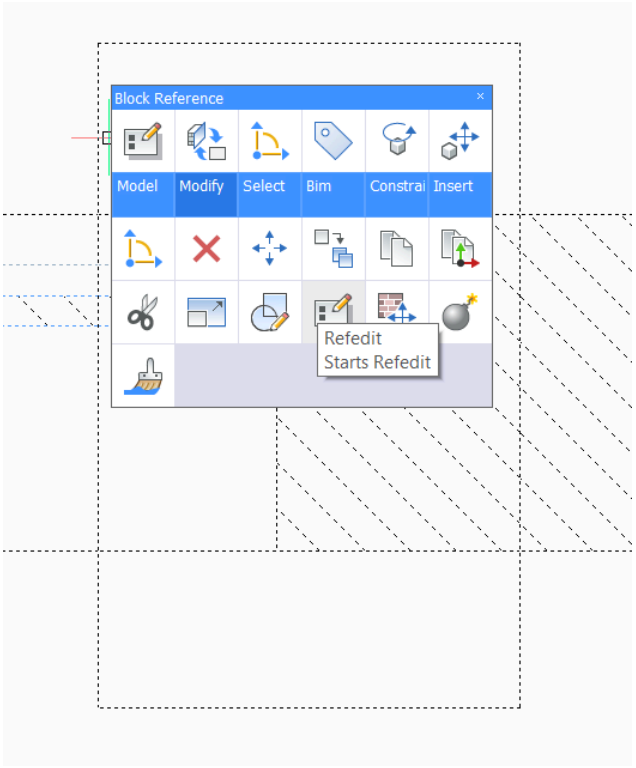


3 Launch the BIMPATCH command in the model space.

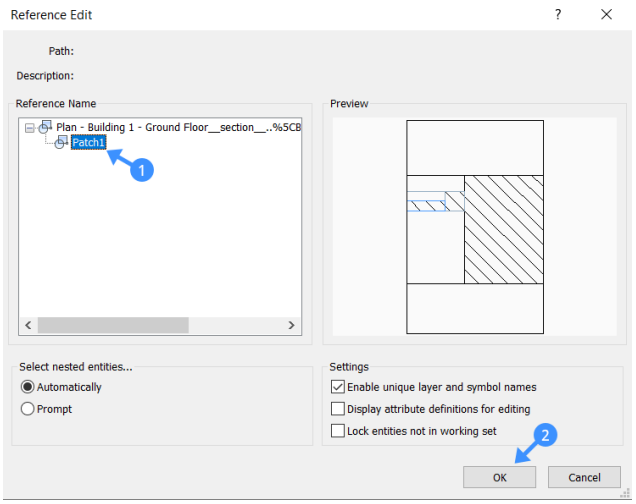
4 Draw the rectangle over the detail and hit enter.



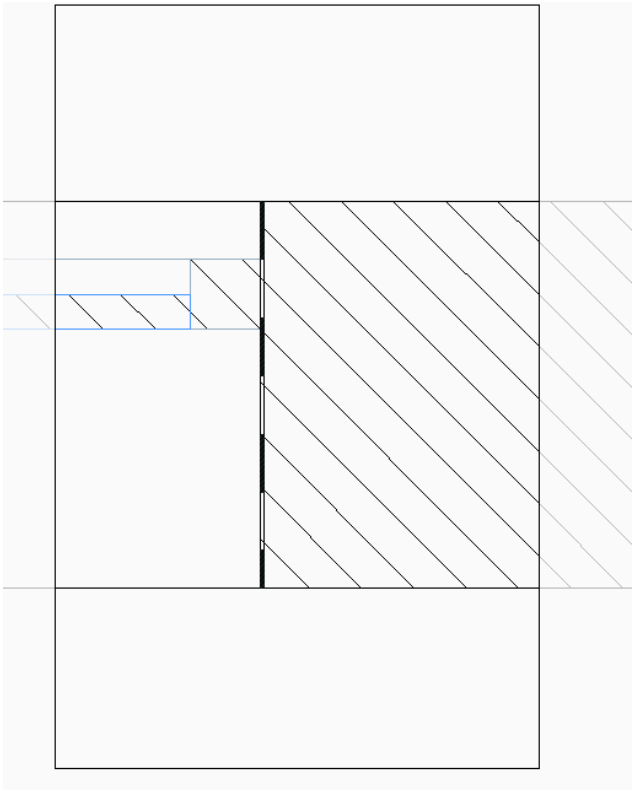
5 Hover over the section object and launch **Refedit** from the quad.



6 A menu appears, select the patch (1) and click **OK** (2).

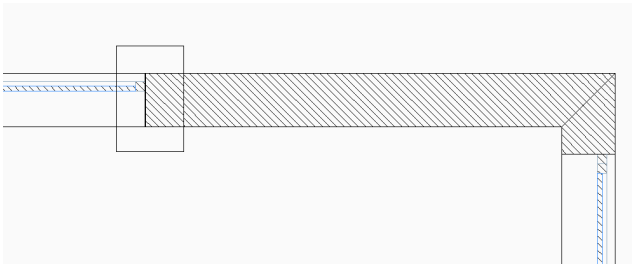


7 Draw the required detail for the window-wall joint. We will add a vapor barrier.

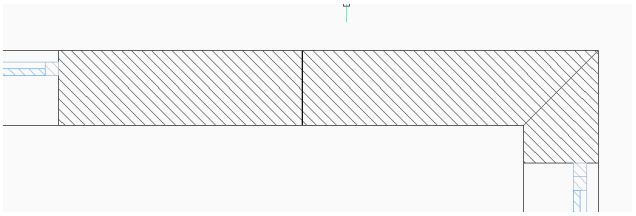


8 Save the changes made in the **RefEdit** session with the **Save** option of the REFCLOSE command.

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.



Uwaga: 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)>

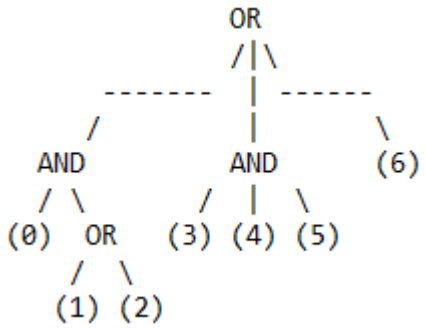
The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BIMCLASSIFY command to learn more about BIM entity types.

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.

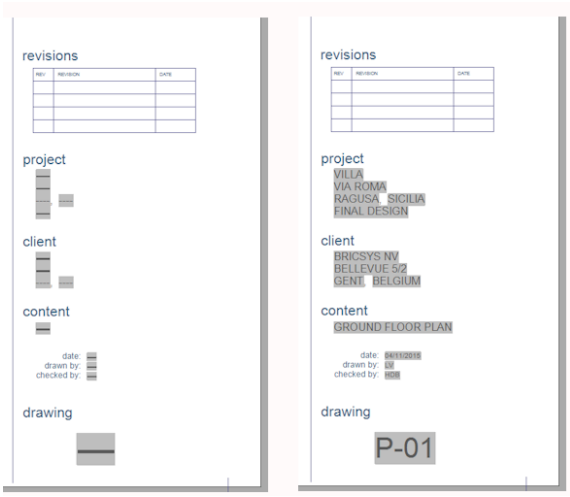
In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm)

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.

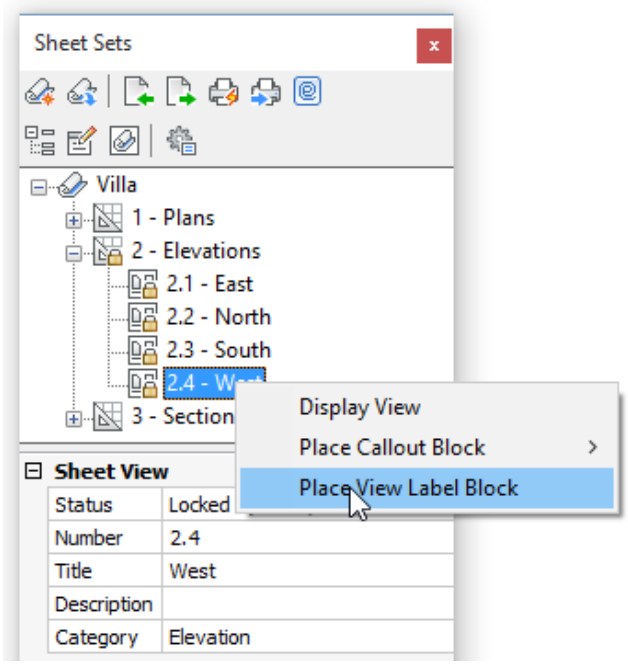
5 Select the **Title** block, then click the **Update Fields...** tool button ( ) in the **Redraw/Regen** toolbar.



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>:

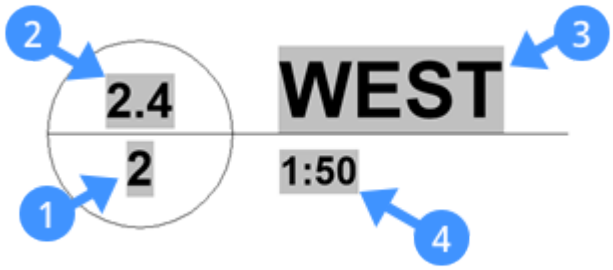
6 Specify the scale factor or right-click to accept the current scale.

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.

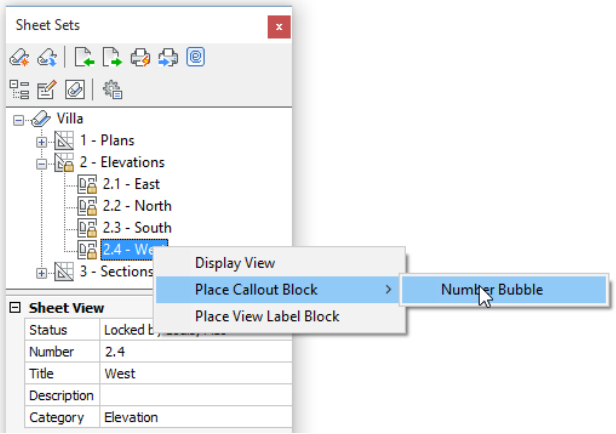
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.



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>:

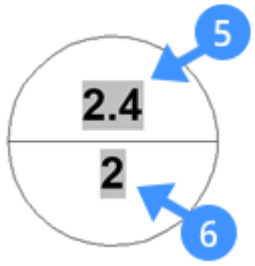
8 Specify the scale factor or right-click to accept the current scale.

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

Uwaga: **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**.

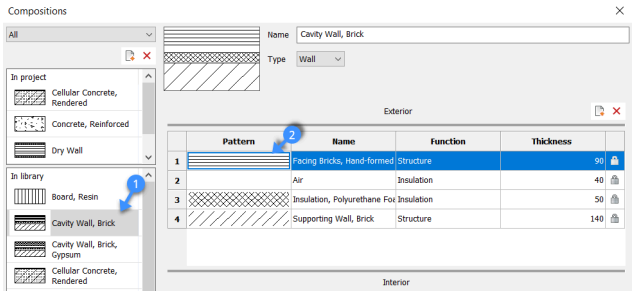
7 Select the viewport and choose **Update BIM Section** in the **Model** command tab in the Quad.

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.

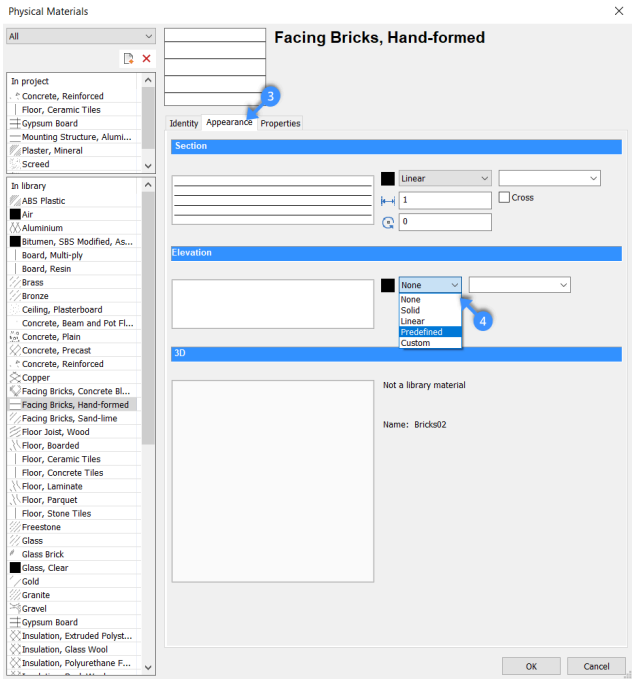
20.7.10.3 Configure the hatch

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).

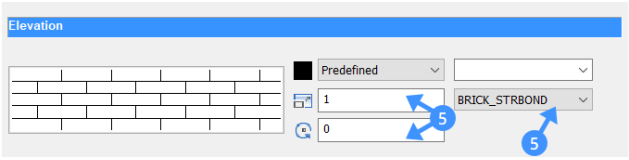


3 To configure the texture or solid fill, go to the **Appearance** tab page (3). The elevation hatch is set by default to **None**. Click the elevation hatch type button and select the type of hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom** (4).



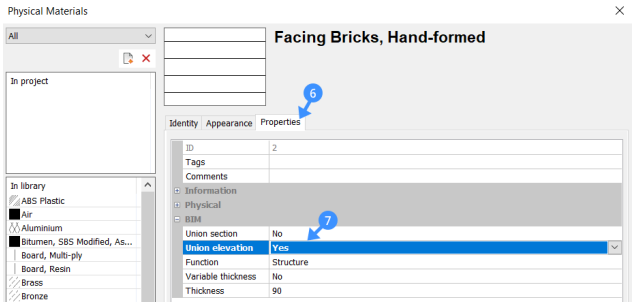
4 Configure the hatch. Define the scale, angle, color and/or layer of the pattern. (5)

Uwaga: Type a name in the layer field or click the arrow button and select a layer in the list.

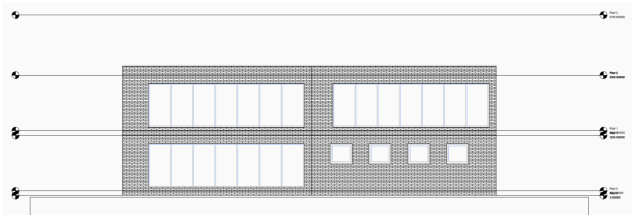


5 To remove connection lines in the elevation between different solids of the material go to the **Properties** tab (6) and set **Union elevation** to **Yes** (7).

Uwaga: **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.

Uwaga: 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

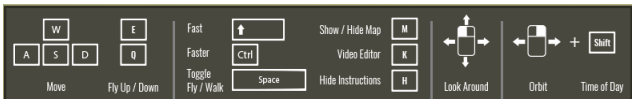
1 Click on **Start Enscape** in the ribbon **or** type in STARTENSCAPE in the Command line.

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.

Uwaga: 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.

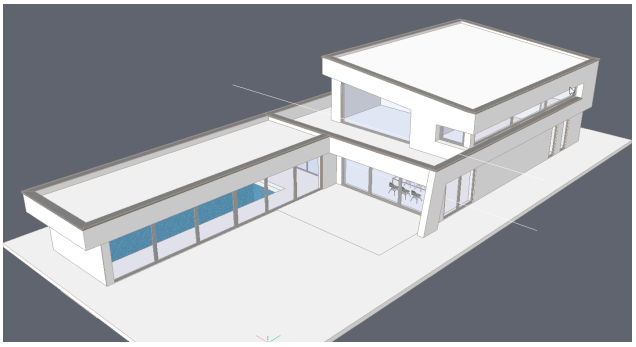
Render materials with the Render Materials panel

- 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.



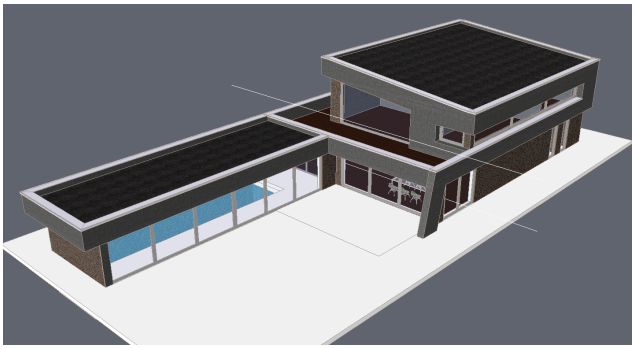


Figure 1: Level of detail = 0,Figure 2: Level of detail = 2

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**.

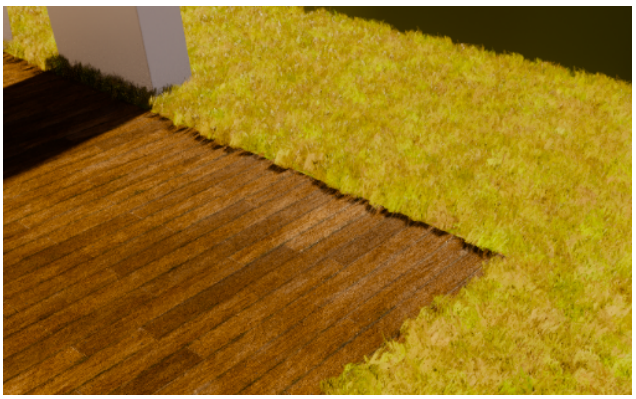
3 Give the new material a name that includes the name 'Grass', for example, ‘Grass_ground’.

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.

Uwaga: 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.

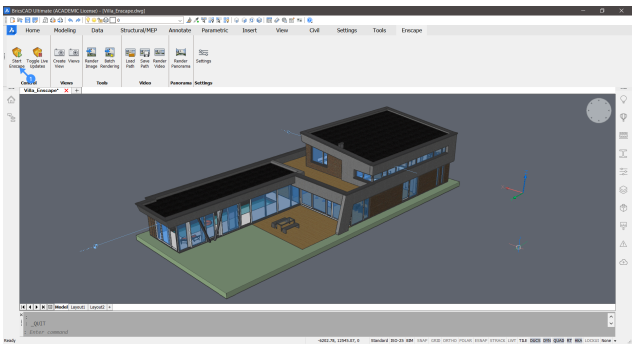
4 Select **Batch Rendering** in the ribbon, or type in ENSCAPEBATCHSCREENSHOT in the Command line, to render all the saved views to a folder.

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

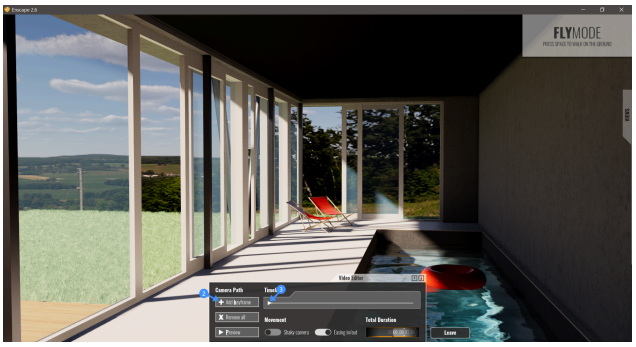
1 Click on **Start Enscape** in the ribbon or type in STARTENSCAPE in the Command line.



2 In the Enscape window: type K to open the **Video Editor**.

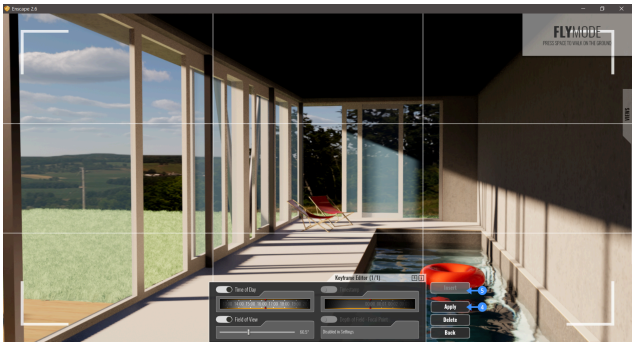
3 Click **Add Keyframe** (2) to add the first keyframe.

4 Click on the triangle to open the keyframe settings (3).



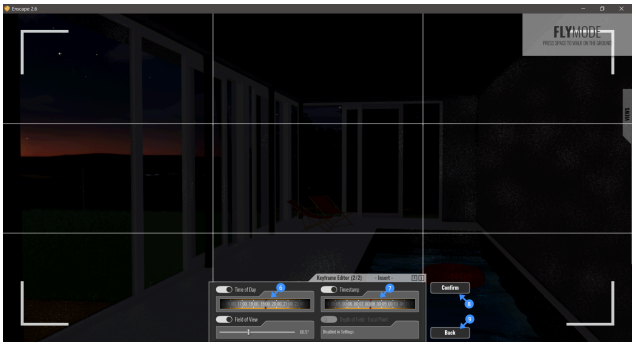
5 Change the parameters as required.

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).

Uwaga: 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

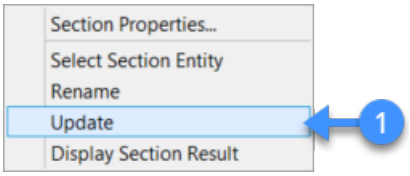
In the example above 2 plans, 4 elevations, and 2 cross-sections were generated and exported to 3 drawings.

Project browser

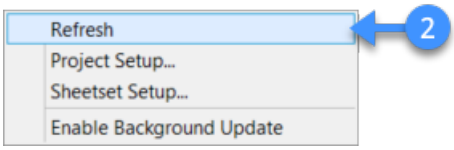
1 In the **Sections** segment of the **Project Browser**, right-click the section.

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.

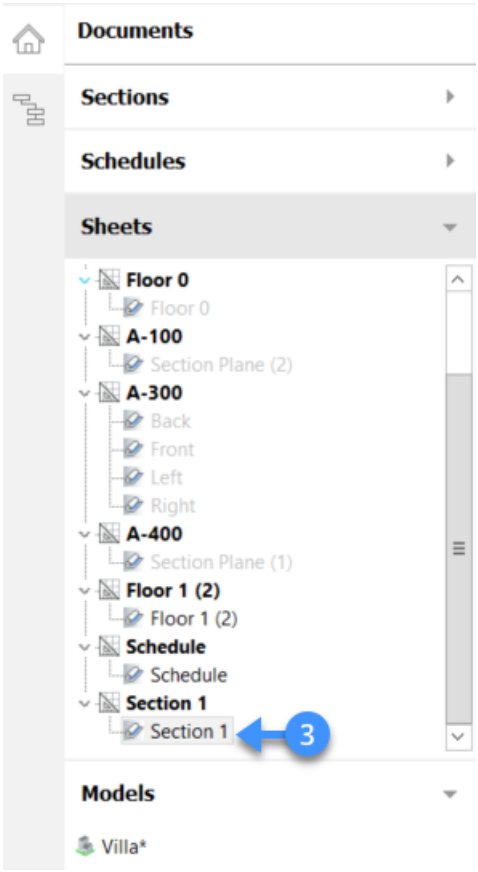
3 The 2D section drawing(s) open(s).

Project browser

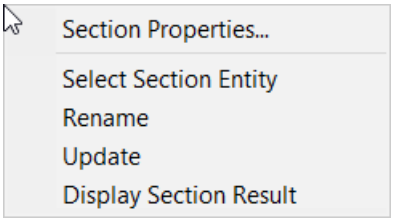
To open the 2D section drawings from the **Project Browser**, do one of the following:

- 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.

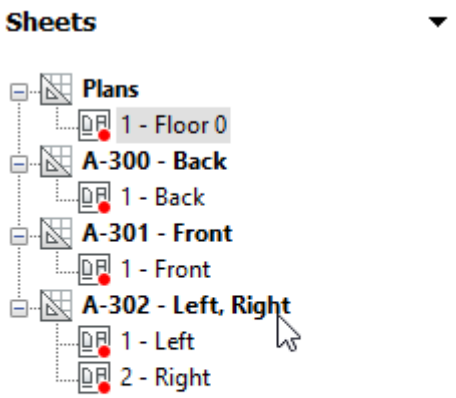


20.7.12.5 Update BIM sections

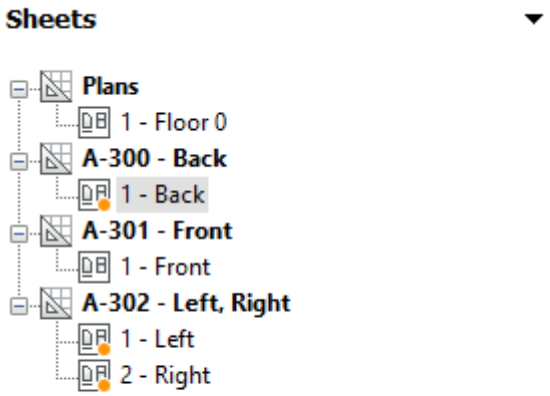
BIM Sections can be updated in different ways:

- 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.

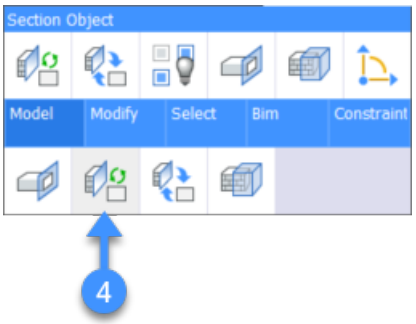
Update BIM sections in the 2D documentation file

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).

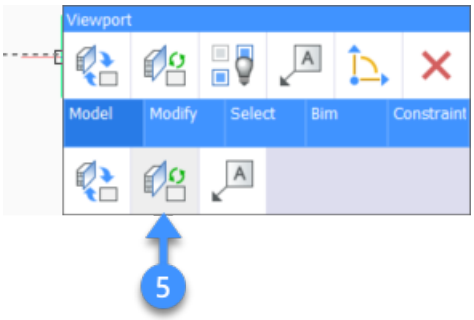


Update a BIM section in a layout

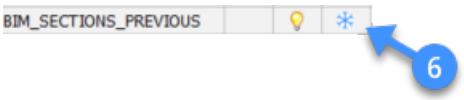
1 Click the **Layout** tab.

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).



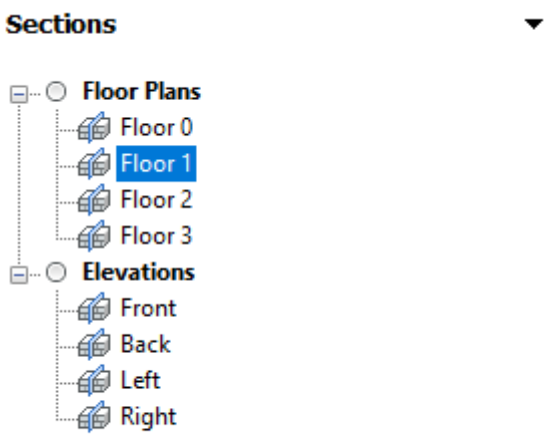
The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer (6) to check the modifications.



The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta.

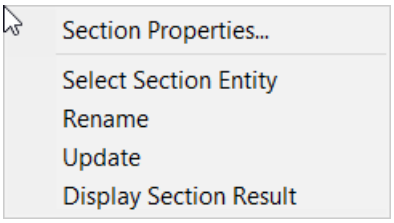


Update BIM sections in the project browser



1 Right-click on a section in the **Project Browser**.

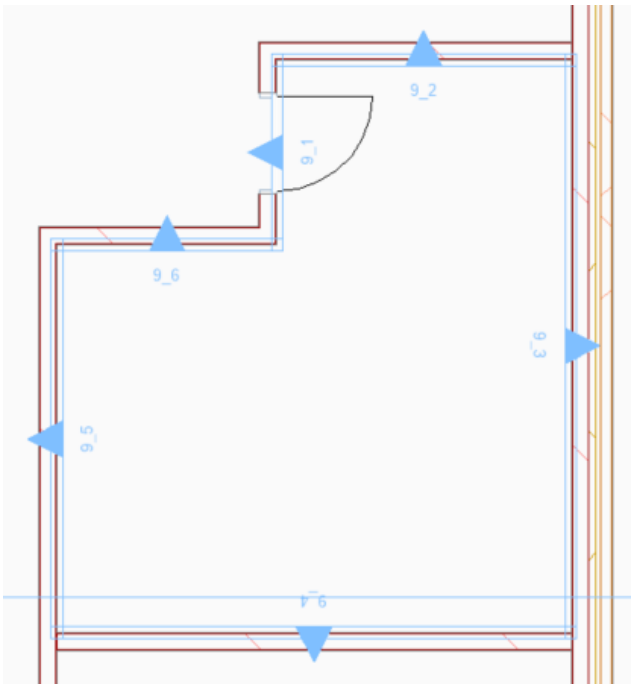
2 Select **Update** in the context menu.



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.

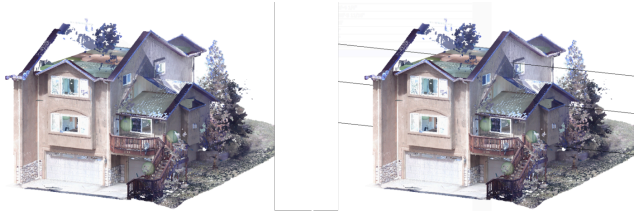
Detecting floors from Pointclouds

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.



Uwaga: 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

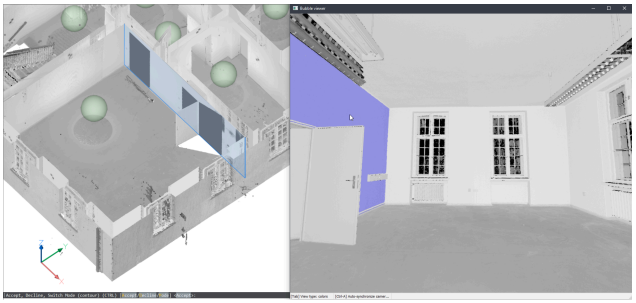
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.

1 Double-click a green point cloud bubble to open the **Bubble viewer**.

2 Type POINTCLOUDFITPLANAR in the Command line.

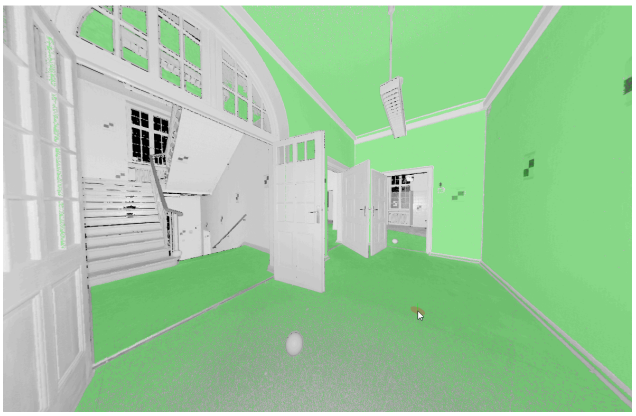
3 Select the desired wall.

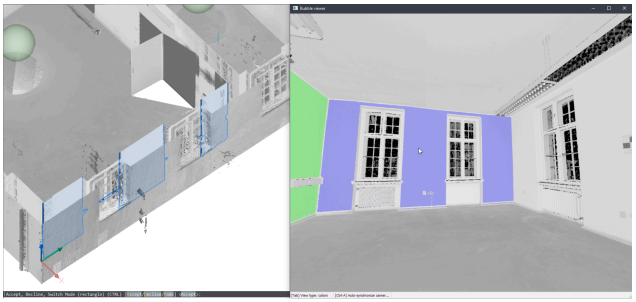




4 Press A to accept the selection.

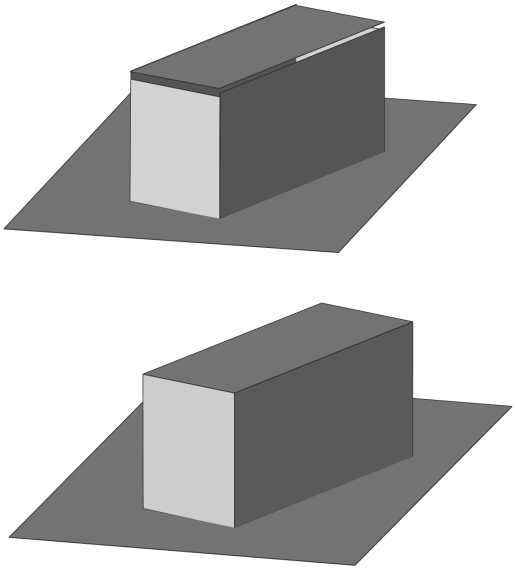
5 **(Optional)** Select more walls.





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:

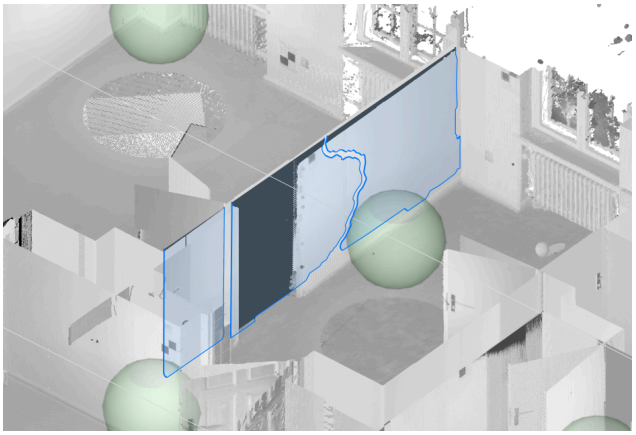
- It can create larger surfaces where only parts are visible in each scan position.
- It can detect wall and slab thickness since it can take the opposite surface into account.

1 Type POINTCLOUDFITPLANAR in the Command line.

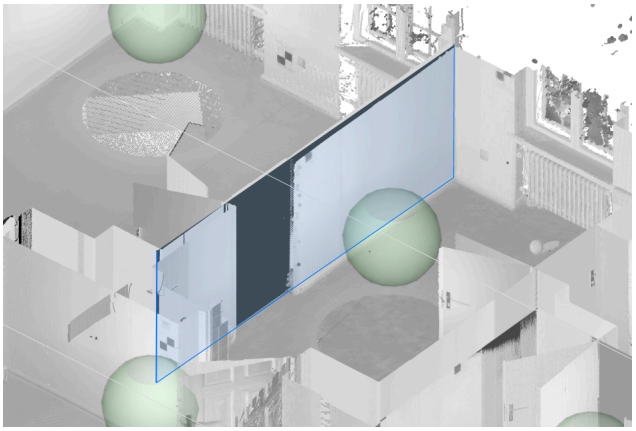
2 Select a wall where you want to create a plane or solid.

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.

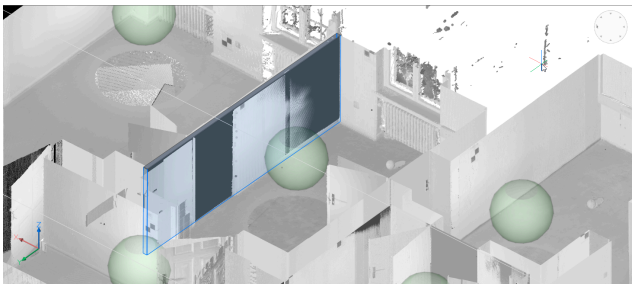
Contour mode will create the best fitting plane:



Rectangular mode will create a rectangular plane around region of planar points:



Solid mode will create a solid from two parallel point cloud planes:



Uwaga: 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.

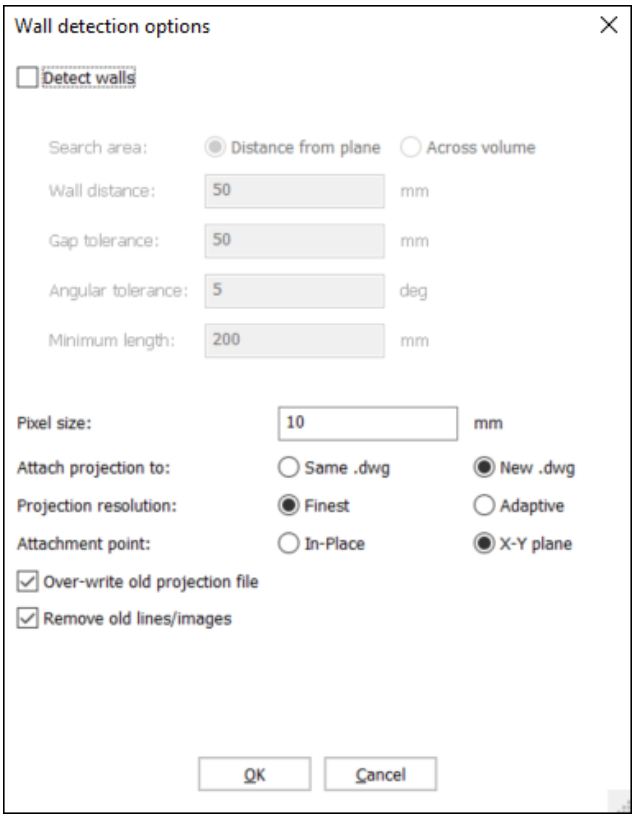
20.7.13.5 Procedure: creating a raster image from a point cloud

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.

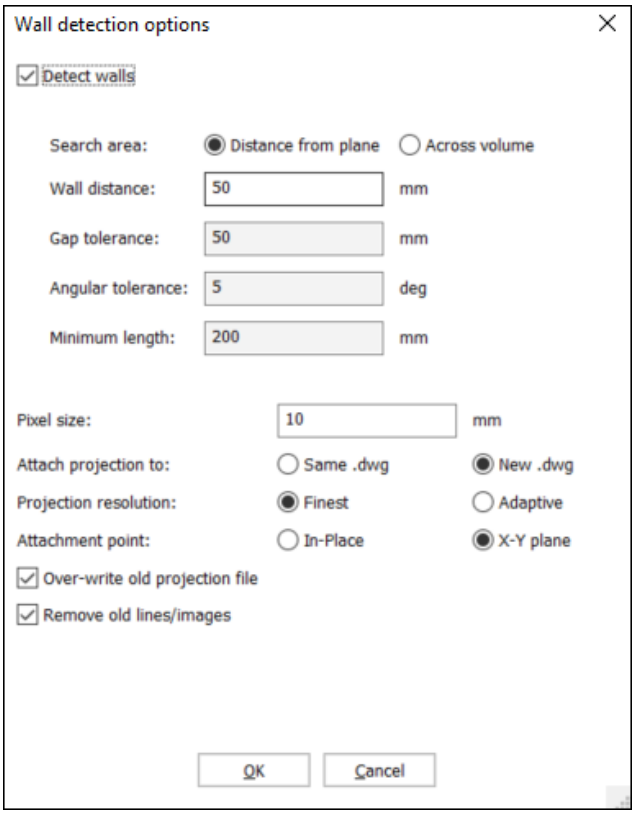
2 Select a section volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section.

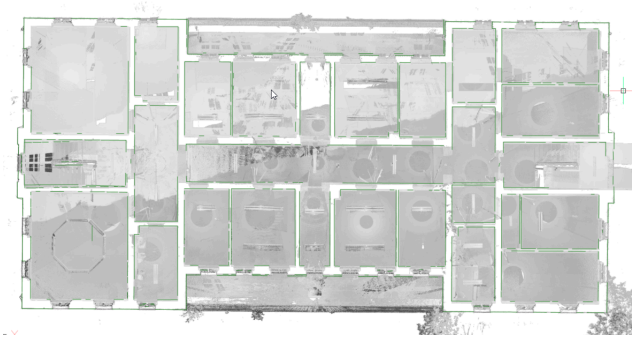
Uwaga: See the command reference page for more information about the customization options;

Uwaga: Turn on **Detect Walls** to generate the green lines as seen in the image below.



4 Press Enter to create the projection.

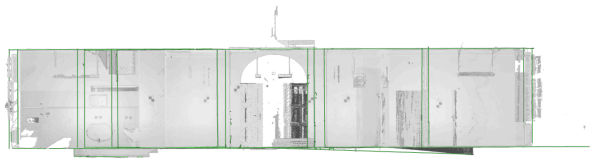
Uwaga: If there are some missing lines, you can draw them yourself by tracing the background image.



Making vertical sections from point clouds

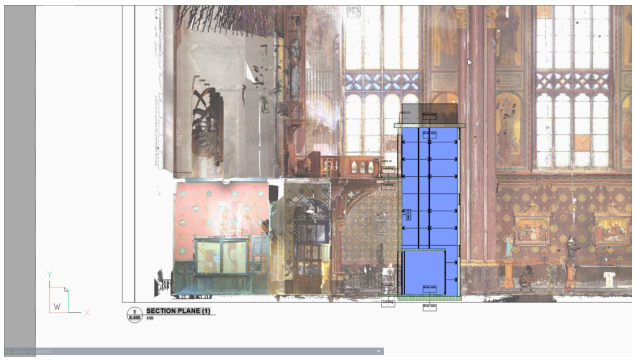
Make a vertical section box on the place you want the vertical projection.

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.

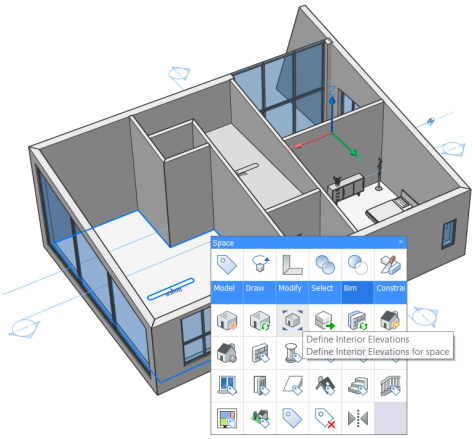


For more information about this command, please read BIMSECTION.

20.7.14.3 Create interior elevations from a BIM space

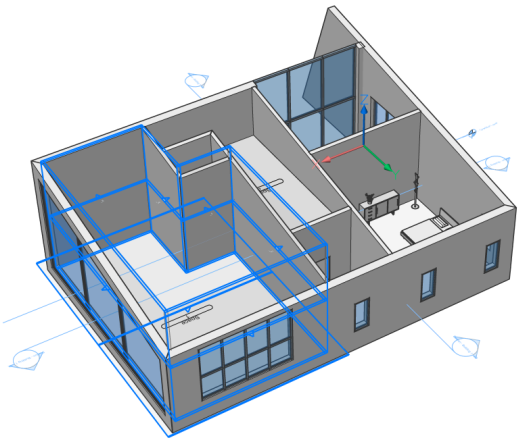
1 Hover the cursor over a Space Tag and select **Define Interior Elevations** from the **BIM** Tab in the Quad
Or type BIMSECTION in the Command line, followed by i and Enter to choose the interior option.

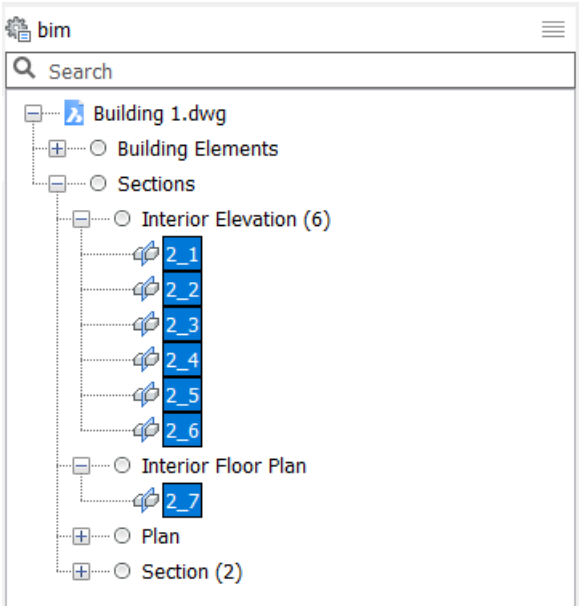
You are prompted: Select BIM Spaces to generate Interior Elevations or [Attach section]:



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**.

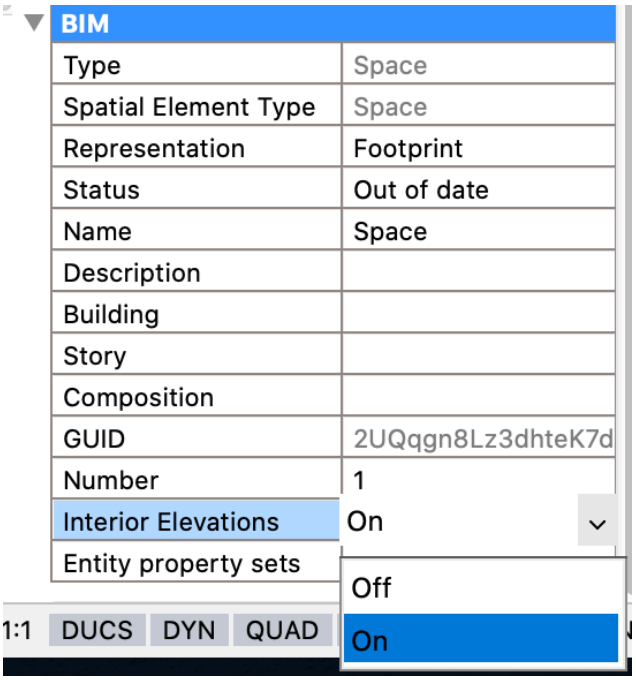




Six interior elevations and one interior floor plan have been created and will appear in the structure browser.

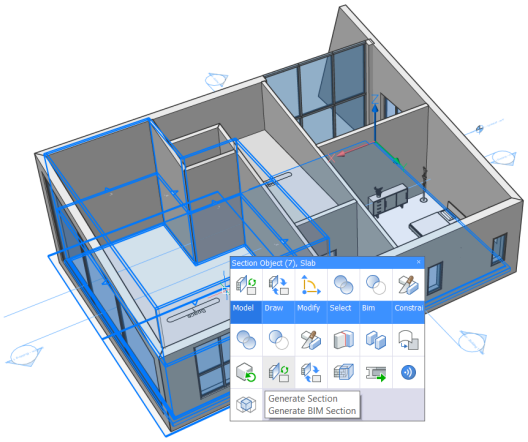
Uwaga: If there already are interior sections for a Space, running BIMSECTION again on that Space will have no effect. To regenerate the interior elevations, you must delete the existing interior elevations in that Space and run BIMSECTION again.

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.



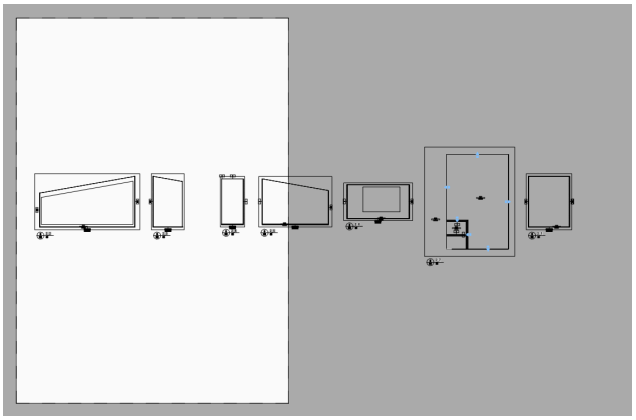
5 Select all elevations, as illustrated above and select **Generate Section** from the Quad to generate the

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:



Uwaga: 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.

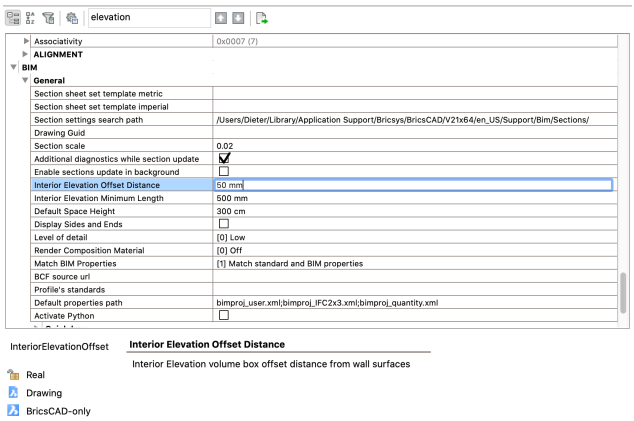
If you need more information on how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see **Annotate** > **Creating linear dimensions**.

20.7.14.4 Edit the size of the interior elevations

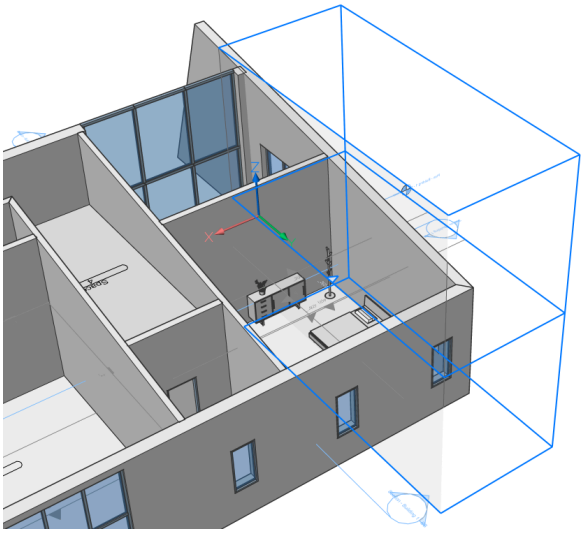
It's possible to edit the size of these sections to include furniture for example. If you create interior elevations for a BIM Space, the offset value is 50mm by default. To include furniture in a section, you have to change this value.

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.

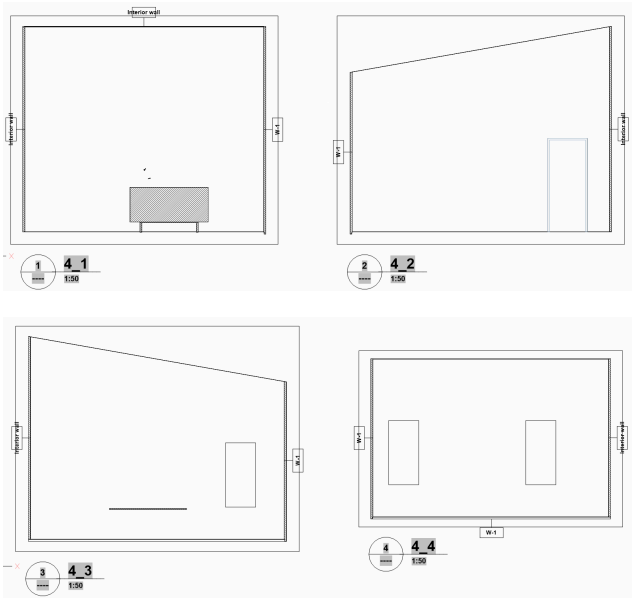
You can also change the offset by typing Settings in the Command line, search for elevation and adapt the **Interior Elevation Offset Distance** value.



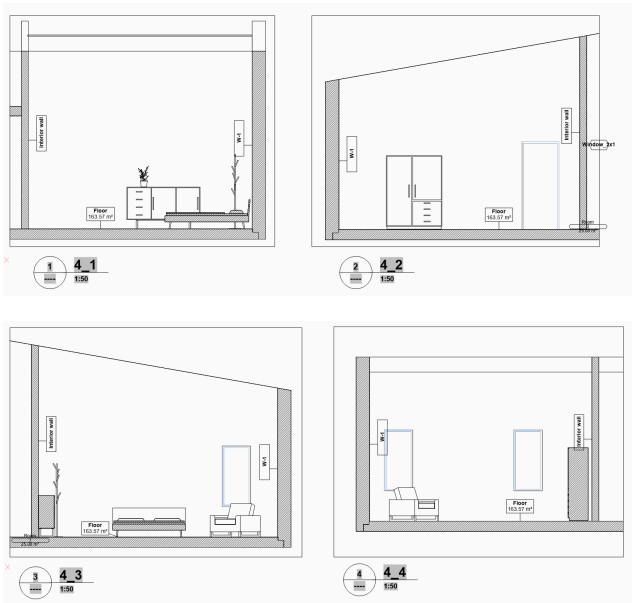
2 If you make new Interior Elevations, the offset distance will be the chosen value. This is also visible when you select the interior elevations in your drawing:



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

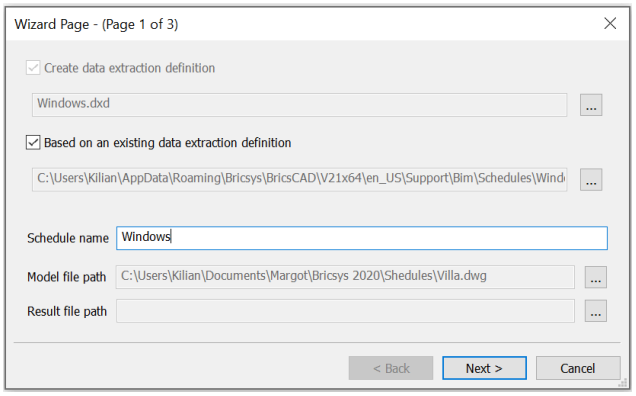
For more information about creating a project, see the article Project browser.

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.

Uwaga: 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 (...).

Uwaga: 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.

Uwaga: 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.

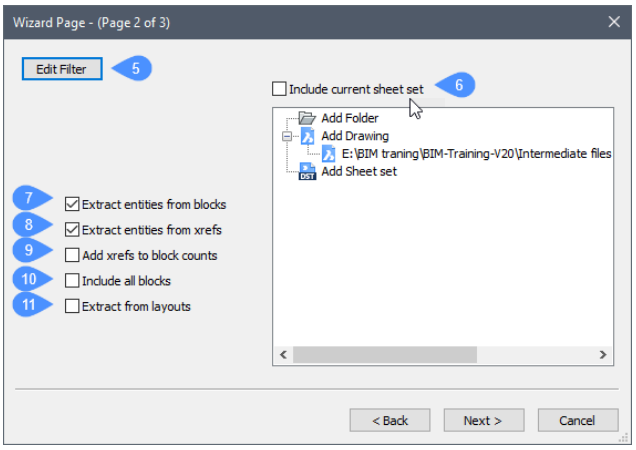
(3) Model file path: The model path links to the model. This is filled in automatically if the wizard is started in the model file. If the wizard is started in another file from the project, link the model file:

a Click the **Browse** button (...).
b In the file dialog box, select the model file.
c Click the **Open** button.

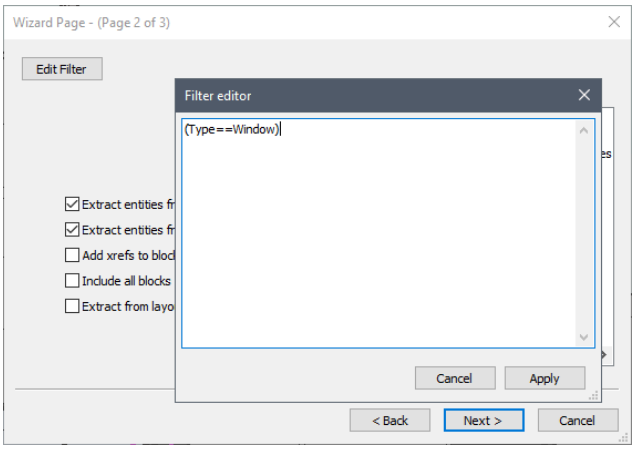
(4) Result file path: The **Result file path** determines where the schedule is created. When left blank, the Schedule is created as a new sheet:

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.

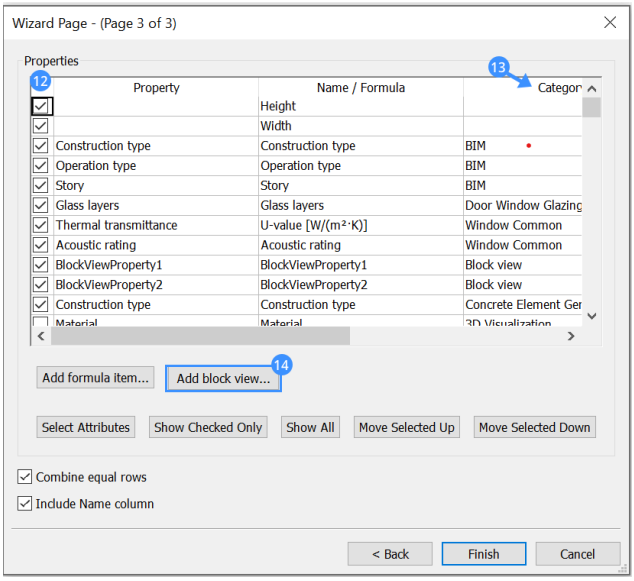
(9) Add xrefs to block counts: Counts selected xrefs as blocks.

(10) Include all blocks: Includes all blocks into the schedule.

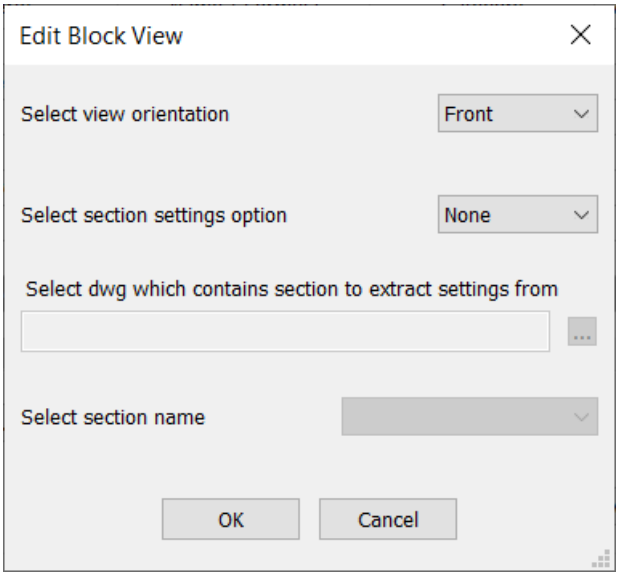
(11) Extract from layouts: When toggled on, all entities layouts will also be processed in the schedule.

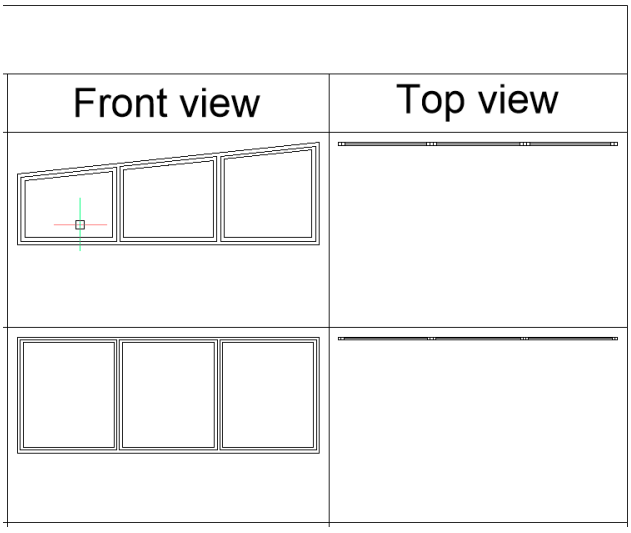
Uwaga: The Wizard automatically selects all entities in the drawing.

- Page 3



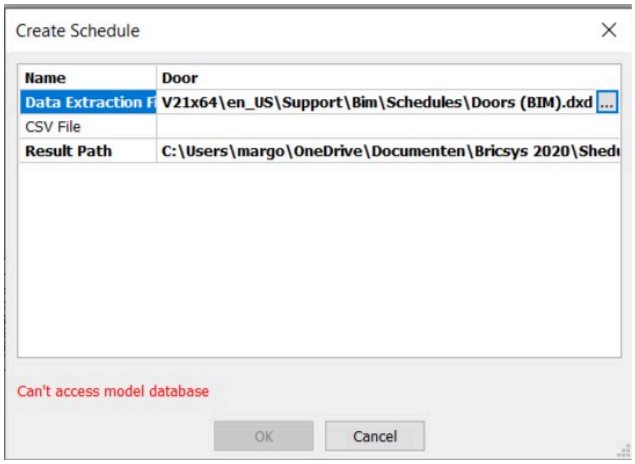
(12) Select the properties to be included in the schedule: click the checkbox to check/uncheck a property: selected (☑) or not selected (☐).
(**13**) Click on the **Category** column to change how the property is shown in the schedule.
(**14**) To include plan and elevation symbols, click on the **Add block view...** button.





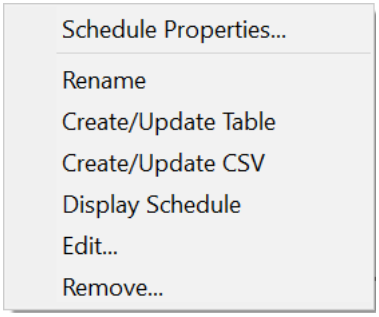
To sort the list, click the title in the column heading. Click again to sort in reverse order.
Click **Finish** to close the wizard (or click **Back** to switch back to page 2).

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

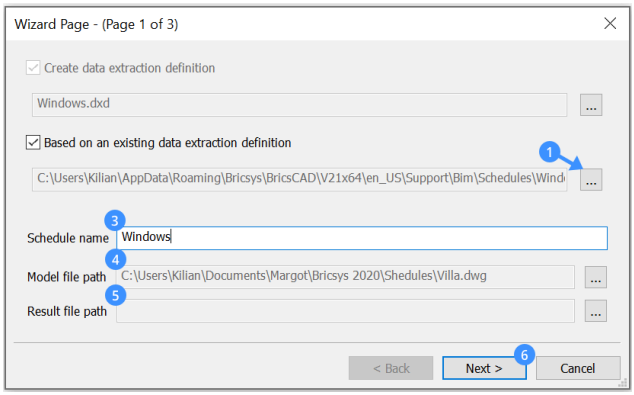
1 Open the BIM **Project Browser**.

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.

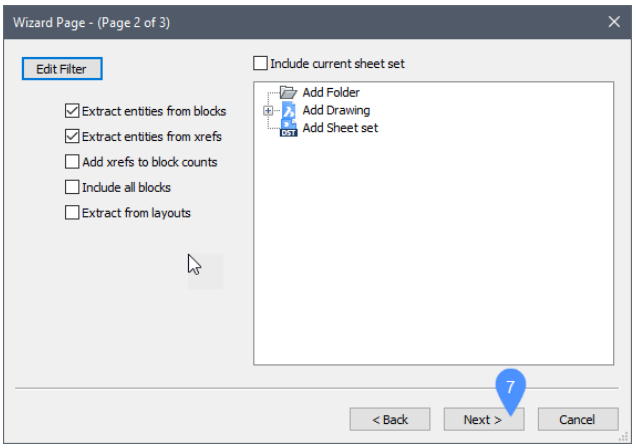
b In the folder, select *'Windows.dxd'*.

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).

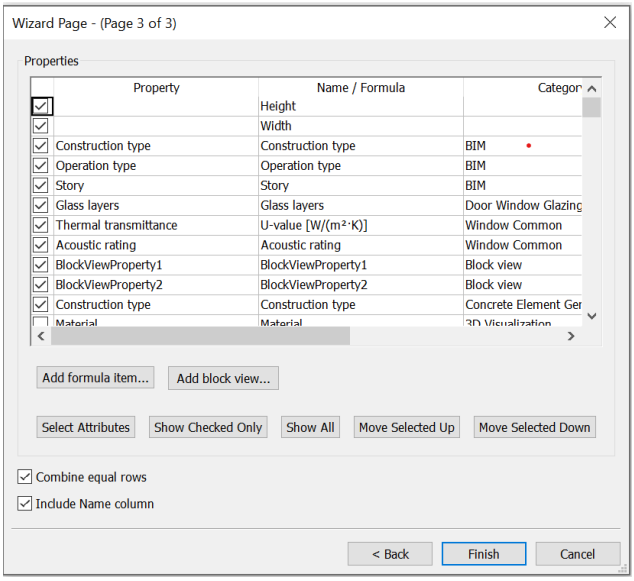
- Page 2



a The wizard automatically selects all the entities in the model.

b Click **Next** (7).

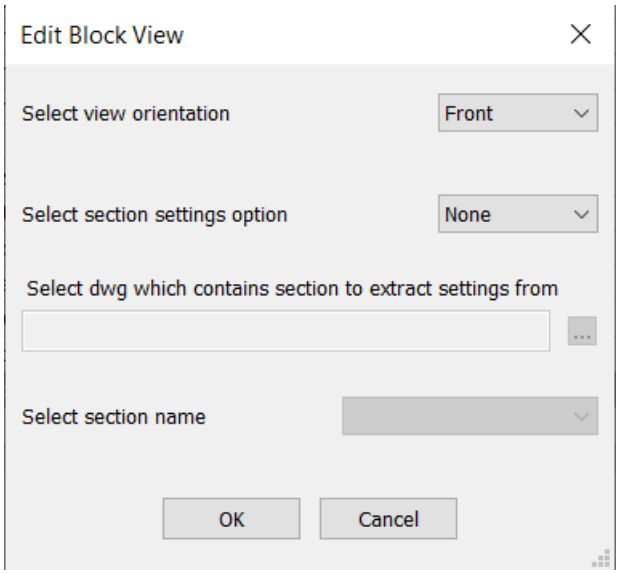
- Page 3



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**.

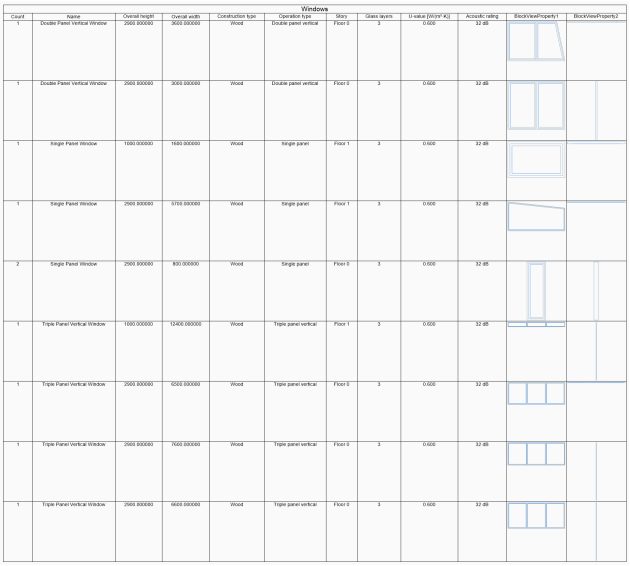
Step 3: open the schedule

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:



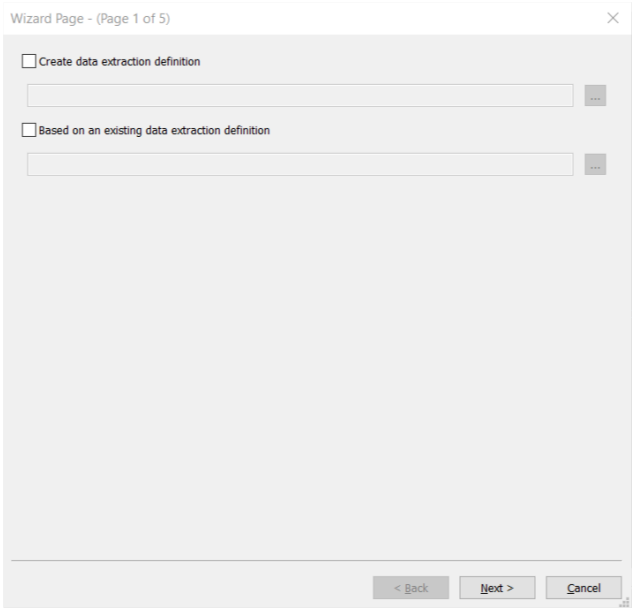
20.7.15.5 Procedure: how to create and edit your own data extraction file: furnishing elements

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:

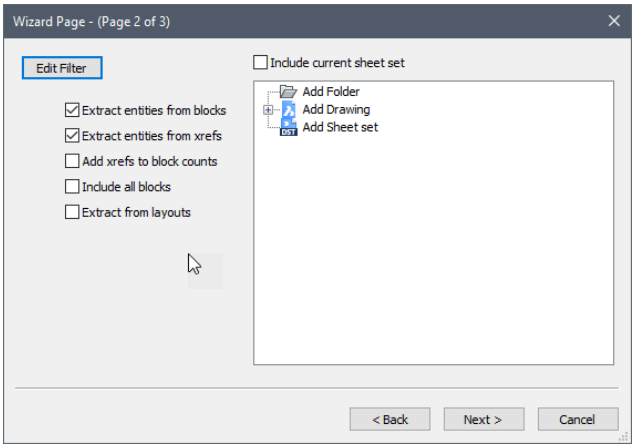


4 Check the **Create data extraction definition** box and choose a file. Name it “FurnishingElements.dxd” and save it in your project folder.

Uwaga: 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.

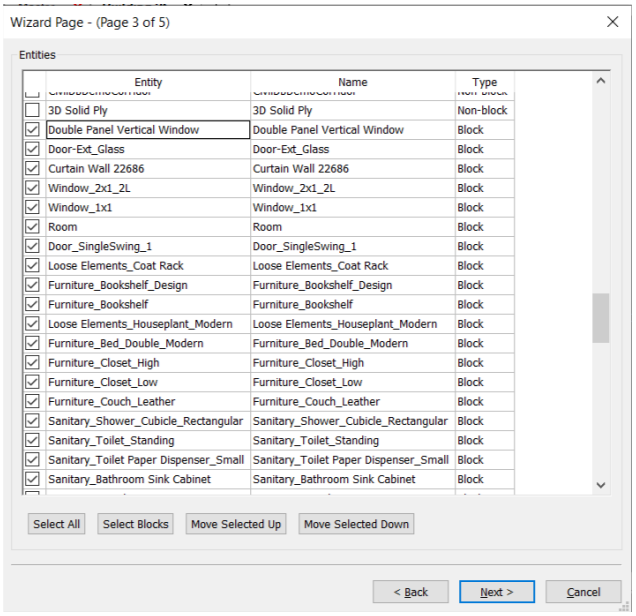
5 Click **Next**.

6 Select all entities you want to include in your schedule.



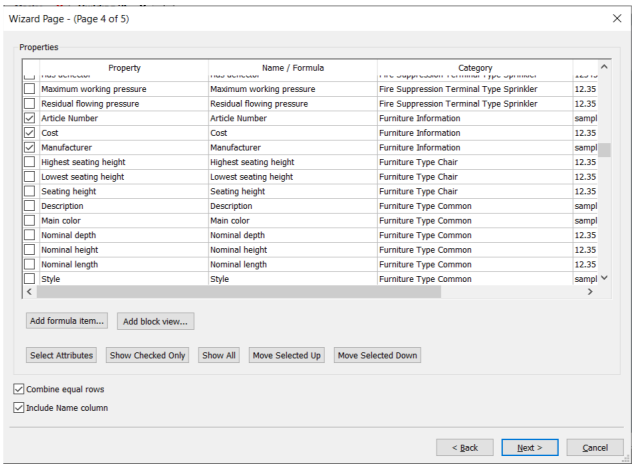
7 Click **Next**.

8 Select all entities you want to have a data extraction of.

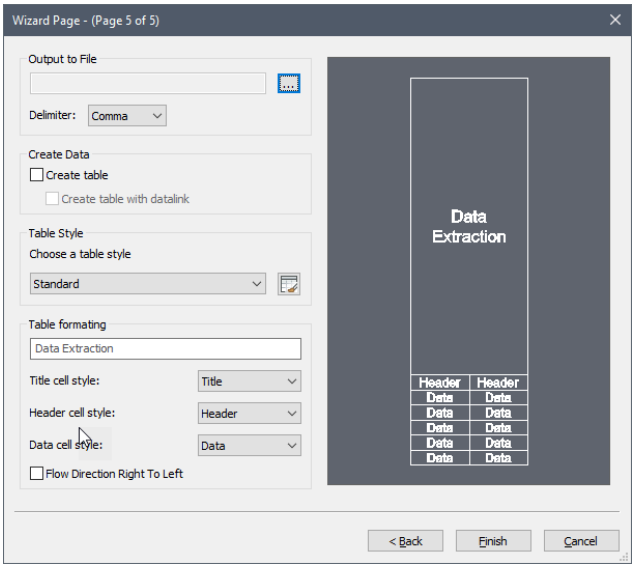


9 Click **Next**.

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]

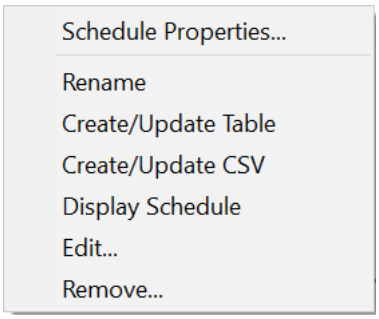
(Type==Furnishing Element)

25 Save this .dxd file in your project folder.

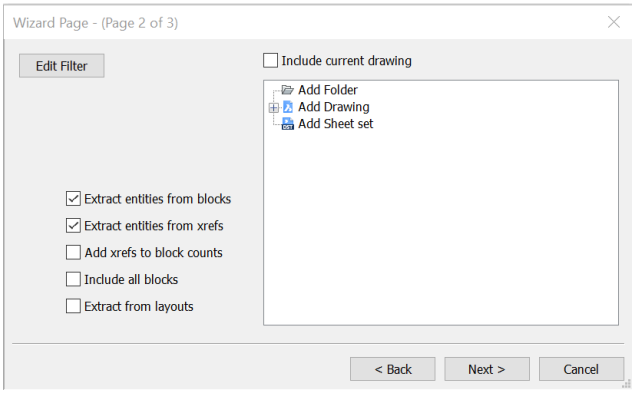
20.7.15.6 Procedure: how to edit a schedule

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]

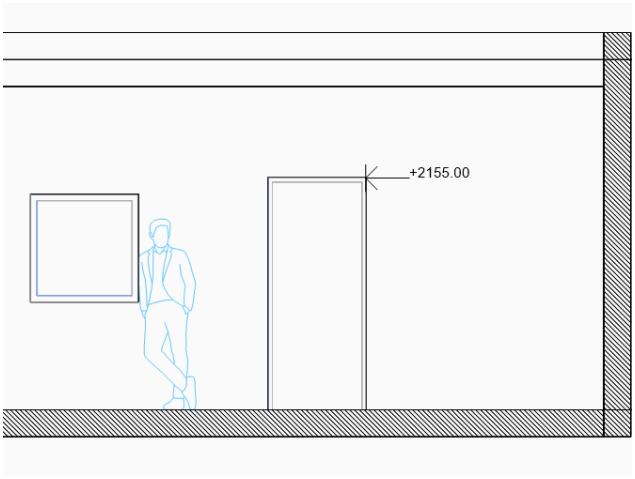
Type **S**to choose **Spot-elevation** and press Enter.

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.

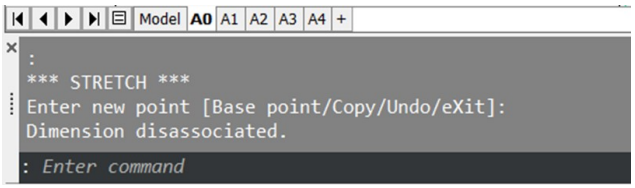
Uwaga: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

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.

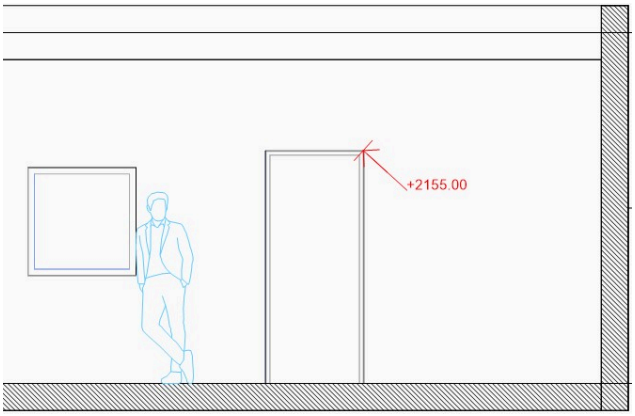


Uwaga: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value becomes dissociated in the Command line.

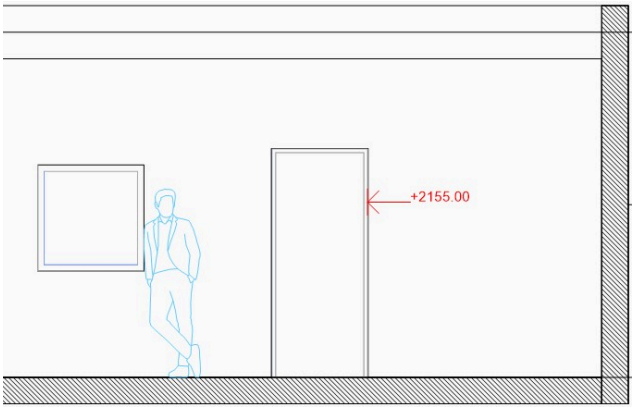


9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

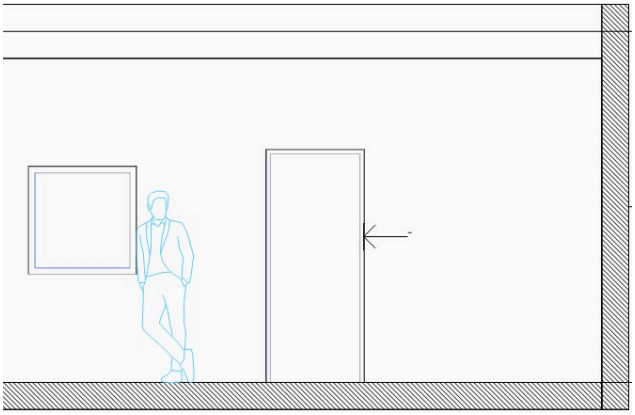
Uwaga:If the tag has been displaced, it may behave irrationally or haphazardly by either turning red or omitting its value, as illustrated below:



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.



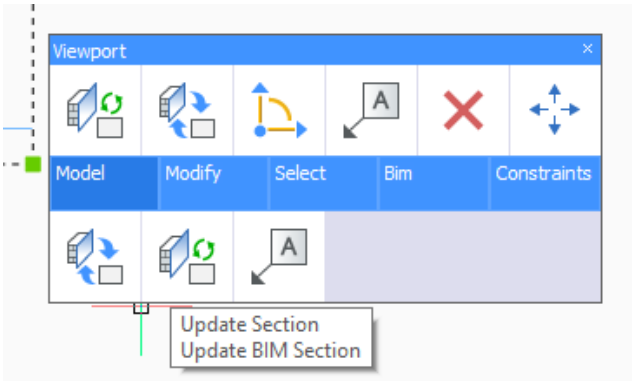
Tag value disappeared after displacing the tag and updating the section.

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.

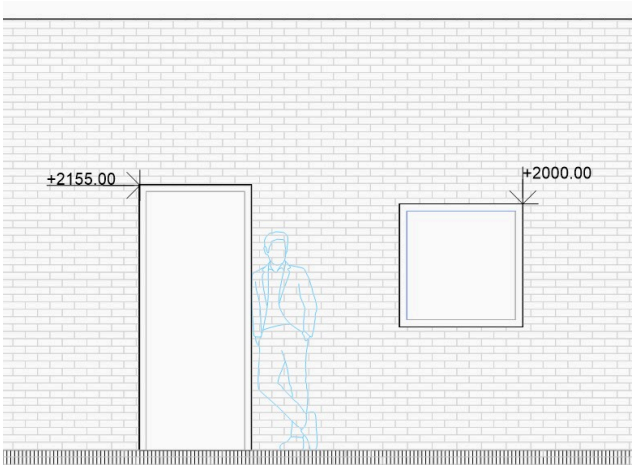
7 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the absolute elevation level measured relative to Floor 0 of the model.

Uwaga: 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.

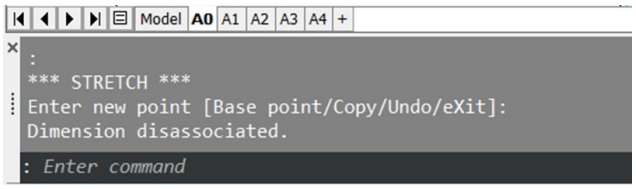
8 When there are several tags possible on the location where you place the tag, hit CTRL to scroll

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.

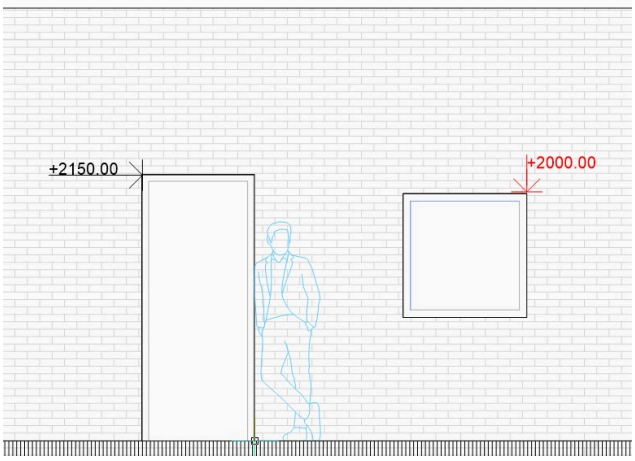


Uwaga: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value is dissociated in the Command line.

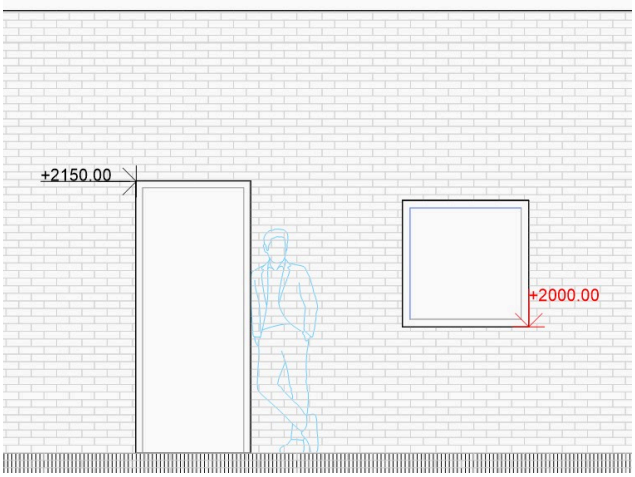


10 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

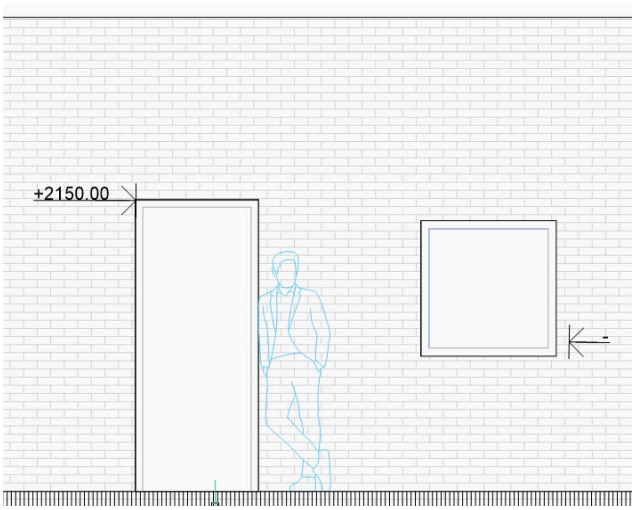
The tag should not behave irrationally or haphazardly by either turning red or omitting its value when it's displaced.



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

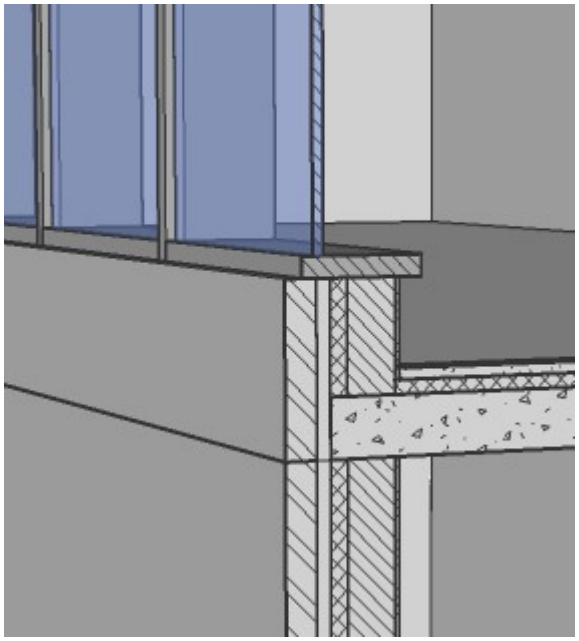
1 Open a floor plan or a reflected ceiling plan from the **Project Browser**.

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**

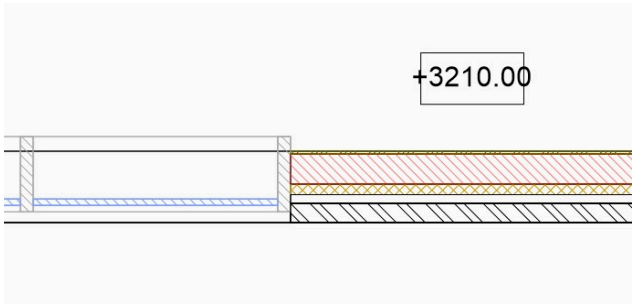
  This annotation tag type, which illustrates with a leader line, is also normally used in the section or elevation sheets. You can use this tag on entities that do not reach or aligned to the floor/ceiling level, such as a window sill in the picture below.

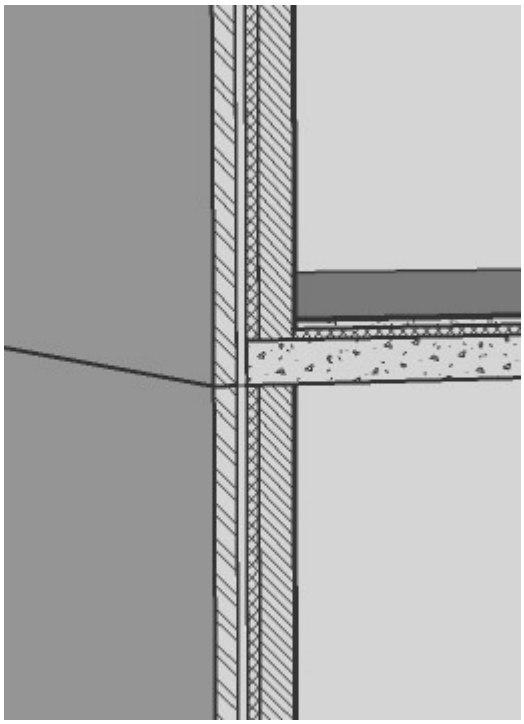


Spot elevation level tag on window sill level which is not on floor level.

- **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.



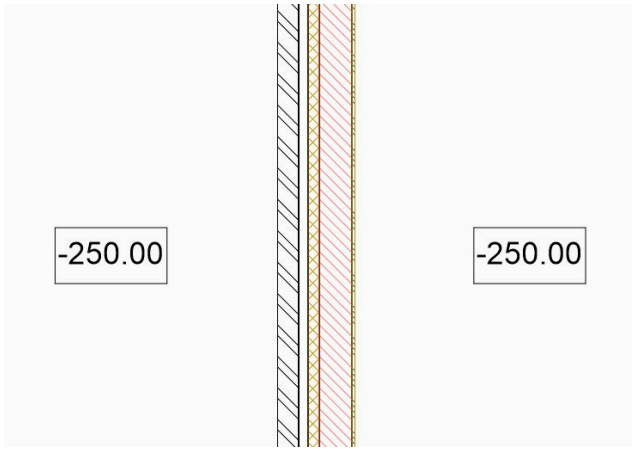


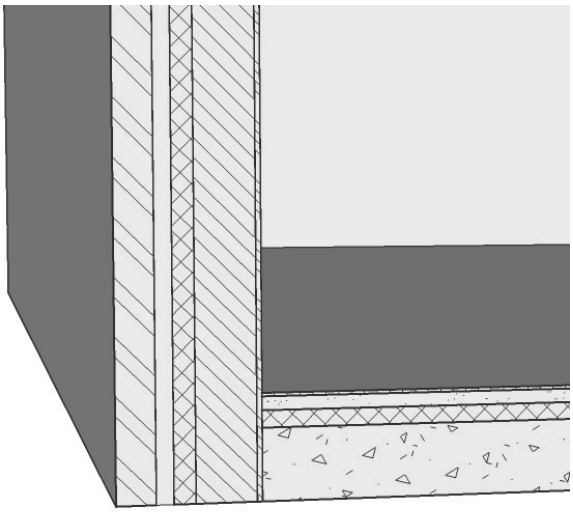
Spot elevation level tag on wall that reaches till the floor slab.

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.

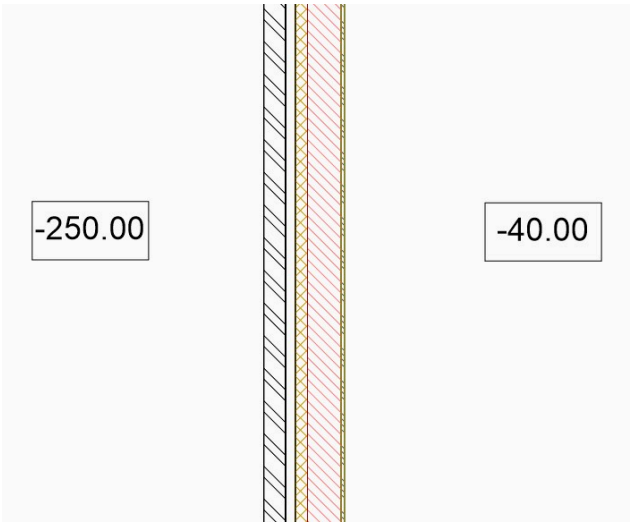


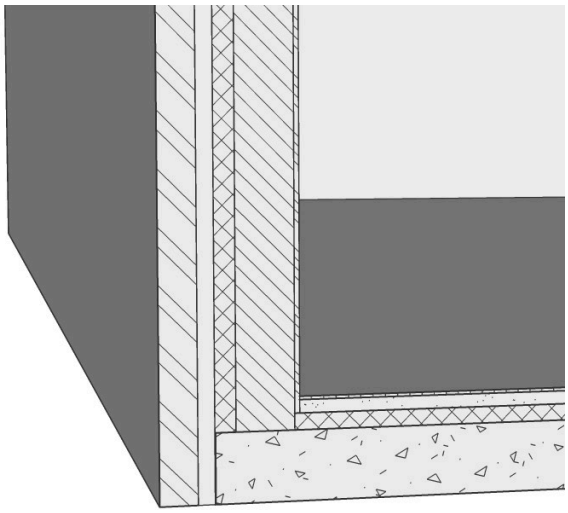


Uwaga: The value is -250 instead of 0 because Floor 0 is defined as the level of the upper side of the ground floor in this model.

- **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.

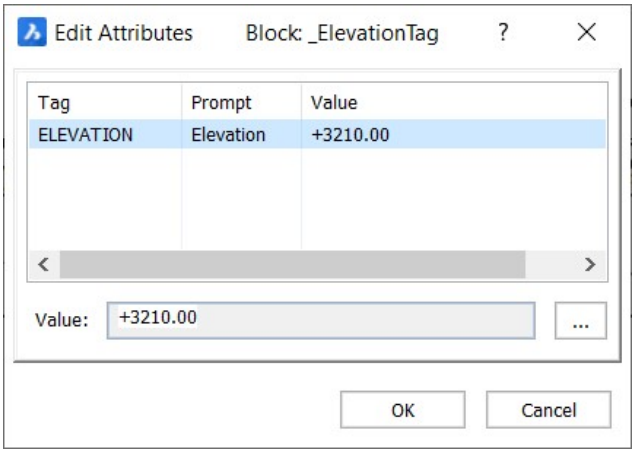




Elevation level of interior ply is higher because ply does not reach till the base level of the floor.

20.7.16.6 Procedure: adjusting the layout of the tag

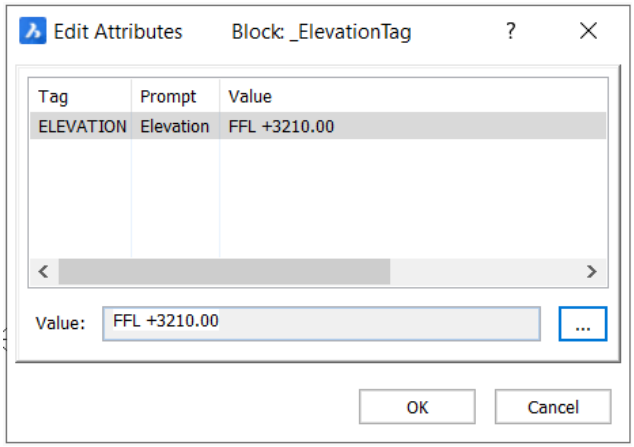
1 Double-click on the value of the spot elevation tag to adjust the layout. A dialog box will pop-up where you can edit the attributes.

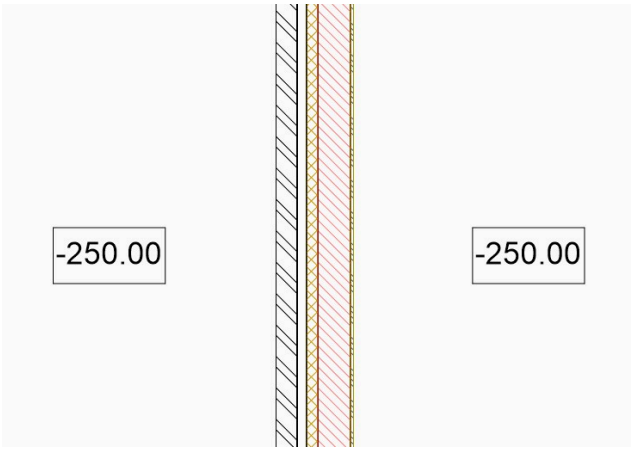


2 Click on the three dots to open the **Text Formatting** dialog box.



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.





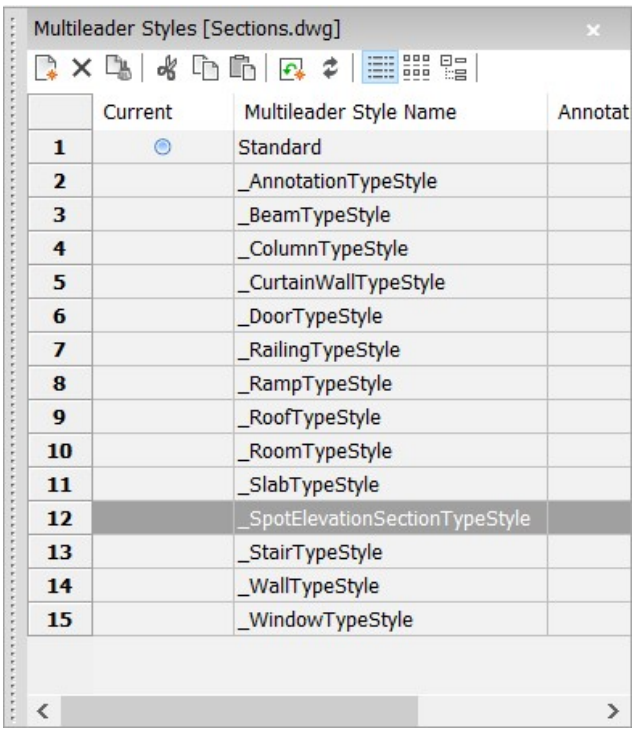
4 If the prefixes and/or suffixes are commonly used, you can create or edit the tag blocks so you do not have to repeat Step 3. For more info, visit the command article BEDIT.

5 For a detailed explanation about attributes, visit the command article MText.

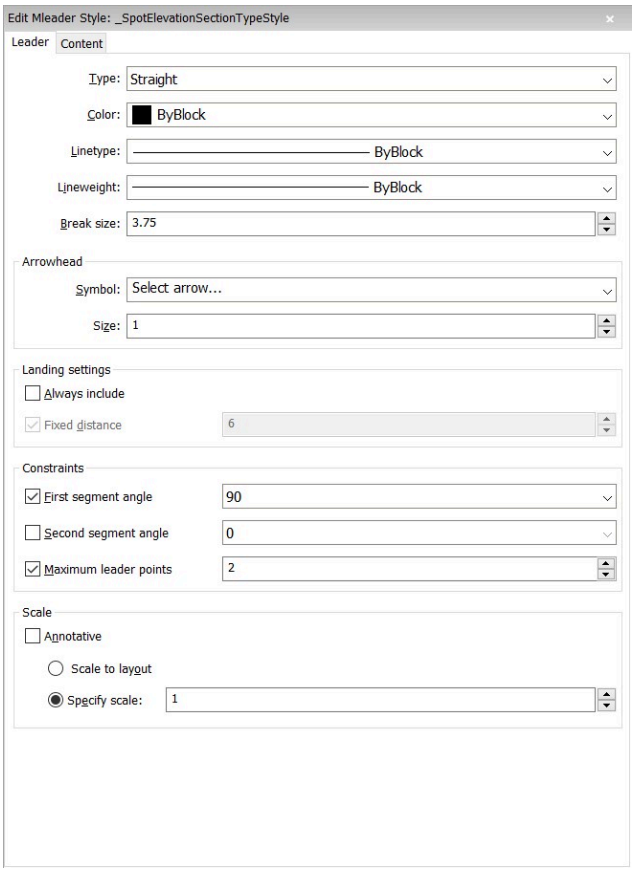
20.7.16.7 Procedure: adjusting the layout of the leader

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.

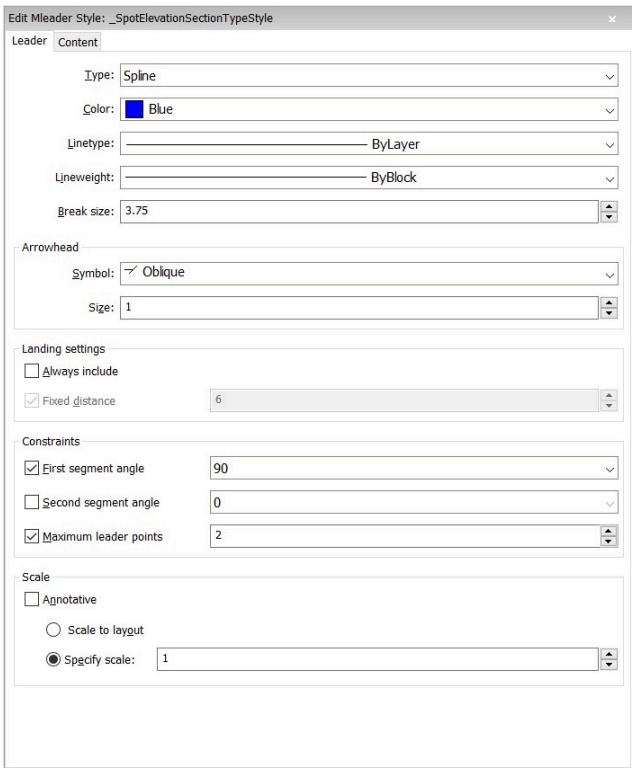


3 You can edit the settings of the leader in the **Edit MLeader Style** dialog box. For more info about the MLeader style settings, visit the article MLeaderStyle.

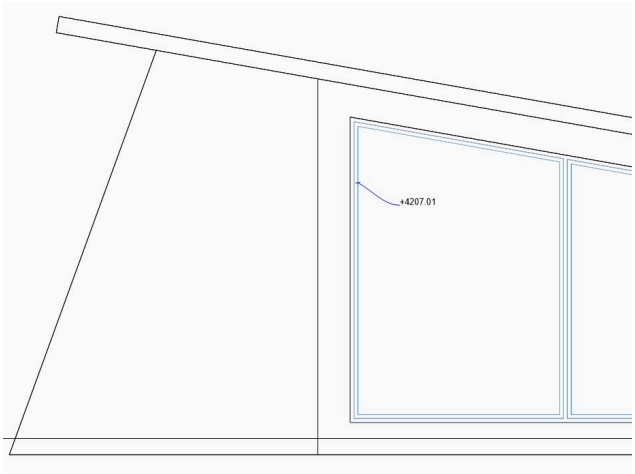


4 After adjusting the settings, click on regen, close the **Drawing Explorer** and update the section to load the new settings.

Uwaga: 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.

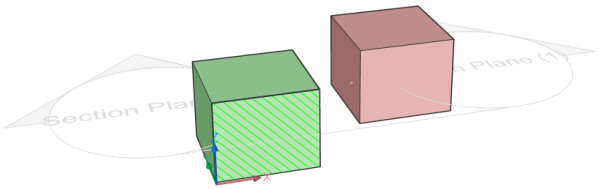
20.7.17.2 Procedure: understanding the created layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

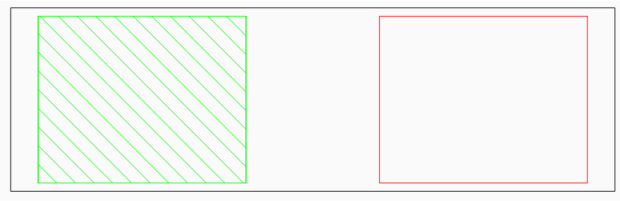
2 Draw two boxes of 500x500x400, one being a bit behind the other.

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.

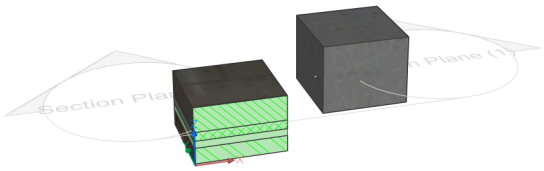
10 Drag the **Cavity Wall, Brick** composition from the **Compositions** panel onto the green box.

Uwaga: Set the LEVELOFDETAIL system variable to 1 or 2, to display the ply materials.

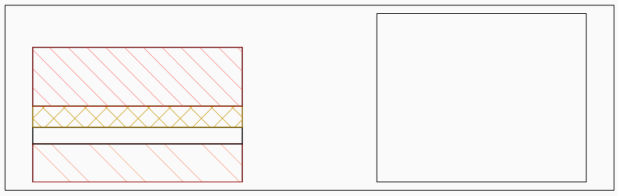
11 Double-click the **Cavity Wall, Brick** composition in the **Compositions** panel.

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.

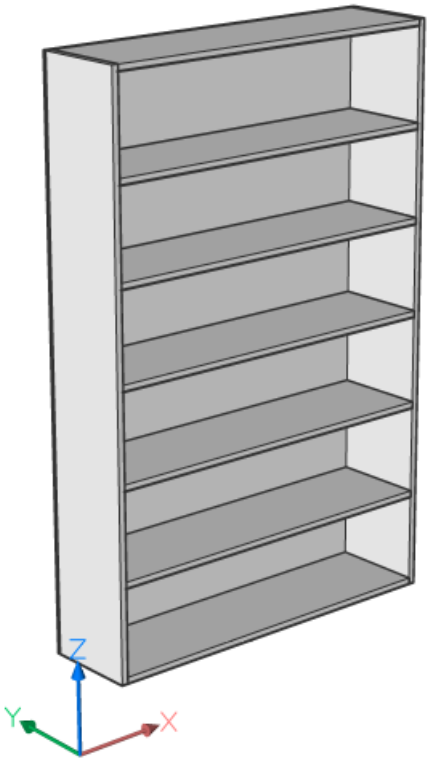
16 Update the section.

17 The outline of the second box is now hatched and the layer **Elevation_Concrete_Reinforced** is added.

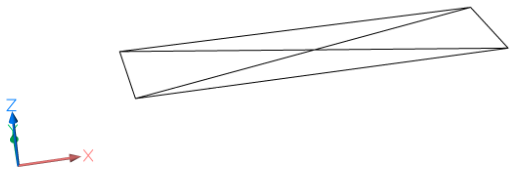
20.7.17.3 Procedure: understanding the BRX_2D layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

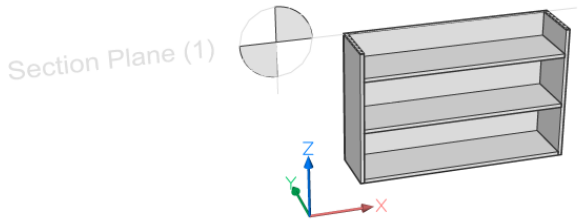
2 Drag the component **Furniture Bookshelf** into the drawing. You can find the component in the **Furnishing Elements** category in the **Library** panel (LIBRARYPANELOPEN command).



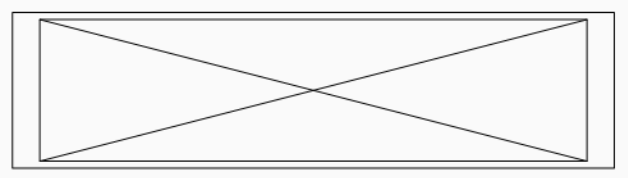
3 If you now show and isolate the **BRX_2D_Symbols** layer, you will see a symbol drawn at the insertion point of the bookshelf.



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.

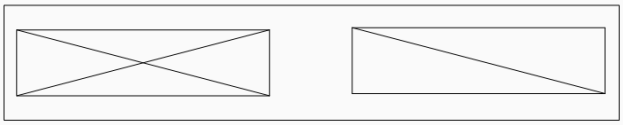
9 Delete one of the lines (of the cross).

10 **(Optional)** Draw extra 2D lines on the **BRX_2D_Symbols** layer.

11 Save the drawing.

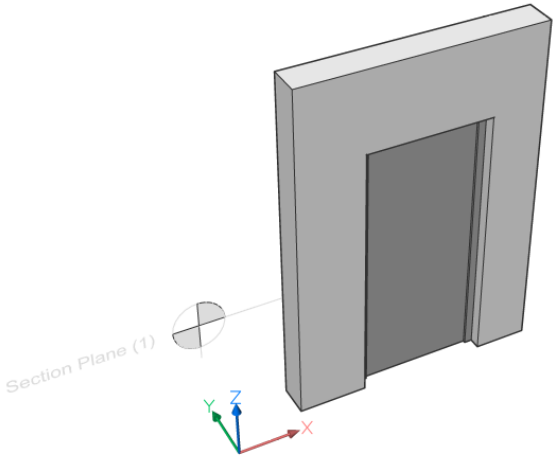
12 Go back to the original 3D model and type in BMINSERT in the Command line and go to the file path where you just saved the new bookshelf. Click **OK** and choose a point to insert the bookshelf.

13 Regenerate the section and open it. Widen the viewport if necessary, by dragging the existing viewport.



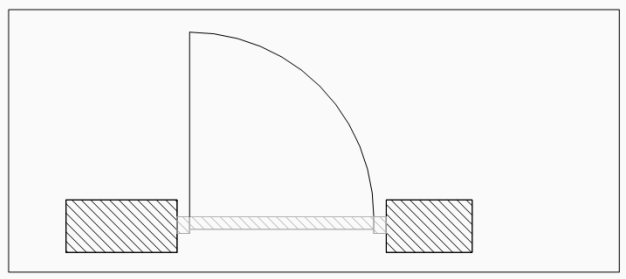
14 You will now see that the second bookshelf has the adapted symbol displayed.

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

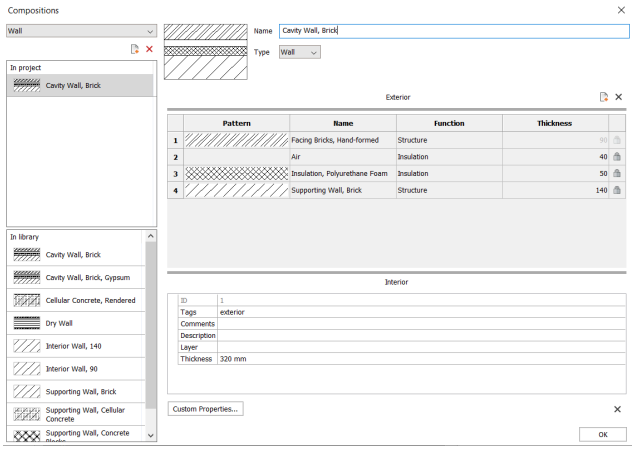
20.7.17.5 Procedure: customizing the hatch patterns

1 Open the drawing made in the above procedure 'Understanding the created layers'.

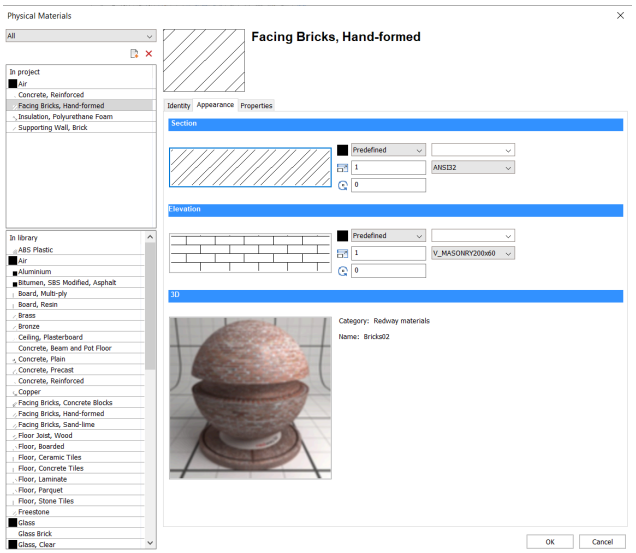
2 Open the **Compositions** panel.

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.



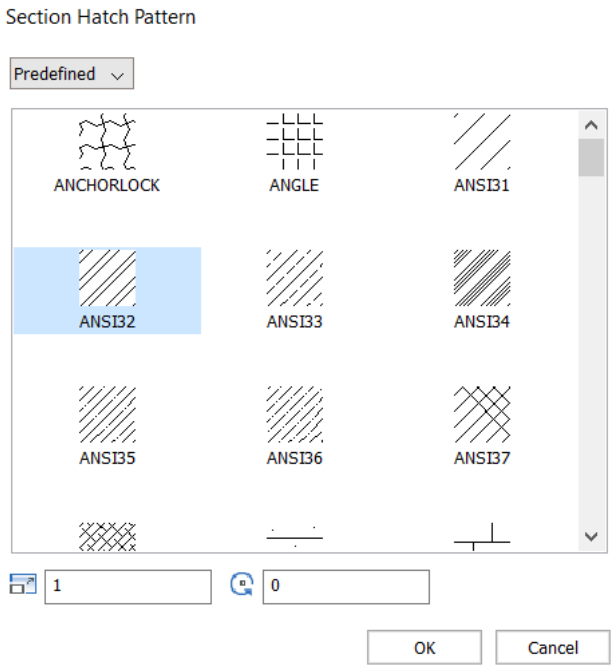
5 Double-click on the row of **Facing Bricks, Hand-formed**. The **Physical Materials** dialog will pop-up on the material **Facing Bricks, Hand-formed**. You could have entered this dialog box by typing in the command BLMATERIALS, after which you could have selected the **Facing Bricks, Hand-formed** 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.



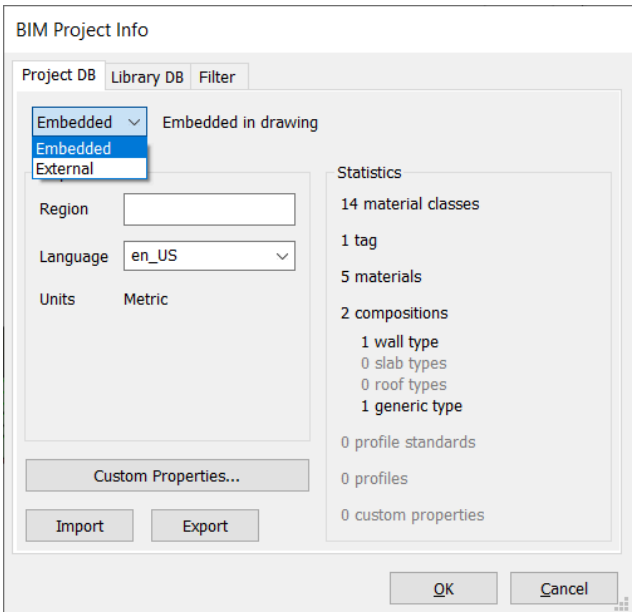
9 If you want to change the appearance of an elevation hatch, you can do so by double-clicking the hatch pattern under the **Elevation** title. The outer most visible material of a composition will be used to determine the elevation hatch.

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.

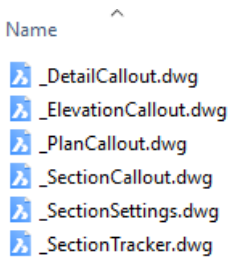
To change the tags in the current drawing, visit the procedure 'Customize tag blocks for the current drawing only'.

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 Co nowego

21.1.1 Przegląd

BricsCAD Mechanical V21 oferuje wszystkie nowe i ulepszone narzędzia dostępne w BricsCAD V21 oraz dodatkowe ulepszenia specjalnie dla mechanicznych przepływów pracy.

Ta sekcja zawiera przegląd wszystkich nowych funkcji i ulepszeń w **narzędziach BricsCAD V21 Mechanical** :

Menedżer BOM na stronie 1530

Biblioteka części na stronie 1533

Parametry i ograniczenia na stronie 1534

Części standardowe na stronie 1534

Arkusz Blachy na stronie 1534

Aby uzyskać informacje o nowych funkcjach i ulepszeniach w narzędziach **BricsCAD V21 Core i Civil** , przejdź do sekcji Co nowego w BricsCAD V21. Zawiera następujące artykuły:

Overview na stronie 31

User interface na stronie 32

Productivity na stronie 35

Drafting na stronie 49

Modeling na stronie 51

Civil tools na stronie 60

Performance and compatibility na stronie 65

Aby uzyskać informacje o nowych funkcjach i ulepszeniach w **BricsCAD V21 BIM**, przejdź do sekcji Co nowego w BricsCAD BIM V21. Zawiera następujące artykuły:

Overview na stronie 958

Modeling techniques na stronie 959

Building data na stronie 966

Project collaboration na stronie 975

Design documentation na stronie 976

Point clouds na stronie 985

21.1.1.1 Informacje o wersji

Kliknij tutaj przegląd informacji o wydaniu.

21.1.1.2 Zbieranie danych diagnostycznych i dotyczących użytkowania

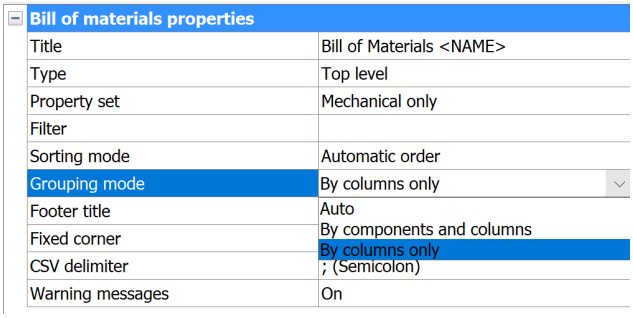
Wiele ulepszeń zostało zainspirowanych prośbami i sugestiami ze strony użytkowników BricsCAD. W Bricsys nieustannie rozwijamy nasze metody gromadzenia opinii użytkowników o nowe narzędzia do zbierania danych. Gdy uruchomisz BricsCAD po raz drugi, pojawi się okno dialogowe z zaproszeniem do wzięcia udziału w Program gromadzenia danych diagnostycznych i użytkowych. Proces zbierania danych umożliwia udostępnianie anonimowych danych diagnostycznych i użytkowych Bricsys, aby pomóc ulepszyć BricsCAD. W każdej chwili możesz wycofać swoją zgodę i usunąć swoje dane z naszych baz danych.

21.1.2 Menedżer BOM

Tworzenie i edytowanie zestawień materiałów jest łatwiejsze niż kiedykolwiek.

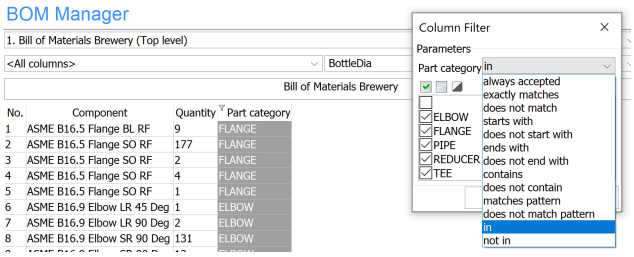
21.1.2.1 Grupowanie

Dzięki dodaniu właściwości grupowania można teraz szybko grupować zawartość tabeli zestawienia komponentów w skondensowane wiersze według wspólnych właściwości istniejących kolumn.

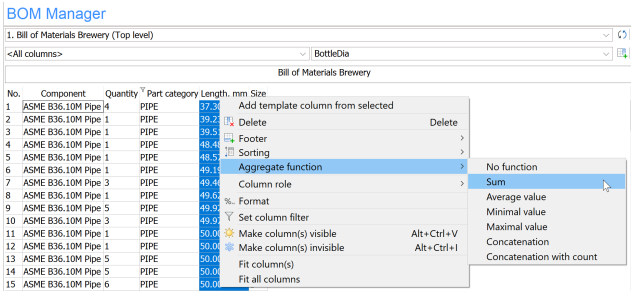


21.1.2.2 Kolumny

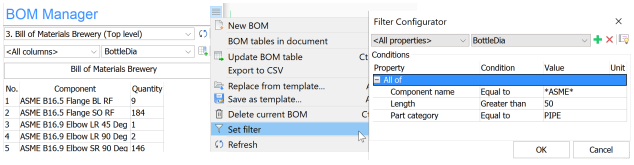
Obecnie można ustawić jeden z kilku wstępnie zdefiniowanych filtrów kolumn dla dowolnej kolumny w zestawieniu komponentów.



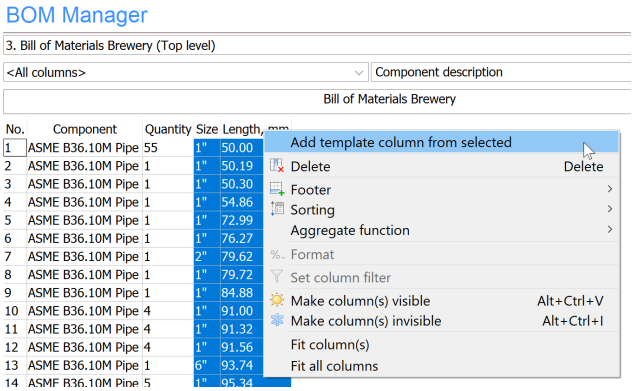
Zagregowane kolumny są teraz obsługiwane w tabelach BOM najwyższego i najniższego poziomu utworzonych za pomocą polecenia BMBOM. Pozwalają użytkownikom obliczyć sumę, minimalną, maksymalną lub średnią wartość danej nieruchomości w odniesieniu do aktualnego zgrupowania.



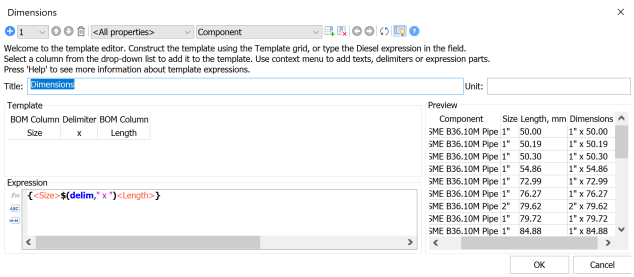
Opcja w menu głównym Menedżera zestawienia komponentów otwiera nowe okno dialogowe Konfigurator filtrów. W Konfiguratorze filtrów możesz przeglądać zawartość swojej tabeli BOM; i wyrażenia wieloznaczne są obsługiwane dla właściwości ciągów w filtrze BOM.



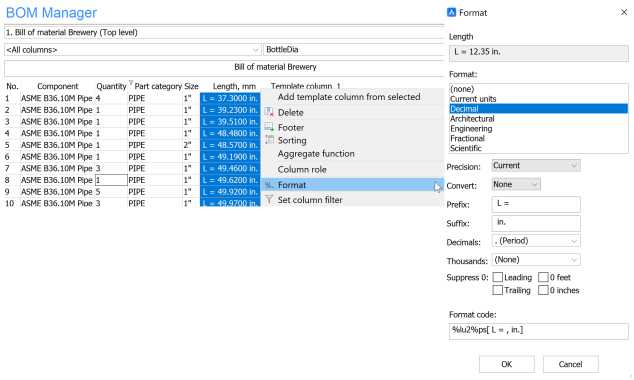
Kolumny szablonu to nowy typ kolumn ocenianych, który umożliwia łączenie wartości kilku właściwości w jedną kolumnę zestawienia komponentów, jako tekst, przy użyciu wyrażeń języka DIESEL.



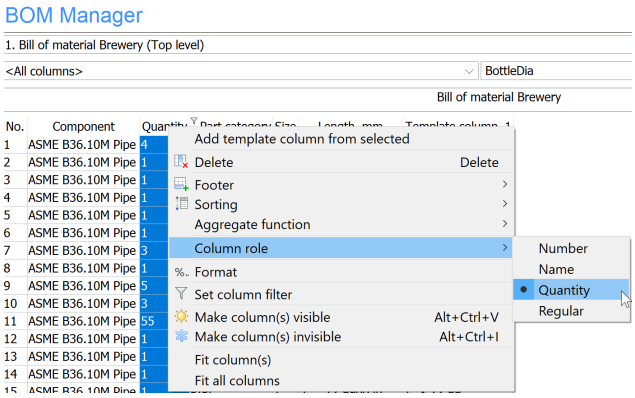
Za pomocą okna dialogowego edytora wyrażeń można konfigurować i wyświetlać podgląd wyrażeń kolumn z formuł lub szablonów.



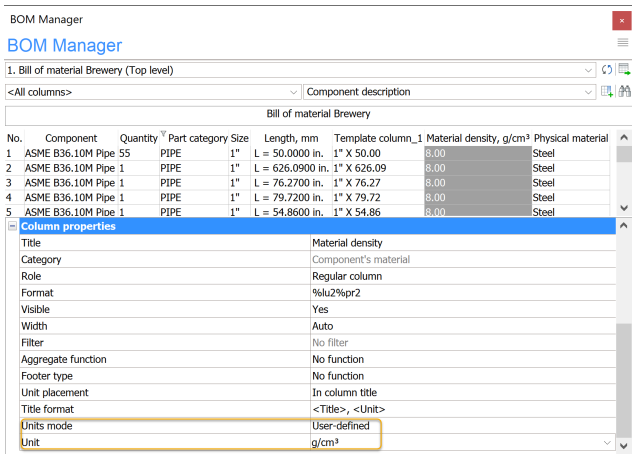
Dzięki dodaniu okna dialogowego Format można teraz dostosować wyrażanie danych w tabelach zestawienia komponentów za pomocą przedrostków, przyrostków i ograniczników.



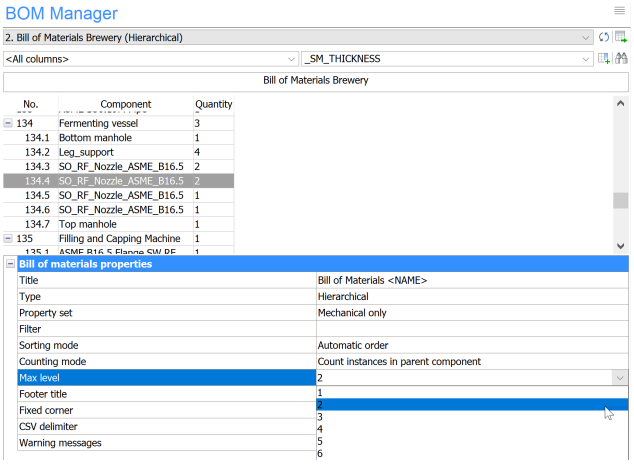
Nowa opcja Rola kolumny umożliwia określenie, która kolumna ma być źródłem odnośników dla widoków rysunków.



Można teraz wybrać jednostkę gęstości materiału, zarówno w oknie dialogowym Materiały fizyczne, jak iw tabelach zestawienia komponentów w sekcji właściwości kolumny. Obsługiwane jednostki to: kg / m³, kg / dm³, g / cm³, lbs / ft³ i lbs / in³.

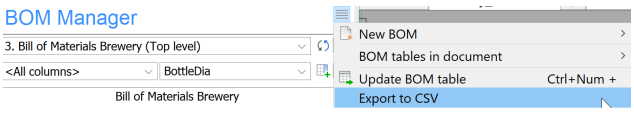


Wprowadzono opcję ograniczenia zgłaszanego poziomu hierarchicznego BOM.



21.1.2.3 Eksportowanie

Możesz teraz wyeksportować swoją tabelę BOM do pliku CSV.

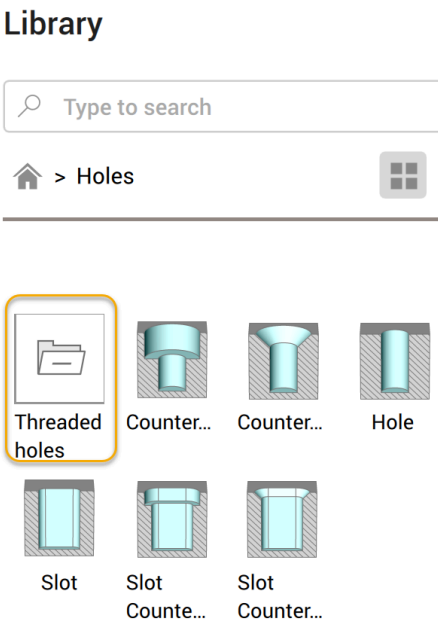


21.1.3 Biblioteka części

Wprowadzenie BricsCAD V21 spowodowało znaczny wzrost liczby standardowych części dostępnych dla Ciebie.

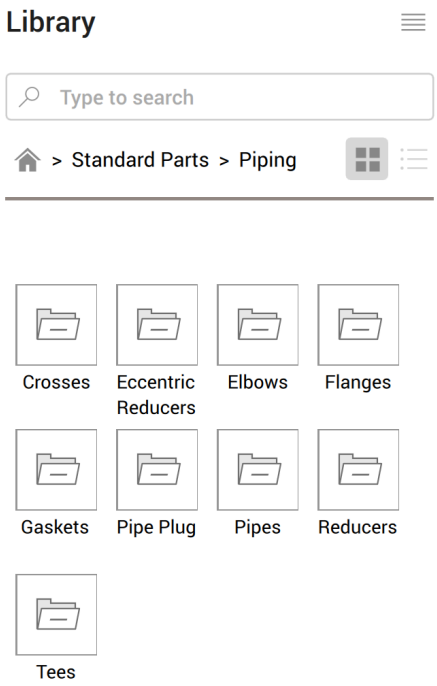
21.1.3.1 Otwory

Dodano bibliotekę komponentów z otworami gwintowanymi.



21.1.3.2 Rura

Dodano bibliotekę elementów rurociągów.



21.1.4 Parametry i ograniczenia

21.1.4.1 Rurociągi

Wraz z wydaniem znormalizowanych części rurociągów, nowe polecenie BMCONNECT automatycznie ograniczy wybrane elementy w oparciu o ich bliskość do siebie.

21.1.5 Części standardowe

Wiązania są teraz automatycznie przywracane po zmianie parametrów części znormalizowanej.

21.1.6 Arkusz Blachy

Narzędzia arkusza blachy w BricsCAD Mechanical V21 oferują większą elastyczność i szybsze przepływy pracy.

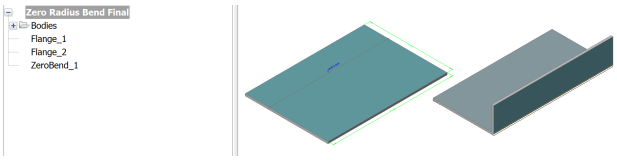
Na przykład, do każdej bryły można przypisać określoną tabelę gięcia, a w tym samym bloku mogą istnieć sztuki o różnej grubości.

21.1.6.1 Fazy

Nowo dodana funkcja Bevel jest obsługiwana w poleceniu SMCONVERT, SMREPAIR i SMUNFOLD.

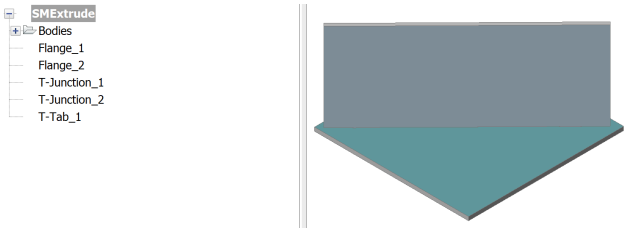
21.1.6.2 Zero promienia zgięć

Nowa funkcja ZeroBend pozwala na tworzenie gięcia bez wewnętrznego promienia dla elementów z blachy.



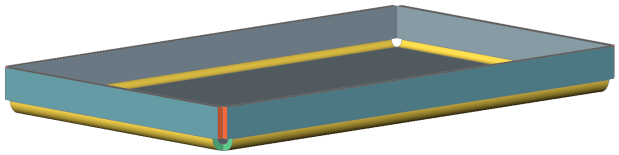
21.1.6.3 Złącze T

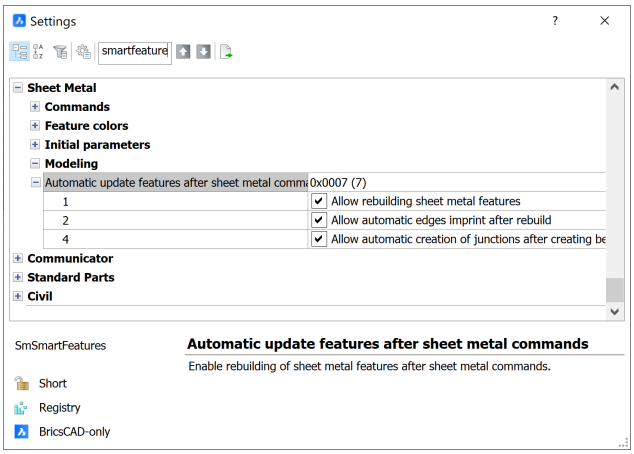
Nowe połączenie T-Junction można utworzyć za pomocą SMEXTRUDE na pojedynczej linii poli-segmentowej leżącej na kołnierzu, aby utworzyć nowy kołnierz ortogonalny do istniejącego. Korzystając z funkcji T-Junction, możemy następnie przekształcić go w T-Tab, który automatycznie doda wypustki do ortogonalnego kołnierza i usunie niezbędny materiał z oryginału.



21.1.6.4 Zaktualizowane funkcje inteligentne

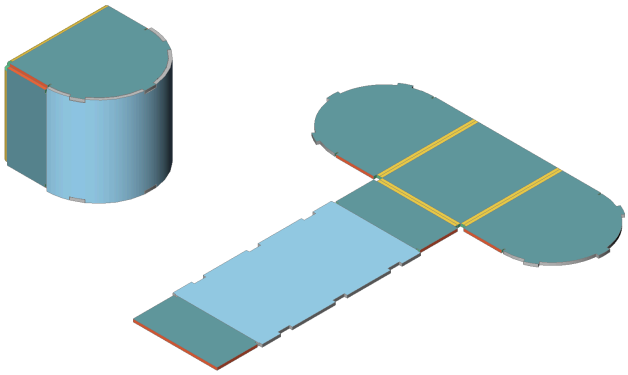
Możesz teraz tworzyć rysunki arkusza blachy jeszcze szybciej, wykorzystując uaktualnione polecenie SMSMARTFEATURE, które umożliwia automatyczne tworzenie połączeń podczas korzystania z narzędzia do gięcia.



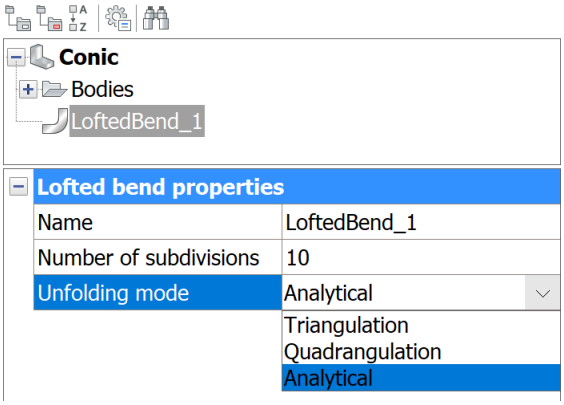


21.1.6.5 Tabulatory

SMTABCREATE Polecenie ma teraz opcję tworzenia zakrzywionej zakładki na zakrzywionej twardej krawędzi, łączącej kołnierz i zagięcie, w tym zagięcia wyciągnięte po profilach.

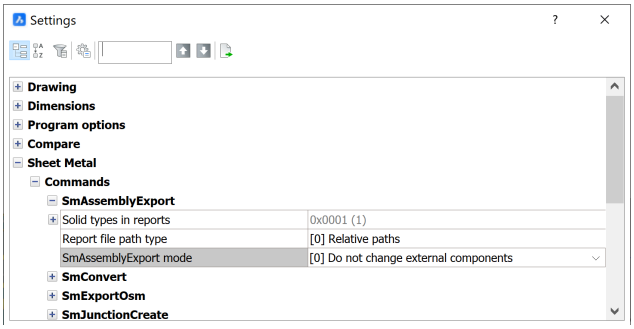


Polecenie SMUNFOLD oferuje ulepszone zachowanie dla stożkowych wyciągniętych gięć. Można je teraz rozłożyć analitycznie, aby uniknąć podzielenia ich na określoną liczbę zagięć i kołnierzy.



21.1.6.6 Eksport zespołu

Zmienna systemowa SMASSEMBLYEXPORTMODE umożliwia zachowanie rozpoznanych operacji dla arkusza blachy i słabo zamodelowanych części arkusza blachy podczas korzystania z polecenia EKSPORTUJ WYSM.



21.2 Rozpocznij pracę

21.2.1 Mechaniczny obszar roboczy

21.2.1.1 Rozpocznij projekt konstrukcji mechanicznej

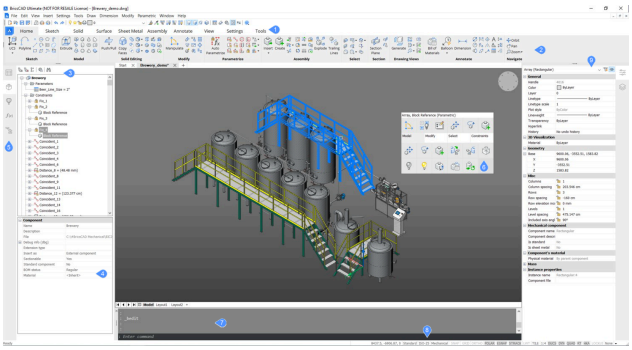
Możesz rozpocząć projekt mechaniczny:

1 Uruchom BricsCAD.

Launcher BricsCAD pojawi się okno.

2 Kliknij **Początek** w sekcji **Konstrukcja mechaniczna** .

21.2.1.2 Okno aplikacji BricsCAD mechanical



1 **Karty wstążki** : kliknij, aby przejść do określonej karty.

2 **Panel wstążki** : każda zakładka zawiera kilka paneli, które łączą w sobie podobne narzędzia.

3 **Przeglądarka Mechanical** : przeglądaj drzewiastą reprezentację swojego modelu z możliwością podświetlenia określonego węzła w obszarze modelu i edycji jego właściwości mechanicznych.

4 **Właściwości mechaniczne** konkretnego węzła: niektóre są tylko do odczytu. Sprawdź właściwości węzła głównego w drzewie.

5 **Dokowalne panele narzędzi** : Przeglądarka i biblioteka mechanical.

6 **Menu z czterema kursorami** : zastosuj określone narzędzie po najechaniu kursorem na element, na który ma ono wpływ.

7 **Wiersz poleceń** : wpisz polecenie.

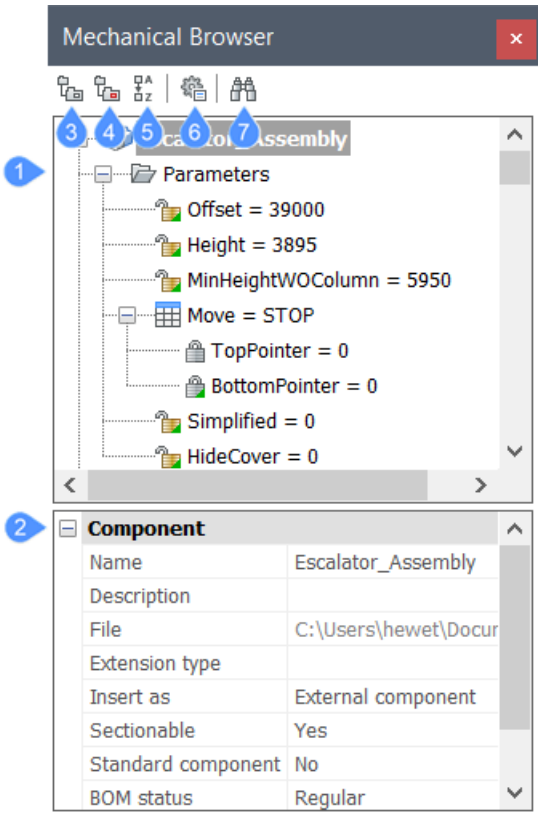
8 **Pasek stanu**: status i przełączane opcje.

9 **Panel Właściwości** : dostosuj niemechaniczne właściwości wybranych elementów; takie jak kolor lub warstwa.

Rysunek 4: Nawigacja w 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.

Uwaga: 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.

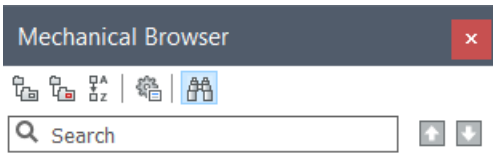
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

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 3D comparison

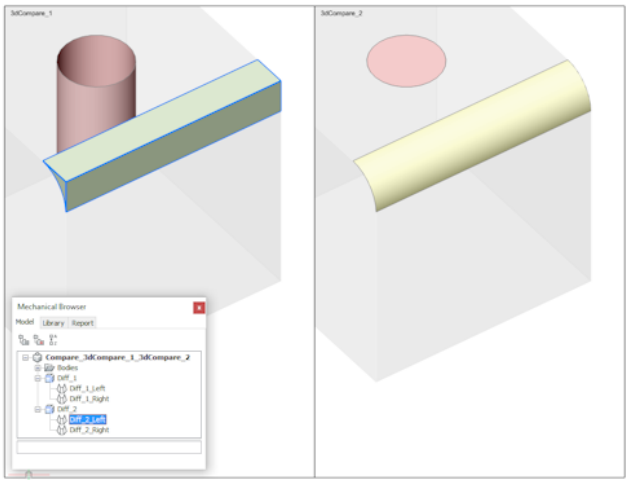
Command: *3DCOMPARE*.

You can compare two 3D models, for example 3D solids and surfaces stored in 2 DWG files, including models imported in BricsCAD from other file formats. The tool can process large models and reports the differences in the **Mechanical Browser**. Multiple visualization modes are supported.

21.2.3.1 Definitions

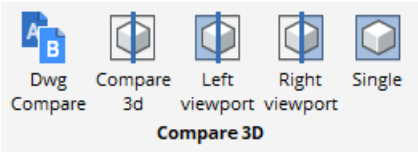
A 3D model is a DWG file with one or more 3D solids or surfaces. When comparing two 3D models, BricsCAD first tries to automatically align them in 3D space to obtain the maximal similarity. Next, all faces of these 3D solids and surfaces are compared geometrically with a given tolerance. If for every face in the first file an equal face in the second file exists and vice versa, such models are considered as equal. However, if for a particular face in one file there is no equal face found in another file, or if only part of this face is equal to a part of a face in the other file, a difference is detected. Therefore, a difference is a full face or a part of a face of a 3D solid or a surface, for which there are no equal parts of full faces found in the other file. Two or more differences are grouped together if they are geometrically adjacent to each other.

A difference node is created in the **Mechanical Browser** for every group of adjacent differences. If you expand this node, you can see the particular differences it consists of. When you select the difference in the **Mechanical Browser**, the corresponding feature is highlighted in the drawing.



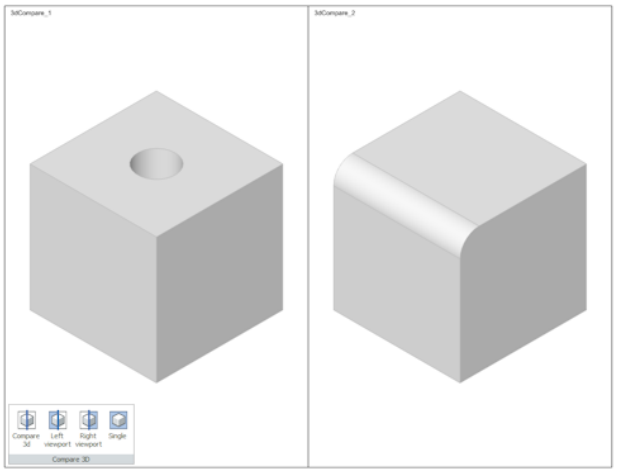
21.2.3.2 Toolbars

You can access the 3D Comparison procedure by using the tools on the **Compare 3D** panel on the **Tools** tab.



21.2.3.3 Visualization

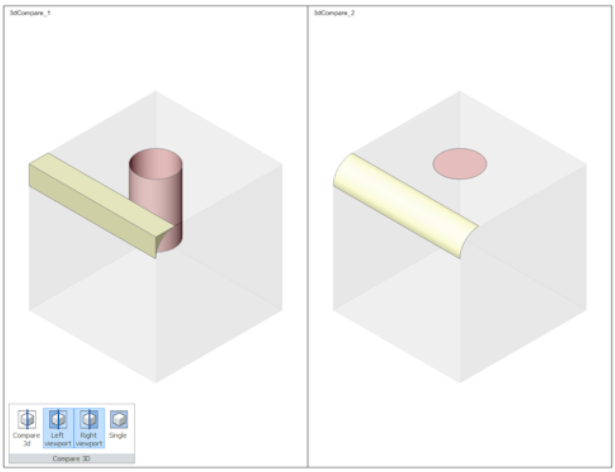
Use the *3DCOMPARE* command to create a new DWG document named **Compare_<File_1>_<File_2>.dwg**. The compared models are attached as external references to this drawing, A special layout named **Comparison** is created automatically with 2 viewports, one for each drawing, illustrating the differences between the drawings by using colors.



The display is controlled via the *3DCOMPAREMODE* system variable. If the value of *3DCOMPAREMODE* is different from 0 (zero), the corresponding model becomes semi-transparent, while the differences are displayed using different colors. Each difference is painted in its individual color. You can zoom in on a particular difference using the Mechanical Browser context menu.

To display the differences:

1 Click the **Left Viewport** tool button ( ) and/or the **Right Viewport** tool button ( ).



2 Double-click in one of the viewports.

3 Use the mouse/keyboard shortcuts to manipulate the display.

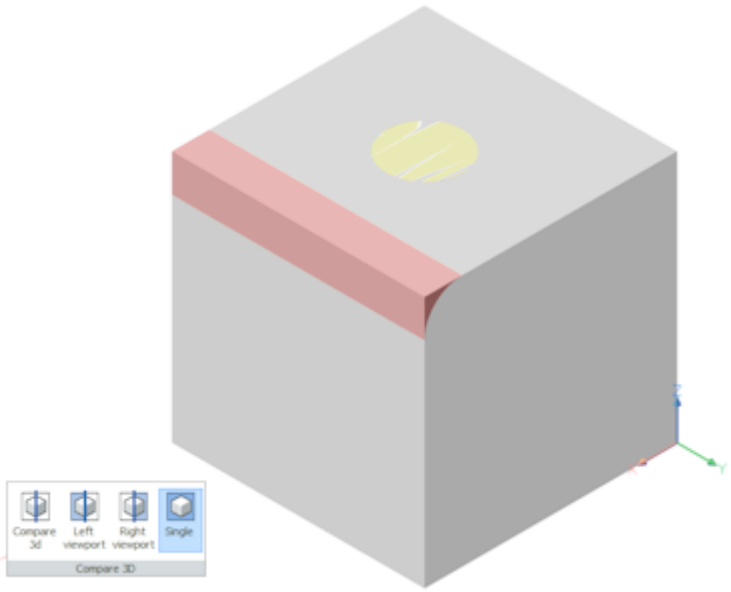
Example: Zoom in/out or rotate.

The display in the other viewport is synchronized automatically. This allows you to compare the same part in both models in detail.

As an alternative, you can:

Click the **Single** tool button ( ).

The differences in model space are displayed.

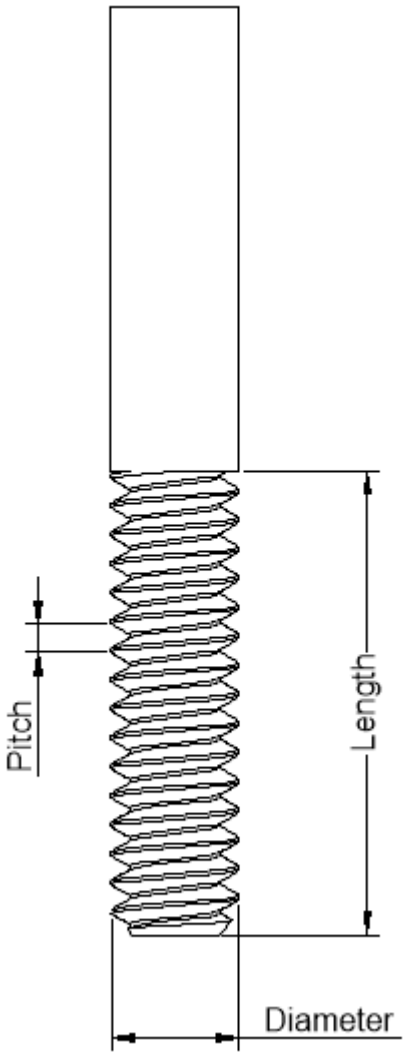


21.3 Projekt części

21.3.1 Funkcja wątku

Użyj polecenia DMTHREAD, aby utworzyć element gwintu.

Poniższy rysunek przedstawia strukturę gwintu:



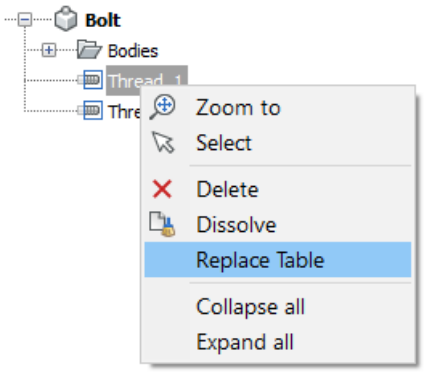
Element gwintu ma następujące parametry:

- **Standard gwintu**

  Jest to zestaw standardowych gwintów. Listę tę można rozszerzyć o niestandardowy gwint. Po zmianie normy geometria zostanie automatycznie przebudowana.
- **Skok**gwintu.
- **Długość** gwintu.
- **Średnica** gwintu.
- **Rodzaj** gwintu - męski lub żeński. Jeśli typ jest ustawiony jako auto, zostanie on określony automatycznie.

21.3.1.1 Dodawanie konfiguracji gwintu

1 Kliknij prawym przyciskiem myszy na element w **Przeglądarce Mechanicznej**.



2 Wybierz **Zastąp Tabelę**.

3 Wybierz plik CSV o następującej konfiguracji:

```
Size,Diameter,Pitch Name1, Diameter_value1, Pitch_value1 Name2,
 Diameter_value2, Pitch_value2 Name3, Diameter_value3, Pitch_value3
```

21.3.2 Metody projektowania części

BricsCAD Mechanical umożliwia tworzenie i modyfikowanie modeli 3D.

Sposoby tworzenia modelu 3D:

- Utwórz prymityw bryłowy 3D, taki jak kostka i walec.
- Szkicowanie profilu 2D i wyciąganie lub obracanie go, aby utworzyć bryłę 3D.
- Modeluj powierzchnię 3D i pogrubiaj ją, aby utworzyć bryłę 3D, lub zszyj kilka powierzchni w celu utworzenia wodoszczelnej bryły 3D.
- Importuj istniejącą geometrię brył i powierzchni 3D z innych programów CAD.

Sposoby edycji i dodawania więcej szczegółów do modeli 3D

- Przez pchanie / ciągnięcie, przesuwanie i obracanie bryły.
- Tworząc zaokrąglenia lub fazowania na krawędziach modelu.
- Poprzez zastosowanie otworów.
- Dodając wypukłości i wgłębienia oraz używając operacji boolowskich, narzędzi do bezpośredniej edycji i modelowania odkształcalnego.

Brak zależności rodzic-dziecko = podejście bez historii

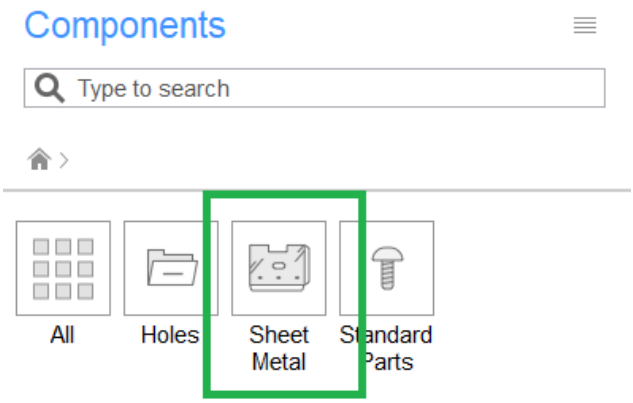
BricsCAD Mechanical nie tworzy zależności rodzic-dziecko między geometrycznymi prymitywami i elementami. Wszystkie zmiany dokonane w Twojej części są lokalne; nie wymuszają regeneracji geometrii. To podejście BEZ historii pozwala skoncentrować się na projekcie zamiast myśleć o tym, jak można go zmienić w przyszłości.

Wiązania 3D = Parametryczne

BricsCAD Mechanical zapewnia potężny zestaw narzędzi do parametryzowania części poprzez dodawanie wiązań 3D do jej powierzchni i krawędzi oraz łączenie ich z parametrami globalnymi za pomocą wyrażeń i tabel projektowych.

21.3.3 Szkicowanie

Zestaw narzędzi do szkicowania umożliwia rysowanie profili 2d, które można wyciągać lub obracać, aby tworzyć bryły i powierzchnie 3D lub dodawać / odejmować objętość do / od istniejących elementów. Narzędzia do szkicowania dostępne są na karcie **Szkic** na wstążce obszaru roboczego **Mechanika:**



Rysunek 5: Mechaniczna wstążka obszaru roboczego

Szkice są tworzone na płaszczyźnie XY bieżącego lokalnego układu współrzędnych (LUW) . Ponieważ dynamiczny LUW jest domyślnie włączony, można tworzyć elementy 2D na powierzchni bryły 3D pod kursorem. Użyj narzędzia Wyrównaj LUW , aby ustawić LUW zgodnie ze swoimi potrzebami.

Narzędzia do rysowania obejmują znane elementy podstawowe 2D, takie jak prostokąt/wielokąt, okrąg/łuk/elipsa, polilinia i splajn.

Użyj narzędzi Przesuń , Kopiuj , Obróć 2D, Lustro 2D i Szyk , aby manipulować szkicami.

Użyj poleceń *ZRÓWNOWAŻYĆ* , *ROZCIĄGAĆ* , *PRZYCINAĆ* , *ROZSZERZYĆ* , *FILET* / *ŚCIĘCIE* , oraz *PRZERWA* aby edytować szkice.

Użyj więzów 2D, aby szkice stały się parametryczne.

> ***Uwaga:*** Wartość *DELOBJ* Zmienna systemowa (Usuń element) kontroluje, czy elementy szkicowania są usuwane, gdy są używane przez *DMEXTRUDE* lub *DMREVOLVE* do tworzenia brył lub powierzchni 3D.

21.3.4 Solid Modeling

BricsCAD Mechanical provides a rich toolset for solid modeling, including traditional 3D modeling tools like Extrude or Subtract and intuitive Direct Editing tools like; Push/Pull, Copy Faces and powerful 3D Constraints, which allow you to adjust the shape and size of your 3D part.

3D solids tools are available on the **Solid** tab of the **Mechanical** workspace ribbon:



Rysunek 6: Solid

Tabela 1: Solid Modeling

Tool/toolset	Description
Extrude	Create 3D solids from sketches.
Revolve	
Loft	
Sweep	
Thicken	
Primitive toolset: Box, Cylinder, Cone, Pyramid, Wedge, Sphere, and Torus	Create 3D solids directly, without underlying sketches.
Push/Pull	Edit 3D solids (applied to faces and/or edges).
Move	
Rotate	
Copy Faces	
Fillet	
Chamfer	

Tabela 1: Solid Modeling

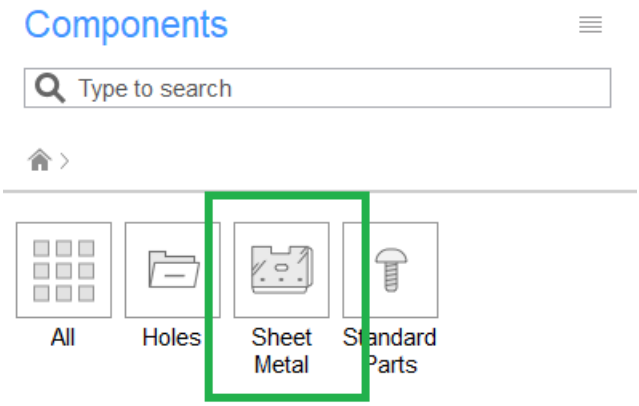
Tool/toolset	Description
Shell	Edit 3D solids.
Slice	
Boolean toolset: Union, Subtract, Intersect, and Separate	
Scale	
Mirror	
Array	
Erase	Can be applied to the whole 3D solid, and to a set of its faces forming a feature (for example a fillet or a hole).
Audit	Find and fix errors in imported geometry.
Simplify	Automatically recognize canonical faces (planar, cylindrical, conical, spherical, toroidal) from spline faces of your 3D solid. This is often needed before applying 3D constraints.
Stitch	Stitch touching surfaces and regions, that form a watertight boundary, into a single 3D solid.
Deform toolset: Twist, Move Point, Move Edge, and Transform Curve	Deform your 3D solid uniformly.
Smart Selection	Select faces and edges of your 3D solids based on particular geometric and topological criteria, such as length, area and smoothness.
Parametrize	Add 3D constraints to faces and edges of 3D solids, to align and resize them. Make your 3D part parametric.
Design Intent	Automatically apply geometric 3D constraints, such as parallel, perpendicular, coaxial, and tangent.

Uwaga: BricsCAD is a history-free modeler; it does not create parent-child dependencies between features, nor regenerates geometry on parameter change. You can embed full-blown parametric control into any 3D part (including imported ones) using 3D constraints.

21.3.5 Modelowanie powierzchni

BricsCAD Mechanical zapewnia potężny zestaw narzędzi do modelowania powierzchni i konwersji brył 3D na powierzchnie i odwrotnie.

Narzędzia do modelowania powierzchni są dostępne na **Powierzchnia** zakładka **Mechaniczny** wstążka obszaru roboczego:



Rysunek 7: Powierzchnia

Modelowanie powierzchni i modelowanie brył to podobne narzędzia dotyczące edycji i stosowania wiązań 3D. Jedyną różnicą jest to, że powierzchnia Wyciągnięcie / Obrót / poddasze / czyszczące narzędzia mogą tworzyć jednostki powierzchni, z profili zamkniętych 2D, a także zaakceptować profile no-sklonowane.

Narzędzie Region tworzy region z zamkniętego profilu 2D, a regiony można zszyć w celu utworzenia większej powierzchni.

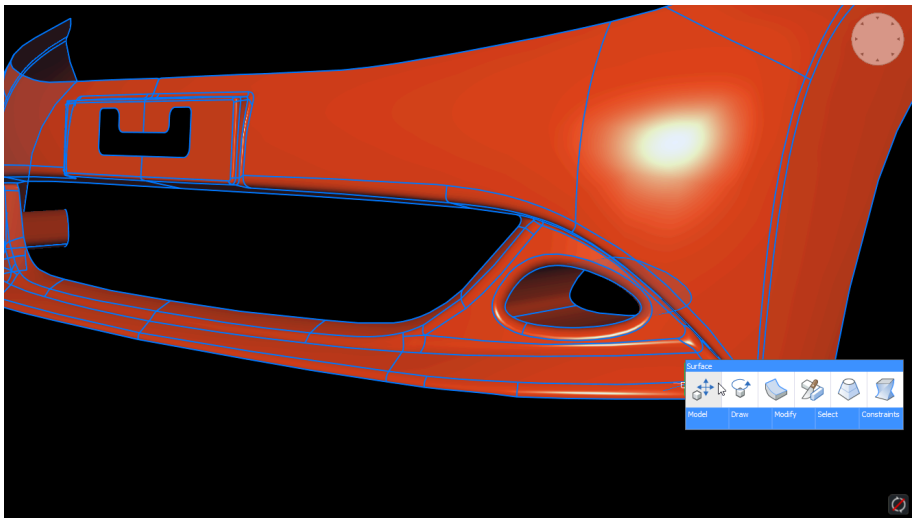
Inne metody tworzenia powierzchni obejmują wyodrębnienie jednej lub więcej sąsiednich ścian bryły / powierzchni 3D oraz rozbicie bryły 3D na zestaw powierzchni i regionów.

21.3.6 Swobodne modelowanie powierzchni

21.3.6.1 Przegląd

Dodaj złożone zniekształcenia do modeli bryłowych lub powierzchniowych za pomocą narzędzi deformacji BricsCAD Mechanical. Możesz tworzyć gładkie lub organiczne kształty. Deformuj bryły i powierzchnie, przesuwając, obracając i skręcając ich krawędzie, aby niezależnie zmieniać geometrię bez historii projektowania.

To podejście BEZ historii pozwala skoncentrować się na projekcie zamiast myśleć o tym, jak można go zmienić w przyszłości.



Rysunek 8: Swobodne modelowanie powierzchni

21.3.6.2 Narzędzia deformacji

Możesz użyć następujących narzędzi deformacji:



Rysunek 9: Deformuj

Tabela 2: Narzędzia deformacji

Narzędzie	Polecenie	Opis
Przesuń Krawędź	*DMDEFORMMOVE*	Deformuj ściany, przesuwając i obracając ich krawędzie.
Przesuń Punkt	*DMDEFORMPOINT*	Zdeformuj wybrany zestaw ścian, przesuwając punkt.
Przekształć Krzywe	*DMDEFORMCURVE*	Zdeformuj ściany, zmieniając geometrię ich krawędzi.
Skręt	*MBSKRĘĆ*	Deformuj obiekt 3D, skręcając część wokół określonej osi.

Wskazówka: Obejrzyj samouczek: Korzystanie z modelowania powierzchni i modelowania odkształcalnego.

21.4 Projekt zespołu

21.4.1 Bottom-up and Top-down Assembly Design

21.4.1.1 Overview

Assembly modeling represents the structure of a designed product and consists of using components to group geometric entities in hierarchies. Assembly modeling is commonly used in Mechanical Computer-Aided Design (MCAD) applications, it simplifies the design of complex products containing thousands or millions of geometric entities.

The following terms are used in this guide:

- An assembly is a component that contains inserts of other components
- A subassembly is a separate assembly that is inserted into another assembly
- A part is a component that does not contain inserts of another component.

An assembly with inserted components (which in turn can be assembled with other inserted components) forms a tree that is called a mechanical structure of the 3D model. This tree is displayed in the Mechanical Browser.

There are 2 main ways to design mechanical products: **top-down** and **bottom-up** approaches. You can also add mechanical structure to existing geometry, designed in BricsCAD or imported from another source.

Use the *BMNEW* command to create a new document for a new mechanical component.

Use the *BMMECH* command to convert existing blocks and external references to components. Use the *BMUNMECH* command to do the opposite.

21.4.1.2 Bottom-Up Design

With bottom-up approach, you can start to design a product by detailing its low-level component. These components are then grouped into higher-level components. The process is continued until the very top component is corresponding to the product itself.

The main command for bottom-up design is *BMINSERT*. Use this command to insert an existing component into the current drawing.

21.4.1.3 Top-Down Design

With top-down approach, you can start to design a product by defining, but not detailing, its first-level components. Each component is then refined in more detail by defining its subcomponents. The process is continued till all low-level components are defined and detailed.

Use the *BMFORM* command to create a new subcomponent for an existing component.

21.4.1.4 Adding Mechanical Structure to Existing Geometry

You can add component hierarchy to any existing drawing, created in BricsCAD, or imported from a different CAD format.

Use the *BMFORM* command to convert a selected set of geometric entities to a subcomponent of the current drawing.

21.4.1.5 Changing Mechanical Structure

During the design process, you can realize that a particular component must be moved up or down in the component hierarchy.

Use the *BMFORM* command to form a new subcomponent for the current drawing that will contain a selected set of a component.

Use the *BMDISSOLVE* command to delete a component and move all its subcomponents to the level of the dissolved component.

21.4.1.6 Using Standard Components

A few standard components are:

- screws
- nuts
- washers
- bearings
- holes.

BricsCAD provides a huge library of standard components, more than 30,000 details in a variety of sizes. You can significantly increase design performance by using standard parts.

Use the Components panel to insert standard components into your parts and assemblies.

21.4.1.7 Assembly Constraints

Assembly constraints are ordinary 3D Constraints, that link faces and edges of bodies to represent the components of an assembly. You can link 3D Constraints at the top-level and at the subcomponents level.

21.4.1.8 Component Visibility

A component within a complex assembly may seem invisible due to other components surrounding it.

Example: a piston is invisible inside a cylinder block. To make such a component visible, you may need to hide the other components surrounding it.

You can hide any subcomponent inserted into the current drawing or any subcomponents, on any level of nesting.

You can also make a component transparent and/or visualize it as a wireframe model.

Use the *BMHIDE* command to hide a particular subcomponent.

Use the *BMSHOW* command to make a hidden subcomponent visible.

21.4.2 Polecenia i paski narzędzi projektowania złożeń

21.4.2.1 Polecenia

Tabela 3: Polecenia projektowania zespołu

Ikona	Polecenie	Opis
	*BMNOWY*	Tworzy nowy komponent mechaniczny jako nowy plik rysunku
	*BMMECH*	Przekształć bieżący rysunek w główny komponent mechaniczny.
	*BMWSTAW*	Wstawia istniejący komponent mechaniczny do bieżącego rysunku.
	*BMWSTAW*	Wstawia istniejący komponent mechaniczny do bieżącego rysunku z automatyczną zmianą rozmiaru nowej części i połączeniem jej z częścią macierzystą.
	*BMHARDWARE*	Otwiera panel Komponenty ze standardowymi komponentami mechanicznymi
	*BMFORM*	Tworzy nowy komponent mechaniczny i wstawia go do bieżącego rysunku.
	*BMCONNECT*	Łączy dwie standardowe części orurowania, tworząc wiązania 3D między ich elementami połączenia.
	*BMZAMIEŃ*	Zastępuje wkładkę komponentu.
	*BMLOKALIZUJ*	Przełącza komponenty zewnętrzne na komponenty lokalne.
	*BMUZEWNĘTRZNIJ*	Przełącza komponenty lokalne na komponenty zewnętrzne.
	*BMOTWÓRZ*	Otwiera rysunek źródłowy zewnętrznego komponentu mechanicznego.
	*BMOTWÓRZKOPIĘ*	Otwiera kopię wstawki komponentu jako nowy rysunek.
	*BMROZWIĄŻ*	Usuwa komponent mechaniczny wstawiony do bieżącego rysunku.

Tabela 3: Polecenia projektowania zespołu

Ikona	Polecenie	Opis
	BMLINK	Zmienia docelowe bryły 3D elementów opartych na komponentach.
	*BMODŁĄCZ*	Odłącz: Przerywa połączenie między wkładką komponentu (np. okno) i bryły 3D (np Ściana) utrzymująca otwór.
	*BMUKRYJ*	Ukrywa komponent mechaniczny lub podkomponent w bieżącym rysunku.
	*BMPOKAŻ*	Pokazuje poprzednio ukryty komponent mechaniczny lub komponent podrzędny w bieżącym rysunku.
	*BMVSTYL*	Stosuje styl wizualny do komponentu mechanicznego
	*BMBALONY*	Tworzy skojarzone elementy numeru pozycji dla komponentów zespołu w obszarze modelu i dla ich wygenerowanych widoków w układzie obszaru papieru.
	*BMBALONY*	Umożliwia automatyczne umieszczanie skojarzonych odnośników na wszystkich komponentach w określonym widoku rysunku w układzie obszaru papieru.
	*BMRAPORT*	Wstawia tabelę zestawienia komponentów (BOM) do bieżącego rysunku
	*BMPARAMFIZ*	Oblicza właściwości masy dla bieżącego modelu przy użyciu gęstości przypisanych do jego komponentów.
	*BMAKTUALIZUJ*	Wczytuje ponownie wszystkie odnośne komponenty z plików zewnętrznych i aktualizuje tabele BOM.
	*BMAKTUALIZUJ*	Wczytuje ponownie wszystkie wybrane komponenty, do których istnieją odniesienia, z plików zewnętrznych i aktualizuje tabele BOM.
	*BMROZBIJ*	Tworzy rozstrzeloną reprezentację dla złożenia, przesuwając komponenty, aby wszystkie były widoczne, przechowuje je w nowym bloku i wstawia blok w Obszarze modelu.

Tabela 3: Polecenia projektowania zespołu

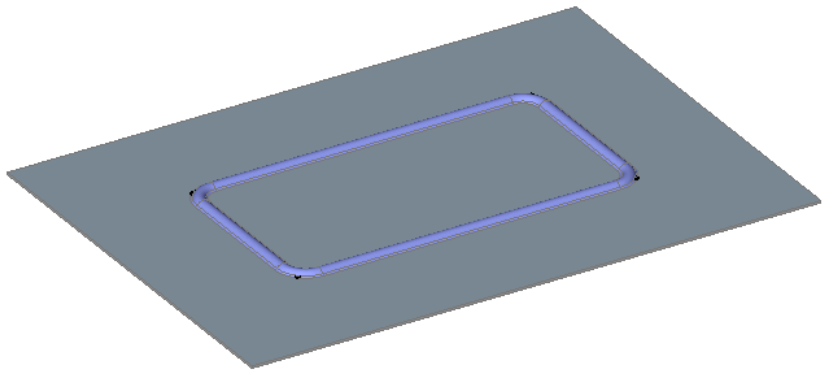
Ikona	Polecenie	Opis
	*BMPRZESUŃEKSPLODOWANE*	Przesuwa wybrane części w celu utworzenia rozstrzelonej reprezentacji w podanym kierunku, biorąc pod uwagę możliwe fizyczne kolizje między komponentami (podobnie jak w opcji Liniowa polecenia *BmExplode*).
	*BMLINIEŚLEDZENIA*	Tworzy wszystkie niezbędne linie końcowe dla wybranych części.
	*OTWÓRZPRZEGMECHANICZNĄ*	Wyświetla przeglądarkę mechaniczną.
	*OTWÓRZPANELPARAMETRÓW*	Wyświetla Menedżera parametrów.
	*BMZALEŻNOŚCI*	Wyświetla wszystkie pliki w oknie poleceń, zawierające definicje komponentów wstawione do zespołu,.
	*BMNAPRAW*	Przywraca zepsutą strukturę mechaniczną
	*BMNIEMECH*	Konwertuje bieżący komponent mechaniczny na zwykły rysunek.
	BMXKONWERTUJ	Konwertuje bryły X-Hardware w bieżącym rysunku na komponenty mechaniczne.

21.4.2.2 Narzędzia

Do pracy z komponentami mechanicznymi można używać następujących narzędzi:

1 Na paskach narzędzi **montażowych**:

- **Złożenie**



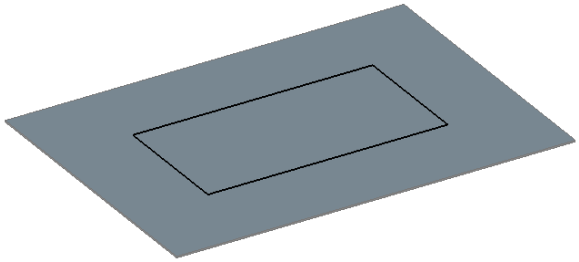
Rysunek 10: Złożenie

- **Wstaw złożenie**



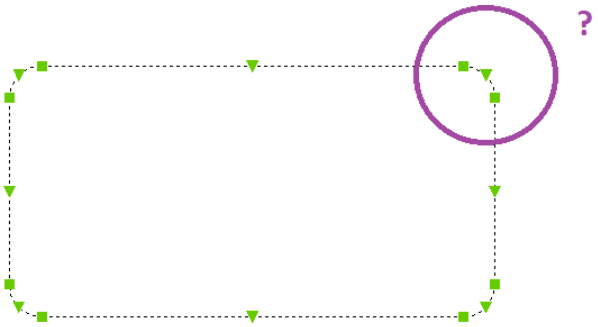
Rysunek 11: Wstaw złożenie

- **Rozstrzelenie Złożenia**



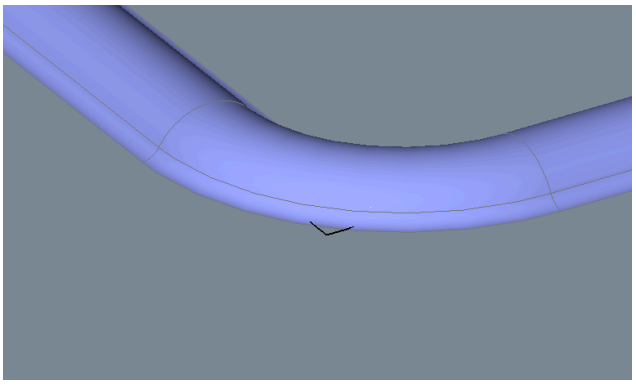
Rysunek 12: Rozstrzelenie Złożenia

- **Wizualizacja Złożenia**



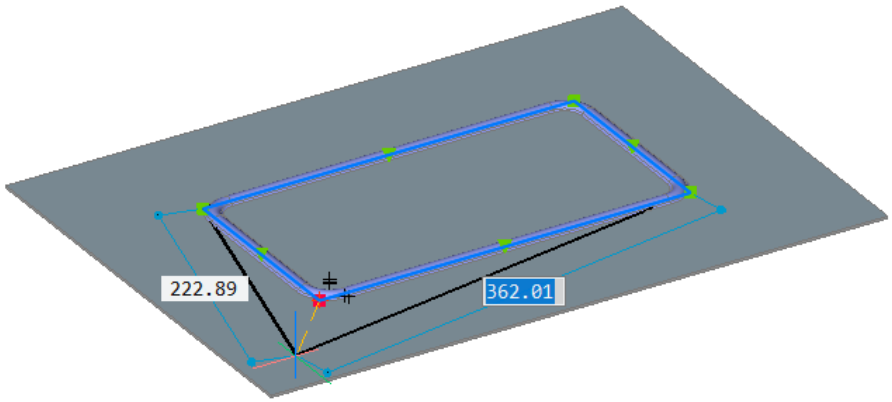
Rysunek 13: Wizualizacja Złożenia

2 W menu **Złożenie** (tylko Mechanical Workspace):

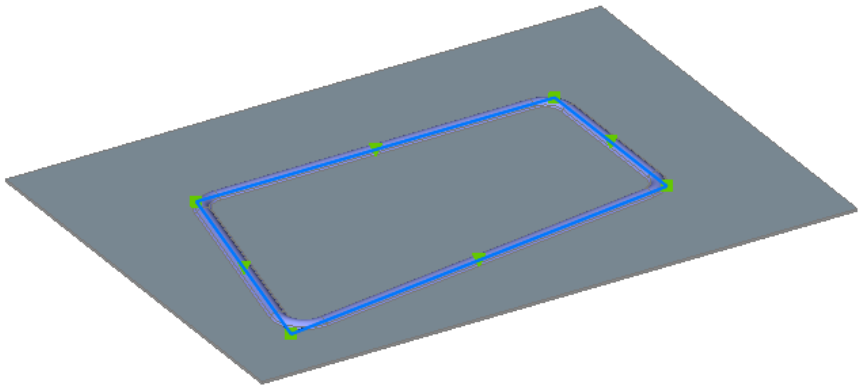


Rysunek 14: Menu montażu

3 Na karcie **Złożenia** a wstążce Mechaniczny obszar roboczy:



Rysunek 15: Złożenie



Rysunek 16: Narzędzia

21.4.3 Component Properties

Component properties and context menus are located in the **Mechanical Browser** panel.

Component context menus display when you right-click the component. Component properties display automatically when selecting one or more components in the Mechanical Browser panel.



Rysunek 17: Mechanical Browser

Components can be: **Main**, **Insert** and **Subcomponent**.

Tabela 4: Components

Type of component	Rendered visual style	Wireframe visual style	Hidden
Component without subcomponents	*	*	*
Component with subcomponents	*	*	*
Standard part component			

* External components contain a green arrow: or .

Tabela 5: Component Properties

Context menus	
Main Component context menu	Right-click the main component name:  **Rysunek 18:** Main Component context menu • **Update**: update 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**: apply the current **Visual Style** to all components in the assembly (see the *SHADEMODE* command). • **Insert standard hardware**: insert a standard hardware part as a mechanical component in the current drawing. • **Switch all to local**: switch all components to internal components (see the *BMLOCALIZE* command). • **Switch all to external**: switch all components to external components (see the *BMEXTERNALIZE* command). • **Add new parameter**: create a new parameter in the assembly. • **Select all**: select all components with the same definition. • **Select same**: select all components with the same name and the same parameter values. • **Highlight all**: highlight all components with the same definition. • **Highlight same**: highlight all components with the same name and the same parameter values. • **Create exploded view**: create a block with an exploded representation of the current assembly. • **Collapse all**: collapse the main component and all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.

Tabela 5: Component Properties

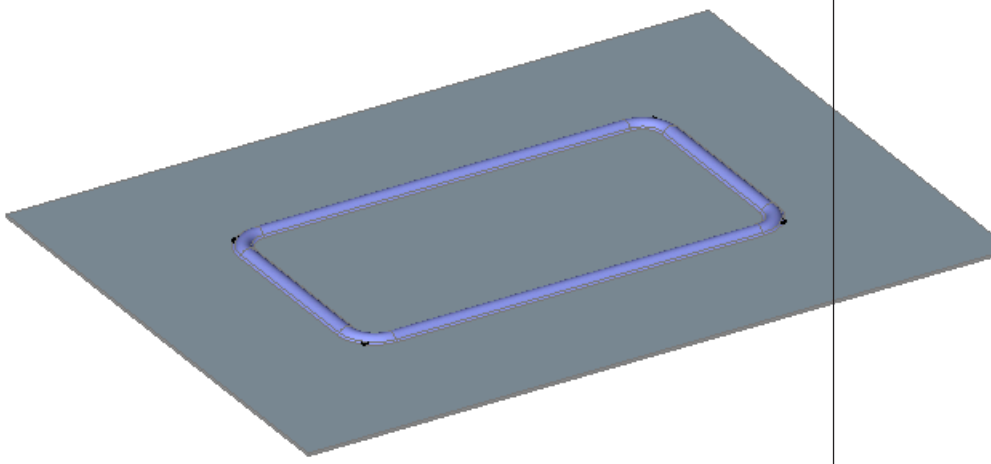
Context menus	
Component context menu	Right-click a component name. External components context menu (left), local components context menu (right):  • **Open**: opens the referenced drawing (see the*BMOPEN* command). • **Open a copy**: open a copy of a component insert as a new drawing (see the *BMOPENCOPY* command). • **Update**: reload all referenced components from external files and update BOM tables (see the *BMUPDATE* command). • **Replace**: replace a component insert (see the *BMREPLACE* command). Replacing a local insert turns it into an external insert. • **Replace all inserts**: replace all inserts that refer to the same source (see the *BMREPLACE* command). Replacing local inserts turns them into external inserts. • **Switch to local**: switch an external component to an internal component (see the *BMLOCALIZE* command). • **Switch to external**: switch an internal component to an external component (see the *BMEXTERNALIZE* command). • **Set material to component**: assign a physical material to a local component. • **Remove material from component**: remove the physical material definition from a local component. • **BOM status**: parameter to control the appearance of the component in BOM tables. • **Dissolve**: dissolve a mechanical component inserted in the current drawing (see the *BMDISSOLVE* command). • **Unlink**: break the connection between a component insert (for example a Window) and a 3D solid (for example a Wall) retaining the opening. • **Hide** ( ) / **Show** ( ): hide or show the selected component. • **Exclude all inserts from section**: set the **Sectionable** property of all similar inserts to *NO*. Define whether an insert is affected by the *VIEWSECTION* command. • **Visual Style**: display the **Visual Style** menu. All Visual Styles that are saved in the current drawing are available. - **By parent component**: render the selected component according to the shademode of the parent component (see the *SHADEMODE* command). - **All by Viewport**: render all components according to the current viewport shademode (see the *SHADEMODE* command). • **Zoom to**: zoom to the selection set.

Tabela 5: Component Properties

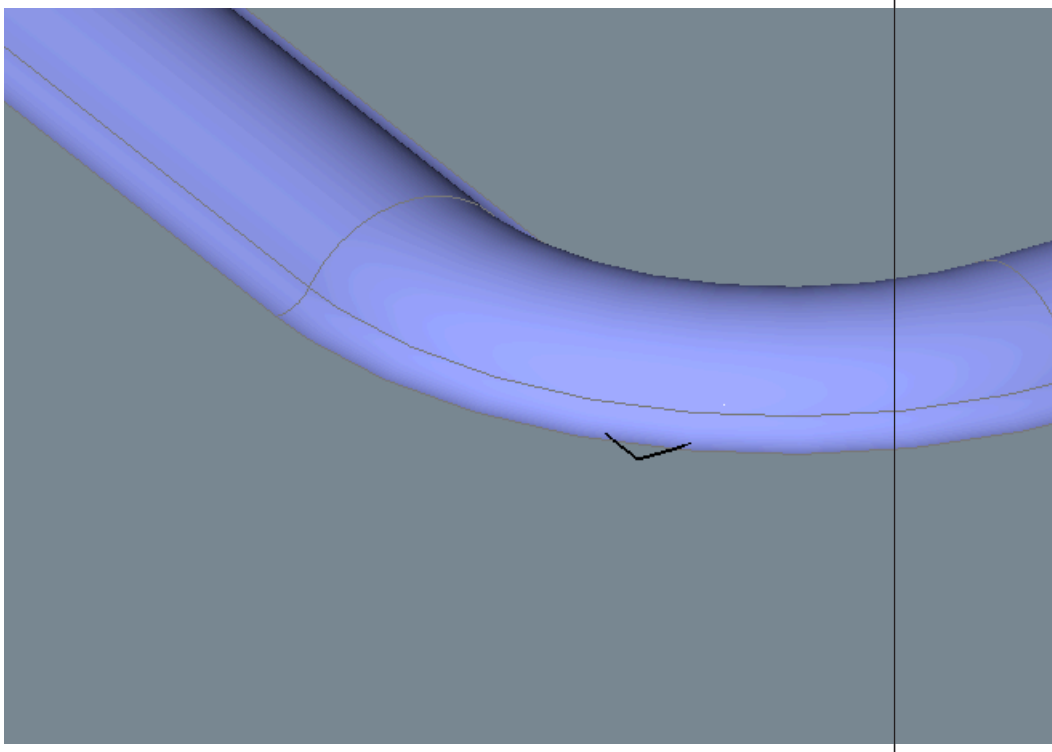
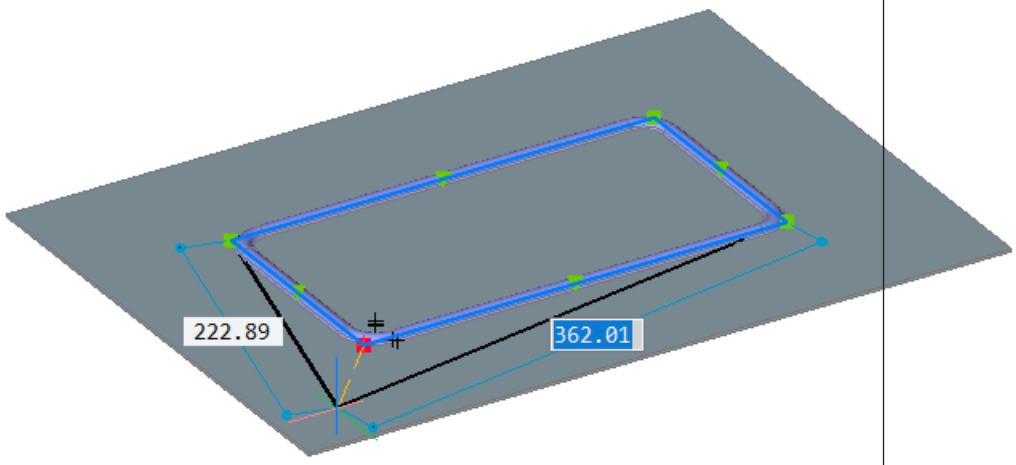
Context menus	
Subcomponent context menu	External subcomponents context menu (left), local subcomponents context menu (right):   • **Open**: open the referenced drawing (see the *BMOPEN* command). • **Open a copy**: open a copy of a component insert as a new drawing (see the *BMOPENCOPY* command). • **Hide** ( ) / **Show** ( ): hide or show the selected subcomponent. • **Exclude all inserts from section**: sets the **Sectionable** property of all similar inserts to *NO*. Define whether an insert is affected by command *VIEWSECTION*. • **Visual Style**: see Component Visual Style. • **Zoom to**: zoom to the selection set. • **Select all**: select all components with the same definition. • **Select same**: select all components with the same definition and the same parameter values. • **Highlight all**: highlight all components with the same definition. • **Highlight same**: highlight all components with the same definition and the same parameter values. • **Collapse all**: collapse the main component and all components and subcomponents.

Tabela 5: Component Properties

Context menus	
Component parameter context menu	Right-click a parameter: • **Link to parameter**: create a link between one or more component parameters and assembly parameters. • **Collapse all**: collapse the main component and all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.
Properties	

Tabela 5: Component Properties

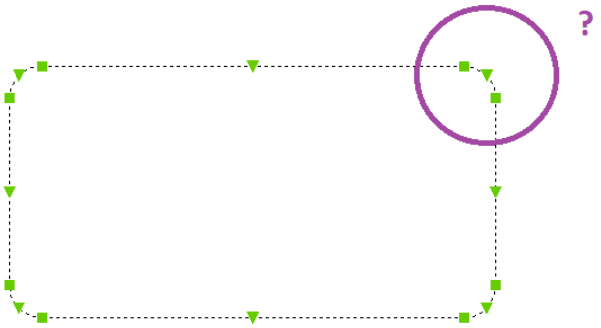
Context menus	
Main component properties	Click the main component name:  **Rysunek 19:** Main component properties • **Name**: type a name for the main component. By default, the name of the main component equals the file name. • **Description**: optional description of the main component. • **File**: show the path and name of the drawing. • **Extension type**: identify the component as a BIM Component or a Sheet Metal Feature. • **Insert as**: select **Internal Component** or **External Component**. • **Sectionable**: define whether the component is affected by the *VIEWSECTION* command. • **Standard component**: define whether this component is a standard component or not. • **BOM status**: define the behavior of this component in a BOM table. • **Material**: material of the main component. The **Name**, **Density**, and the optional **Description** are defined in the Physical Materials dialog box. Select the **Material** node and perform one of the following: - Click the **Browse** button (...) to open the Physical Materials dialog and choose a material in the Project or Library list. The Density property of the selected material is used by the *BMMASSPROP* command. - Click the **Delete** button (x). The **Material** field now reads <*Inherit*>. When the component is inserted in an assembly, the **Material** definition is copied from the main component of the assembly.

Tabela 5: Component Properties

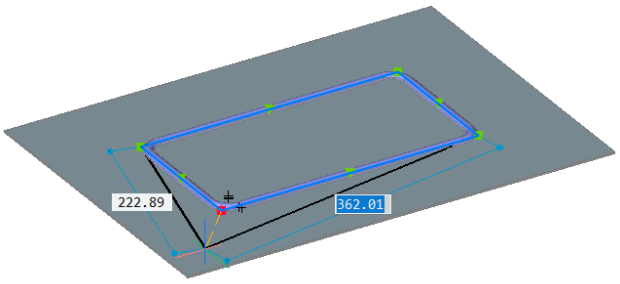
Context menus	
Component insert properties	Click the component name:  **Rysunek 20:** Component insert properties • **Name**: type a name for the component. By default, the name of the component equals the source file name. • **Component name**: name of the component as defined in the source file. • **Description**: description of the component as defined in the source file. • **Visible**: show or hide the component in the assembly. Choose between: - Click the **Visible** field, click the down arrow button and choose *Yes* or *No*. - Double-click the **Visible** field to toggle the visibility of the component. • **Sectionable**: define whether the component is affected by the *VIEWSECTION* command. • **Standard component**: define whether this component is standard or not. • **BOM status**: define the behavior of this component in a BOM table. • **Visual style**: allow to control the visual style for the component. By default, the visual style is inherited from the viewport. Click the **Visual style** field, click the down arrow button, and choose a visual style in the list. • **File**: show the path and name of the referenced drawing. • **Material**: show the material of the component. If set to *<Inherit>*, the material is copied from the main component. If the material is defined explicitly in the source file, this material is used.

Tabela 5: Component Properties

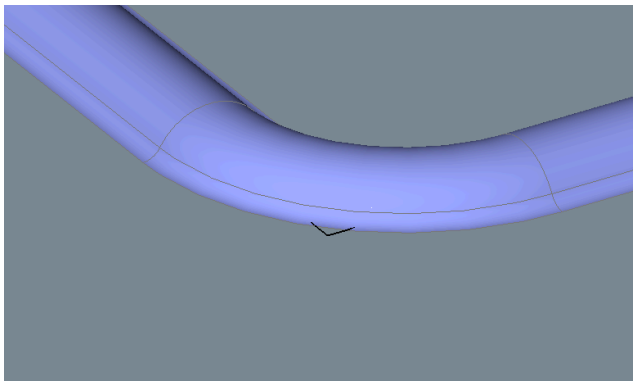
Context menus	
Subcomponent properties	Click the subcomponent name:  **Rysunek 21:** Subcomponent properties • **Name**: name of the component as defined in the source file of the parent component. • **Component name**: name of the component as defined in the source file. • **Description**: description of the component as defined in the source file. • **Visible**: show or hide the component in the assembly. Choose between: - Click the **Visible** field, click the down arrow button and choose *Yes* or *No*. - Double-click the **Visible** field to toggle the visibility of the component. • **Sectionable**: define whether the component is affected by the *VIEWSECTION* command. • **Standard component**: define whether this component is standard or not. • **BOM status**: define the behavior of this component in a BOM table. • **Visual style**: control the visual style for the component. By default, the visual style is inherited from the viewport. Click the **Visual style** field, click the down arrow button, and choose a visual style in the list. • **File**: show the path and name of the referenced drawing. • **Material**: show the material of the subcomponent. If set to *<Inherit>*, the material is copied from the main component. If the material is defined explicitly in the source file, this material is used.
Selection methods	

Tabela 5: Component Properties

Context menus	
Select component(s) in the model	The component(s) highlights in the Mechanical Browser.
Click a component in the Mechanical Browser panel	The component highlights in the drawing. Press the space bar to select the component.

21.4.4 Właściwości masy w Złożeniach

21.4.4.1 Przegląd

Użyj polecenia *BMPARAMFIZ*.

Zespół może składać się z wielu części wykonanych z różnych materiałów. Aby poprawnie obliczyć masę, środek ciężkości (środek ciężkości) i bezwładność takich zespołów, BricsCAD obsługuje indywidualne gęstości komponentów.

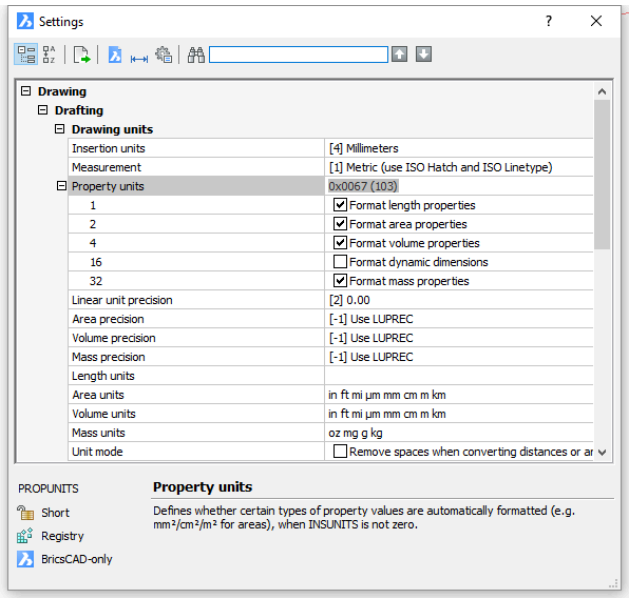
Materiał fizyczny dla komponentu mechanicznego jest zdefiniowany we właściwościach Materiału w **Przeglądarce Mechanicznej**.

Domyślnie komponent dziedziczy gęstość złożenia, do którego został wstawiony, lub z którego został utworzony.

21.4.4.2 Ustaw jednostki masy

1 Otwórz okno dialogowe *Ustawienia* .

2 Rozwiń ustawienia **Właściwości jednostek** w obszarze **Rysunek / Rysowanie / Jednostki rysunku**.



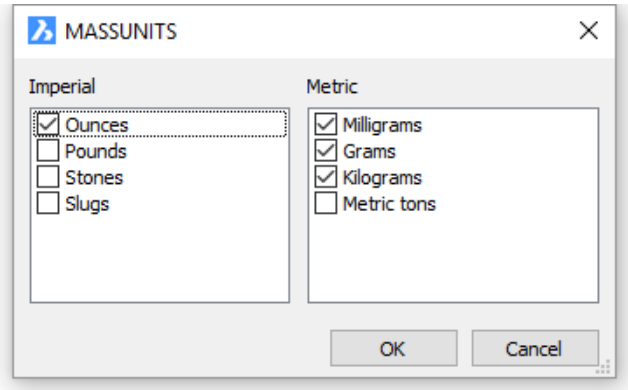
Rysunek 22: Ustawienia: Właściwości jednostek

3 Zaznacz pole wyboru **Formatuj właściwości masy**, aby włączyć formatowanie wartości masy za pomocą odpowiednich jednostek.

4 Wybierz pole **Jednostki masy**.

5 Kliknij przycisk **Przeglądaj** ....

6 Określ żądane jednostki masy w oknie dialogowym *JEDNOSTKI MASY*.



Rysunek 23: JEDNOSTKI MASY

Uwaga: Ustawienie **Jednostki masy** wpływa tylko na wartości masy. Inne właściwości masy, takie jak gęstość lub momenty bezwładności, zostaną sformatowane w jednostkach SI dla układu metrycznego oraz w jednostkach imperialnych w przypadku systemu imperialnego, niezależnie od wartości ustawienia **Jednostki masy**.

7 Określ żądaną dokładność w polu **Precyzja masy**.

8 Otwórz okno dialogowe *Ustawienia* .

21.4.4.3 Oblicz właściwości masy

Użyj polecenia *BMPARAMFIZ*.

Można obliczyć właściwości masy dla całego modelu lub dla wybranych brył 3D i / lub elementów składowych. Na pasku poleceń podawane są wartości masy, objętości, środka ciężkości, momentów bezwładności, iloczynów bezwładności, promieni bezwładności, momentów głównych i kierunków XYZ względem środka ciężkości.

Opcjonalnie można wyrównać układ współrzędnych użytkownika z osiami bezwładności. W rezultacie, środka masy lub środka masy składników leży u podstaw UCS, która może mieć przydatna w celu zbadania zachowania się produktu pod wpływem siły grawitacji.

21.4.5 Standard Mechanical Components

21.4.5.1 Overview

Use command *COMPONENTSPANELOPEN*.

BricsCAD comes with a library of more than 1,000 standard mechanical parts available in more than 170,000 different sizes. They are available in the Library panel, and can be drag-and-dropped from the library into your assembly. You can adjust the parameters in the Properties panel, which temporarily lists

parameters of the component being inserted. After insertion, parameters can be edited in Mechanical Browser and the Properties panel.

In the Mechanical workspace, the Library panel contains 3 categories:

- Standard Parts
- Holes
- Sheet Metal

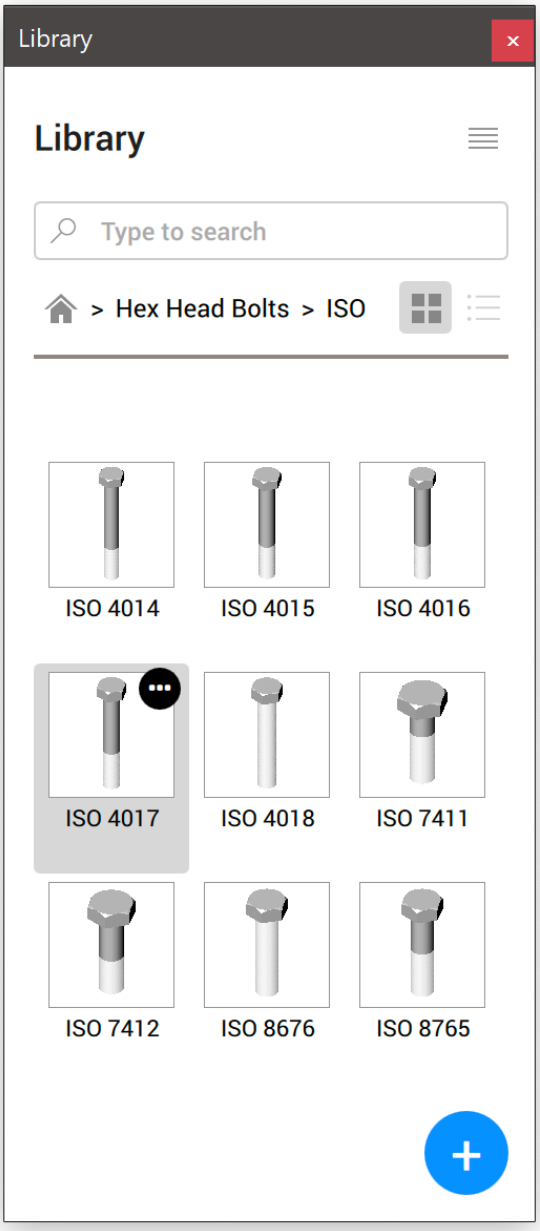
21.4.5.2 Standard Parts (Hardware)

The Standard Parts category contains parametric standard hardware grouped into 4 subcategories:

- Fasteners, such as bolts, nuts, and washers.
- Machine Parts, such as bearings, and sprockets.
- Piping, such as pipes, elbows, and flanges.
- Structural Shapes, such as beams, channels, and bars.

The content of each subcategory is structured by type, subtype, and standards.

Example: Standard Parts / Fasteners / Bolts / Hex Head Bolts / ISO / ISO 4017.



Rysunek 24: Library

Uwaga: There are 2 specific settings you can configure for Standard Parts in the *Settings* dialog box:

- **Thread representation (THREADDISPLAY)** is used to represent (or not to represent) a 3D thread for bolts and nuts. This setting allows to decrease the memory consumption and increase the performance for large assemblies
- **Maximum number of sprocket teeth (GEARTEETHNUMBER)** is used to limit the number of teeth for sprocket wheels (available under **Standard Parts / SPROCKET)**. This setting allows to decrease memory consumption and increase the performance.

By default, these settings are set at *Off*, in order to optimize the performance of the model. When changed, the settings apply to all new documents.

Standard representation of thread in drawing views

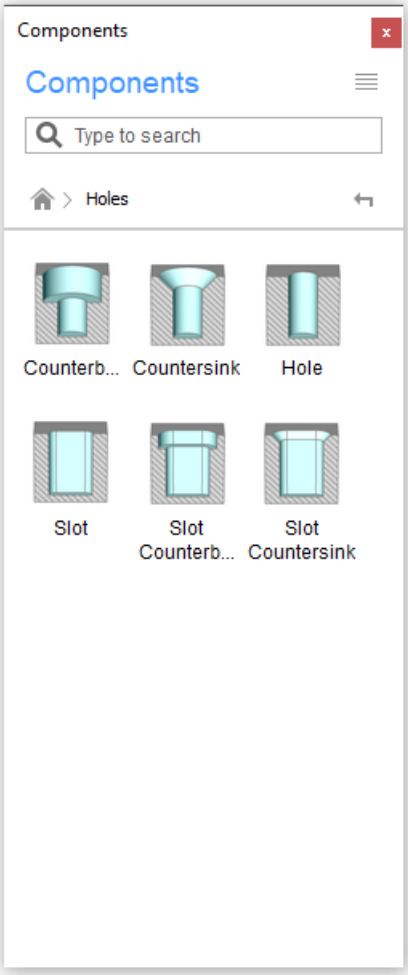
Drawing views of your assemblies, generated with the *VIEWBASE* command, contain thread annotations for standard parts as thin lines for side views, and as a 270-degree thin arc for top/bottom views.



Rysunek 25: Example

21.4.5.3 Holes

The Holes category contains a parametric library of standard holes, such as slots, counterbores, and countersinks.

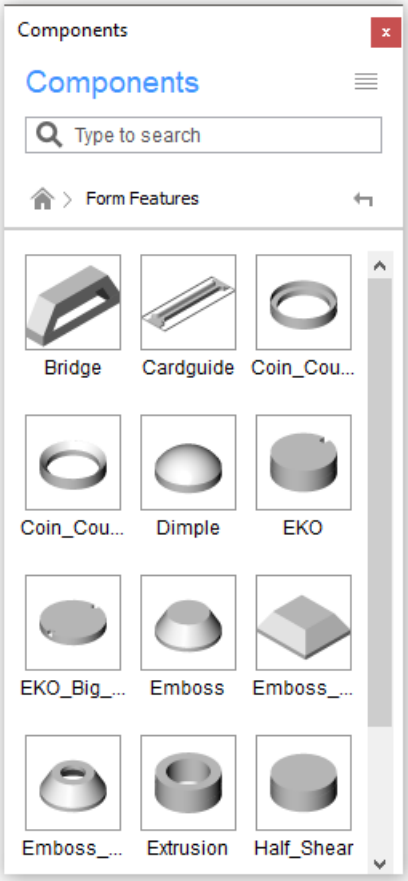


Rysunek 26: Holes

There is also a subcategory of threaded holes, represented in drawing views.

21.4.5.4 Sheet Metal Form Features

The Sheet Metal category contains a parametric library of sheet metal form features, such as bridges and embosses.



Rysunek 27: Form Features

21.4.5.5 Place a standard component into the current drawing

1 Choose a category.

2 Choose a standard.

3 Select a type.

4 Click on the desired standard component.

5 Edit the parameters in the properties panel.

6 Click in the drawing area to insert the selected component.

21.4.5.6 Create your own standard component

Use command *BMCREATECOMPONENT* to create your own standard parts and add them to Components panel.

21.4.5.7 Control the display of threads

1 Open the *Settings* dialog box.

2 Find the *Thread representation* variable (**THREADDISPLAY**).

3 Enable the setting.

4 If applicable, update the document to make threads appear on existing parts:
   - Temporarily set the **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
   - Use command *BMUPDATE* to update the document.
   - If applicable, return the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

21.4.5.8 Control the number of sprocket teeth

1 Open the *Settings* dialog box.

2 Find the *Maximum number of sprocket teeth* variable (**GEARTEETHNUMBER**).

3 Set the value to be sufficient for used sprockets (*500* is recommended).

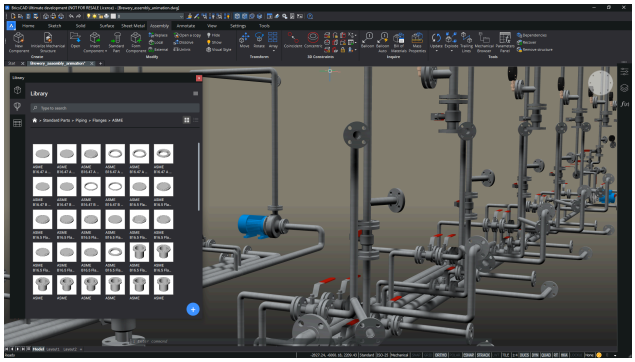
Uwaga: Larger values slow down the update of standard sprockets. If the value is smaller than the required number, only that number of sprocket teeth displays.

4 If applicable, update the document to make sprocket teeth appear on existing parts:
   - Temporarily set the **Assembly components updating mode (BMUPDATEMODE)** to **[1] Update all components**.
   - Use command *BMUPDATE* to update the document.
   - If applicable, return the **Assembly components updating mode (BMUPDATEMODE)** to the previous value.

21.4.6 Piping in Assembly Design

21.4.6.1 Overview

Use commands: *BMINSERT*, *BMCONNECT*, and *BMBOM*.



Rysunek 28: Piping in Assembly Design

Seamless and welded pipes, elbows, tees, crosses, reducers and flanges are widely used in machinery, automotive, and aviation industries. They are also found in oil-and-gas, chemical industry, and water supply. BricsCAD has a library of parametric piping parts of different standards, such as ASME, DIN, and ISO. These standard parts are parametric. You can easily adjust their parameters, link them to different global parameters, and use 3D constraints to properly connect the parts. BricsCAD allows you

to streamline these routine operations using the **Smart mode** of command *BMINSERT* and a dedicated command *BMCONNECT*.

21.4.6.2 Insert piping parts in your assembly

Drag and drop any piping standard part from the **Library** panel into your assembly.

By default, all parts from the Piping category are inserted in Smart mode, which means that if you hover over a previously inserted piping part, BricsCAD:

- Copies parameter expressions of that part to the corresponding parameters of the part being inserted.
- Adds a set of 3D constraints to connect the parts.

Uwaga: Press the **Ctrl** key to switch between Smart mode and Regular mode, in which 3D constraints with other parts are not generated.

21.4.6.3 Create 3D constraints between piping standard parts manually

Piping standard parts have a set of wireframe entities (points and lines) on the STD_PARTS_PIPING_CONNECTION layer, which is switched off by default.

Constraining a fitting

To constrain a fitting (elbow, reducer, flange, ...) with another fitting or a pipe, create 2 Coincident 3D constraints between:

- 2 points of the fittings, which should coincide
- 2 lines of the fittings, which should be collinear.

Constraining a pipe

Constraining pipes is more complex because a pipe can have any length. If a pipe is already constrained with a fitting at one side using 2 Coincident 3D constraints, then constraining it from the other side must be different.

You must create:

- A Coincident 3D constraint between the line, which represents the axis of the pipe, and the corresponding line of the other fitting
- A geometry-driven Distance 3D constraint between the planar face of the pipe on the opposite (already constrained) side and the corresponding point of the fitting, which is being connected.

You must then type the name of the distance constraint in the expression field for the Length parameter of the pipe. Because geometry-driven constraints allow controlling of a particular dimension, the pipe will always have the correct length, and such a parametrization will not limit any further editing of your assembly using direct manipulation of additional non-geometry-driven dimensional constraints.

21.4.6.4 Automatically connect piping parts

The set of constraints is created automatically using the Smart mode of command *BMINSERT* or the dedicated command *BMCONNECT* to connect 2 standard parts.

21.4.6.5 Create a Bill of Materials for a piping assembly

In a BOM you may want all pipes of the same type and size (PartNumber), but with different lengths, listed as the same part

Use the Aggregate function of the BOM Manager to combine all such parts in a single row with their total length.

21.4.7 Tablice komponentów

21.4.7.1 Przegląd

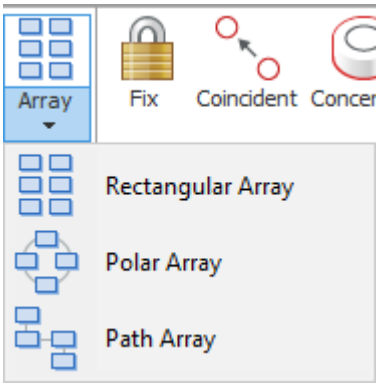
Użyj poleceń *SZYK*, *SZYKPROSTOKĄTNY*, *SZYKBIEGUNOWY*, *SZYKPOŚCIEŻCE* oraz *BMWSTAW*.

BricsCAD pozwala na wstawianie tego samego komponentu wiele razy w sposób regularny. Wszystkie wstawienia mogą być rozmieszczone równomiernie (w tej samej odległości) wzdłuż linii, okręgu lub dowolnej krzywej. Obsługiwane są również wzory wyższego rzędu (na przykład prostokątne lub cylindryczne). Osiąga się to dzięki wykorzystaniu technologii szyków asocjacyjnych: prostokątnego, biegunowego i po ścieżce.

Szyki asocjacyjne to trwałe obiekty, które można edytować (ręcznie lub automatycznie) w dowolnym momencie po utworzeniu - zmienić liczbę elementów lub odległość między nimi. Wszystkie szyki w modelu są wyświetlane w Przeglądarce Mechanicznej, gdzie można je edytować.

Wszystkie komponenty można ustawić w szyku - w tym części, podzespoły, części znormalizowane i otwory.

21.4.7.2 Utwórz szyk komponentów

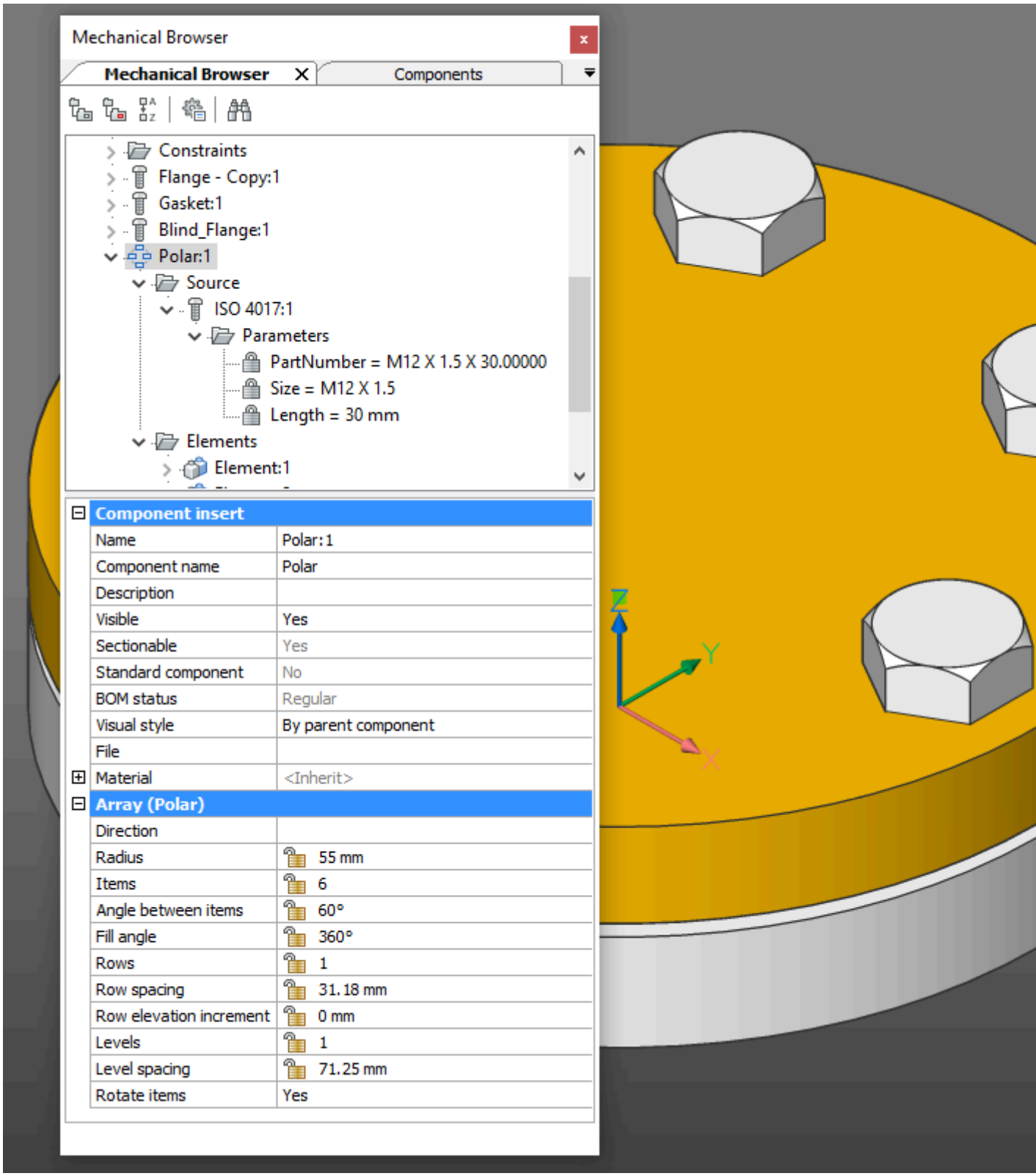


Użyj jednego z narzędzi szyku w panelu **Złożenie** profilu Mechanicznego:

- **Szyk Prostokątny**
- **Szyk Biegunowy**
- **Szyk po Ścieżce**.

Innym sposobem tworzenia szyku komponentów jest użycie opcji Wiele/Szyk w trakcie narzędzia Wstaw komponent .

21.4.7.3 Szyki komponentów w Przeglądarce Mechanicznej



Szyki komponentów są reprezentowane jako specjalne węzły w Przeglądarce Mechanicznej. Ich nazwy pochodzą od typu szyku (Prostokątny, Biegunowy lub Ścieżka), po której następuje średnik i indeks szyku (na przykład Prostokątny:1, Biegunowy:2, Ścieżka:3). Węzły te mają takie same właściwości jak zwykłe szyki asocjacyjne. Właściwości mogą służyć do zmiany liczby elementów w szyku, odległości/kąta między nimi oraz innych parametrów szyku. Do dowolnej właściwości można również przypisać trwałe wyrażenie. Zostanie ono wykorzystane do automatycznego przeliczenia wartości właściwości po zmianie jej parametrów.

Każdy węzeł szyku ma dwa węzły podrzędne: Źródło i Elementy.

- W węźle Źródło można znaleźć wstawiony komponent i jego parametry.
- Węzeł Elementy można rozwinąć do listy elementów szyku, z możliwością ukrycia lub zmiany stylu wizualnego danego elementu wewnątrz szyku.

21.4.7.4 Szyki komponentów i Zestawienie Materiałów

Szyki komponentów są transparentne dla struktury produktu. Komponenty wewnątrz szyków są uważane za wstawione bezpośrednio do tego samego zespołu. Tak więc w zestawieniu materiałów (BOM) najwyższego poziomu komponenty wstawione do szyków najwyższego poziomu są wymienione jako komponenty najwyższego poziomu.

Kolumna Ilość w zestawieniu komponentów jest wypełniona całkowitą liczbą wstawek tego samego komponentu, w tym komponentów znajdujących się w szykach.

21.4.7.5 Szyki komponentów parametrycznych, części znormalizowanych i otworów

Jeśli komponent parametryczny jest wstawiany jako szyk, jego parametry są nadal dostępne do edycji i połączone z parametrami globalnymi. Technicznie parametry są powiązane z elementem źródłowym szyku. Gdy ten element źródłowy jest edytowany parametrycznie, wszystkie inne elementy szyku są zmieniane według niego.

Części znormalizowane i otwory są komponentami parametrycznymi i można edytować ich parametry wewnątrz szyków. Można łatwo utworzyć część z wieloma otworami i parametrycznie kontrolować numer otworu, położenie i rozmiar, a także wstawić parametryczną śrubę. Ta zaawansowana technologia umożliwia tworzenie parametrycznego złożenia kołnierza w wielu konfiguracjach.

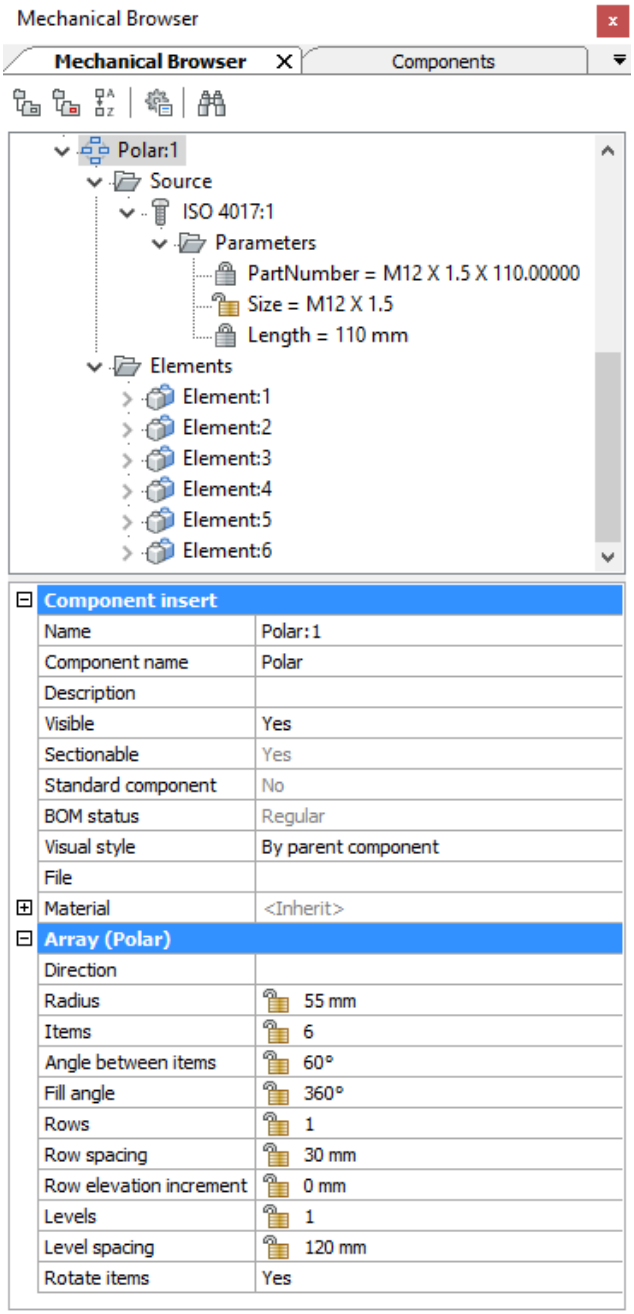
21.4.8 Array Properties

21.4.8.1 Overview

Component array is an associative array made of a parametric component. Component arrays have a special representation in the **Mechanical Browser**.

Component array have 2 main subnodes:

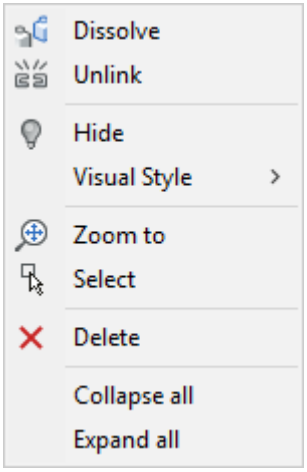
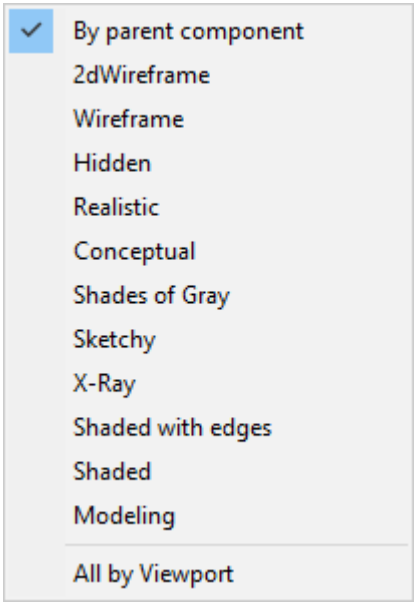
- Source: node that contains source object and its parameters. Source object can be presented as a component or subassembly with specific properties.
- Elements: node that contains all instances of source object.



Rysunek 29: Mechanical Browser

Component array context menu is displayed by a right click on the array node, while array properties are displayed automatically when selecting an array in the Mechanical Browser.

Tabela 6: Context Menu

Context menu	Description
Component array context menu	Right-click an array:  • **Dissolve**: Dissolve an array created in the current drawing (See the *BMDISSOLVE* command). • **Unlink**: Unlink an array from 3D solid by dissolving the underlying component-based feature. • **Hide** / **Show** : Hide or show the selected array. • **Exclude all inserts from section**: Set the *Sectionable* property of all similar inserts to *NO*. Define whether an insert is affected by the *VIEWSECTION* command. • **Visual Style**: display the **Visual Style** menu. Visual Styles that are saved in the current drawing are available.  • **By Viewport**: render the selected array according to the current viewport shademode (see the *SHADEMODE* command). • **All by Viewport**: Render all arrays according to the current viewport shademode (see the *SHADEMODE* command). • **Zoom to**: zoom to the selection set. • **Select**: select the array. • **Delete**: remove the selected array from the assembly. • **Collapse all**: collapse the main component and all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.

21.4.8.2 Related commands

Tabela 7: Related commands

Command	Description
3DARRAY	Create 3D arrays.
ARRAY	Create a polar or rectangular array of entities; display prompts at the Command line.
ARRAYCLASSIC	Create a static polar or rectangular array of entities through a dialog box.
ARRAYEDIT	Edit associative array entities and their source entities.
ARRAYPATH	Associatively distribute entity copies evenly along a path into multiple rows and levels.
ARRAYPOLAR	Associatively distribute entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.
ARRAYRECT	Associatively distribute entity copies into any number of rows, columns, and levels.
MINSERT	Insert arrays of blocks.

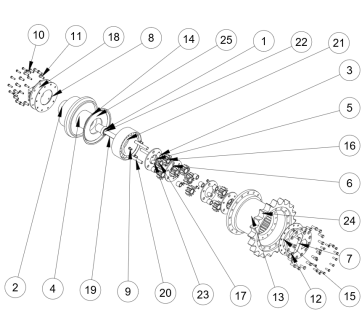
21.4.9 Exploded Representations

21.4.9.1 Overview

Use the *BMEXPLODE*, *BMEXPLODEMOVE*, *BMTRAILINGLINES*, and *VIEWBASE* commands.

The exploded representation functionality creates associative exploded representations of assemblies and does not modify the assembly itself. Exploded representations are stored in dedicated blocks, which can be edited and inserted according to your needs. You can also generate drawing views from the exploded representations.

Exploded representations are associative. Each part in an exploded representation is linked with the corresponding part in the assembly. Use the *BMBALLOON* command to update exploded representations to the current state of the assembly, as well as place balloon annotations on the corresponding drawing view(s).



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

Rysunek 30: Example of Exploded Representation

21.4.9.2 Levels of the exploded representation

Exploded representations contain references to the components.

2 levels of exploded representations are supported: Top and Bottom.

- **Top level**:

  The exploded representation contains references to the top-level components only. Top-level components are components inserted directly into the main assembly. A top-level component in an exploded representation is not exploded further - all its parts and subassemblies remain assembled. A top-level exploded representation allows you to see the basic composition of your assembly.

- **Bottom level**:

  The exploded representation contains references to the bottom-level components only. Bottom-level components are terminal parts, which do not contain other parts or subassemblies. Bottom-level exploded representations allow you to view all parts in isolation.

Uwaga: If **Bottom level** is selected and there are some middle-level components containing not only inserts of components, but also other entities, these entities will be lost in the exploded representation because it is not possible to reference them correctly. Those entities must be converted into corresponding bottom-level components first.

21.4.9.3 Algorithms

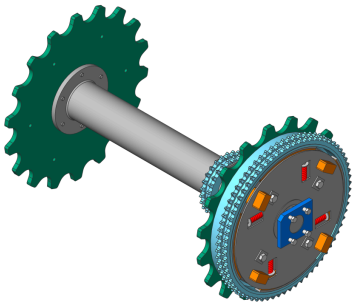
BricsCAD provides 3 algorithms to automatically create exploded representations and a manual algorithm.

Tabela 8: Algorithms

Algorithm	Description
Table by Type	Create a table-like explosion where components of the same type are grouped in rows.

Tabela 8: Algorithms

Algorithm	Description
Linear	Find the disassembly sequence of components in a given direction and order the components. ***Uwaga:*** This considers possible physical collisions between components. The components can be moved by the algorithm only if there are no other components (not yet moved), that block them.
Automatic	Find the disassembly sequence of components with respect to the assembly hierarchy. ***Uwaga:*** This considers all possible physical collisions. For each part or subassembly, the direction of movement is identified automatically.
Manual	Create an exact copy of the assembly ready for custom edit. In this mode, you can create a custom exploded representation without changing the main assembly.



Rysunek 31: Example: The main drive shaft assembly

Tabela 9: Example: main drive shaft assembly

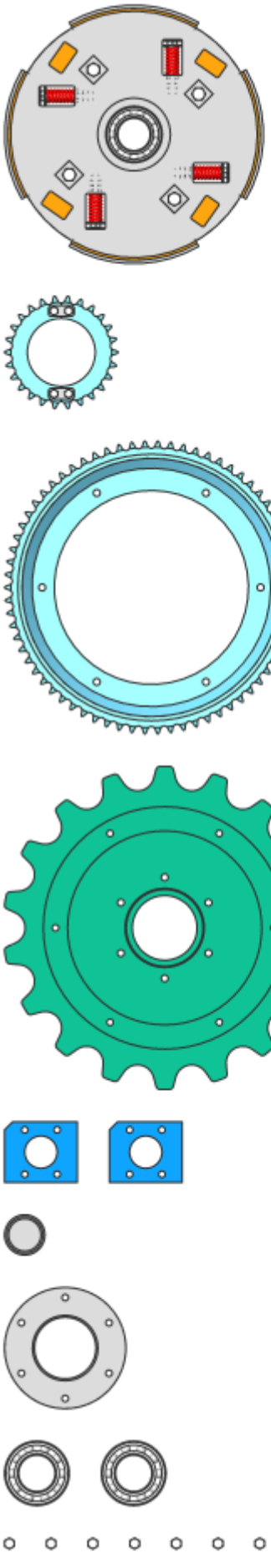
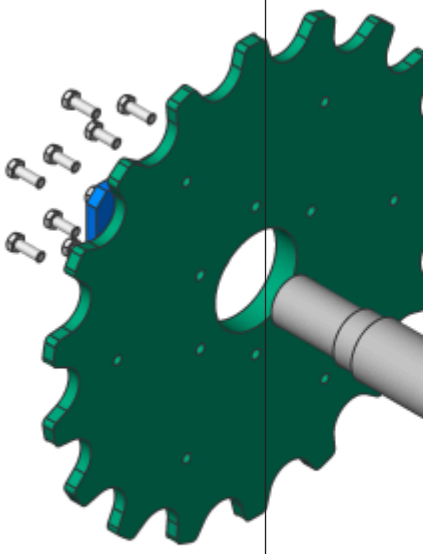
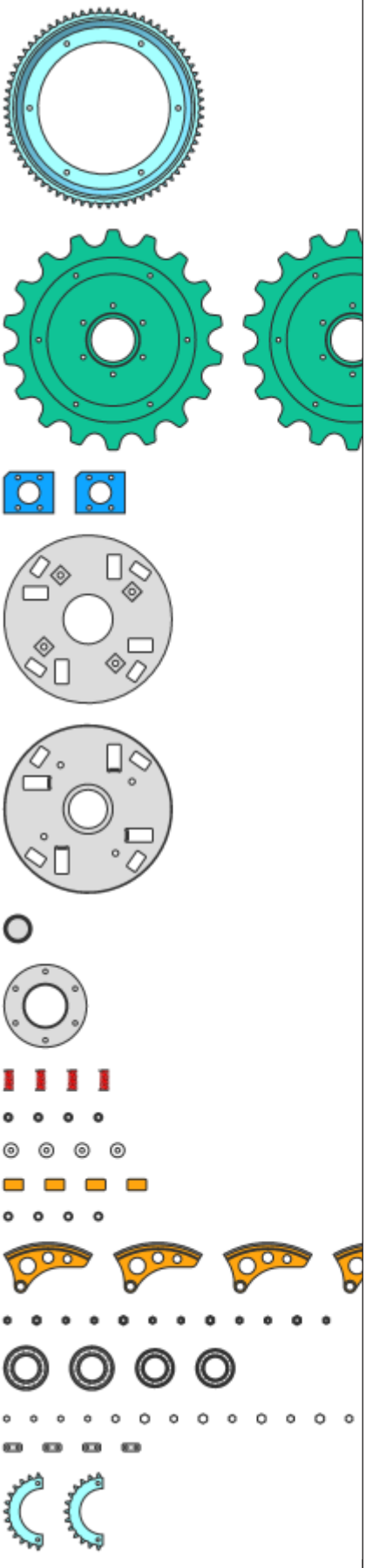
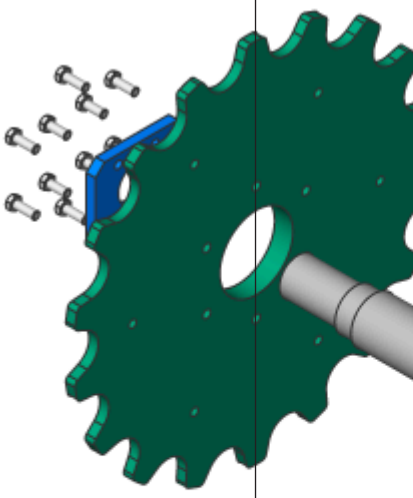
	Table by Type	Linear	Automatic
Top level			

Tabela 9: Example: main drive shaft assembly

	Table by Type	Linear	Automatic
Bottom level			

21.4.9.4 Steps and animations

Exploded views may contain one or several steps, which describe a particular assembly or disassembly sequence. Each step corresponds to a set of components that must be moved at this particular stage in order to assemble or disassemble the main assembly. All exploded representation algorithms automatically create all the required steps. You can also delete, merge, split, move, or add extra steps.

Each step has a unique name that can be used to describe the step.

It is possible to animate a particular step, as well as the entire sequence of steps, both indirect and reverse directions (to get disassembly and assembly animations). In order for the animation to work properly, an initial step that defines the initial state of the assembly must be present in the exploded representation.

Use the **Auto hide** property of the exploded representation to automatically hide components that are not important for a particular step during the animation.

21.4.9.5 Edit exploded representations

Each exploded representation is stored as a block. This allows you to edit the exploded representation with the block editor.

Ważne: Do not use the *REFEDIT* command to edit exploded representations. This command is incompatible with exploded representation steps.

To open the exploded representation for edit:

1 Right-click the required exploded representation in the Mechanical Browser.

2 Select **Edit** from the context menu.

The block editor will open automatically.

Wskazówka: As an alternative, you can open the exploded representation for edit at the end of the *BMEXPLODE* command.

When the exploded representation is open for edit, you can change the state of the current step. By default, this is the last step made by the algorithm. However, you can change this in two ways:

- Use the **Current step** property of the exploded representation in the browser.
- Right-click the desired step in the Mechanical Browser and select **Apply** from the context menu.

Uwaga: If you change the current step, you will discard all unsaved modifications, as well as any operation on steps. Make sure that all modifications are saved before you change the current step or perform any other operation.

Use the *DMMOVE*, *MOVE*, *DMROTATE*, and *BMEXPLODEMOVE* commands to move components, inside the exploded representations, to the required positions. The *BMEXPLODEMOVE* command automatically adds all necessary steps after the current step. For other operations, save the current step after the edit is complete. Only the difference between the current step and the previous one is stored.

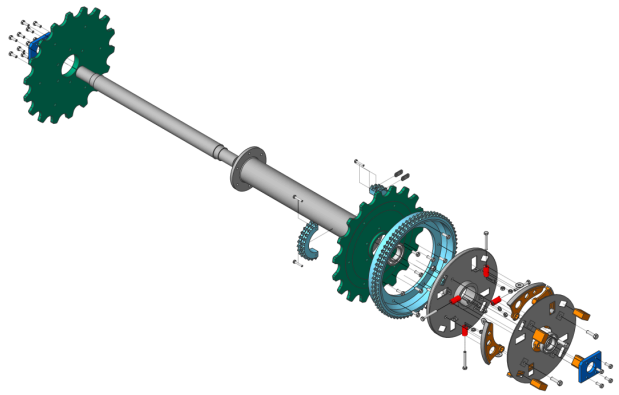
It is also possible to change the exploded representation by performing operations on steps.

Example: if you remove a step, all components moved at this step will be returned to their respective positions before that step.

Save the current step once you have finished the edit. If not, changes will be discarded.

21.4.9.6 Trailing lines

Trailing lines help explain the relationship between parts. It is a 3D polyline that represents a trajectory and displays the movement of a part during the disassembly process.



Rysunek 32: Example of trailing lines

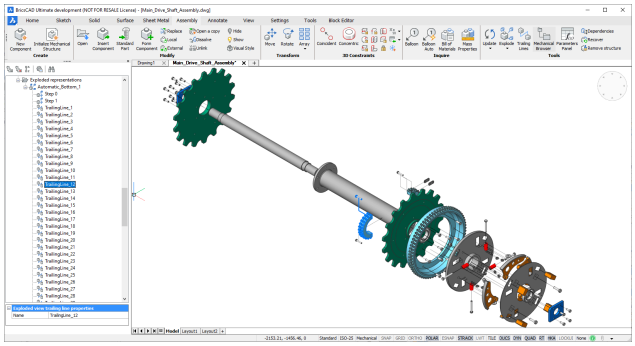
Use the *BMTRAILINGLINES* command to add trailing lines to an exploded representation that is open for edit. This tool creates all trailing lines. The information from the exploded view steps is used to do this.

You can generate trailing lines for all parts or for a subset of parts. For a subset, trailing lines will be built for parts from that subset only, although only necessary trailing lines will be added. Some parts may not have trailing lines at all. You can also manually select 2 parts to add a trailing line between.

When a trailing line is computed, the algorithm automatically determines the location of the part and the location from where the part was removed to compute the trajectory. It considers the movements of both parts.

Each trailing line is displayed in the mechanical browser. With the mechanical browser, you can see all trailing lines in the model; highlight, select and zoom to parts connected with the selected line; rename or remove the trailing line.

You can edit the trailing line(s) with the standard 3D polyline edit tools.



Rysunek 33: Exploded view trailing line properties

21.4.9.7 Create an exploded representation

Use the *BMEXPLODE* command to create an exploded representation of the current assembly.

Uwaga: You must have at least 1 insert of a part in the assembly for the command to work.

1 Choose one of the following actions to launch the *BMEXPLODE* command:

- Click the **Explode** tool button on the **Tools** panel of the **Assembly** tab.
- Click the **Explode** tool button on the **Assembly** toolbar.
- Select **Explode** in the **Assembly** menu.
- Type **bmexplode** in the Command line.

You are prompted: Select explosion algorithm [Table by types/Linear/Manual/Settings]:

2 If applicable, set the *Level* and the *Name* of the exploded representation using the **Settings** option.

3 Select the algorithm.

You are prompted: Select exploded view behavior [Edit/Generate drawing views/Finish]:

4 Perform one of the following actions:

- Select **Edit** to edit the exploded representation.
- Select **Generate drawing views** to generate drawing views of the exploded representation.
- Select **Finish** to finish the command.

21.4.9.8 Generate drawing views of the exploded representation

1 Launch the *VIEWBASE* command.

You are prompted: Select objects or [Entire model/preseTs/Special views] <Entire model>:

2 Select the **Special views** option.

You are prompted: Select view [Exploded view/Unfolded view/Back]:

3 Select the **Exploded view** option.

A dialog box to select the exploded representation appears.

4 Select the exploded representation.

You are prompted: Enter new or existing layout name to make current <Layout1>:

5 If applicable, enter the *Layout name* of the layout for the exploded representation.

6 Position the generated drawing views of the exploded representation in the paper space layout.

Uwaga: To use balloons (see the *BMBALLOON* command) with drawing views of exploded representations, the *GENERATEASSOCVIEWS* system variable must be *ON*.

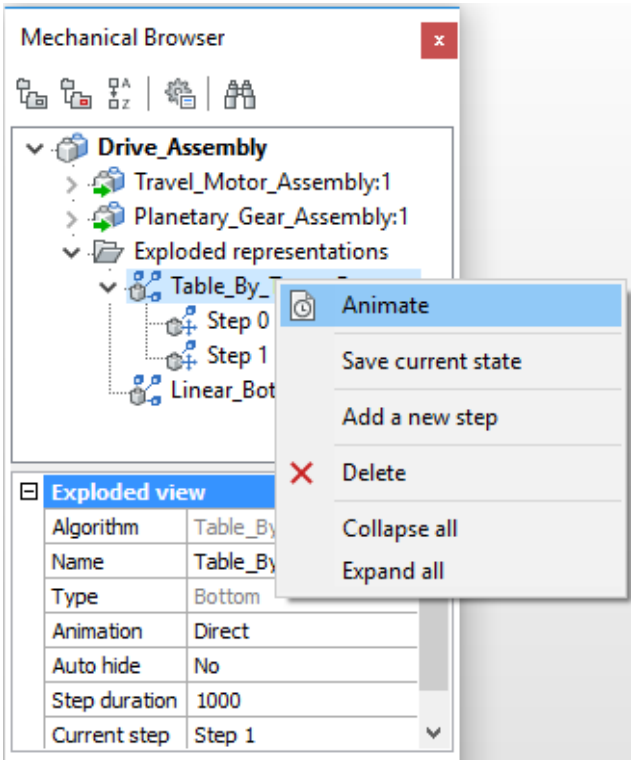
21.4.9.9 Update an exploded representation

Run the *BMUPDATE* command to synchronize exploded representations with the current state of the assembly.

If applicable, adjust the exploded representation. Such adjustments may be required because of the addition and/or deletion of some parts.

Uwaga: The position of parts in exploded representations created in **Manual**and **Linear** modes are not updated automatically.

21.4.9.10 Manage exploded representations



Rysunek 34: Manage exploded representations

Open the Mechanical Browser to see all exploded representations in the document. All exploded representations will be listed under the **Exploded representations** group. Open the context menu for the selected exploded representations to see all supported operations in this context.

Properties of exploded representations

Each exploded representation has a set of properties. You can edit some of the properties

Tabela 10: Properties of exploded representations

Property	Description
Algorithm	The name of the algorithm used to create this exploded representation.
Name	Name of the exploded representation. This must be unique for all exploded representations, as well as for all the blocks in the model.
Type	Type of the exploded representation: *Top* or *Bottom*.

Tabela 10: Properties of exploded representations

Property	Description
Animation	The direction of animation. Choose 1 of 2 values: *Direct* (from start to end) or *Reverse* (from end to start).
Auto hide	If set to *Yes*, all unimportant parts for a particular step will be hidden during the animation.
Step duration	Default duration of each step during the animation (in milliseconds). The default value is 1000, which is equal to 1s duration.
Current step	The current step that corresponds to the state of the exploded representation.

Operations on exploded representations

The supported operations depend on whether or not the exploded representation is opened for edit.

If the exploded representation is **not opened for edit**, the following operations are possible:

Tabela 11: Operations on exploded representations

Operation	Description
Edit	Open the exploded representation for edit. ***Wskazówka:*** You can also edit the exploded representation with the *BEDIT* command.
Delete	Remove the exploded representation and the associated block from the document.

If the exploded representation is **opened for edit**, the following operations are possible:

Tabela 12: Operations on exploded representations

Operation	Description
Animate	Animate the entire sequence of steps.
Save current state	Save the current positions of parts in the current step.
Add a new step	Add a new step after the last one. ***Uwaga:*** The new step will be automatically set as *current*. All unsaved modifications of the exploded representation will be lost.

Tabela 12: Operations on exploded representations

Operation	Description
Delete	Remove the exploded representation and the associated block from the document.

In addition, all steps will be listed in the Mechanical Browser for this exploded representation.

21.4.9.11 Manage exploded representation steps

Open the Mechanical Browser and open the required exploded representation for edit. All steps will be listed under the corresponding node in the tree.

Properties of steps

Tabela 13: Properties of steps

Property	Description
Name	The name of the step. Must be unique for this exploded representation.
Duration, ms	The duration of the step. By default, this is equal to the Step duration set for the exploded representation.
Duration type	Specify if the duration must be taken from the exploded representation, or if it is specific for this particular step.
Duration value (only for Absolute duration type)	The duration for this step in milliseconds.

Operations on steps

Tabela 14: Operations on steps

Operation	Description
Apply	Set the step as the current and update the parts in the exploded representation according to this step.
Animate	Animate the selected step.
Set as previous	Clear the step, so the positions of all parts will be the same as in the previous step.
Merge with previous	Merge this and the previous step. The previous step is removed.

Tabela 14: Operations on steps

Operation	Description
Split	Split the step to a sequence of steps, each of them corresponding to exactly one part.
Move step up	Move the step one position up in the tree.
Move step down	Move the step one position down in the tree.
Delete	Completely remove the step as well as the associated movements of parts. After that, the positions of parts are updated in the exploded representation.

21.4.9.12 Edit exploded representations

1 Select the exploded representation in the Mechanical Browser.

2 Right-click the exploded representation and select **Edit** from the context menu.

Uwaga: As an alternative, you can run the *BEDIT* command and select the block with the exploded representation.

3 Perform the necessary operations to get the desired exploded representation.

4 Save the current step.

5 Save changes using the *BCLOSE* command with the option *Save*.

Uwaga: Do not use the *Discard* option of the *BCLOSE*command, as it can lead to an inconsistent state of the block and exploded representation steps. Use *Undo* instead.

21.4.9.13 Add trailing lines

1 Open the exploded representation for editing.

2 Run the *BMTRAILINGLINES* command.

3 Select the subset of parts to create trailing lines for, or select the entire model.

4 Select the points of the parts to be used in order to create the trajectory: origin or center.

5 If applicable, edit the resulted trailing lines.

6 Save changes.

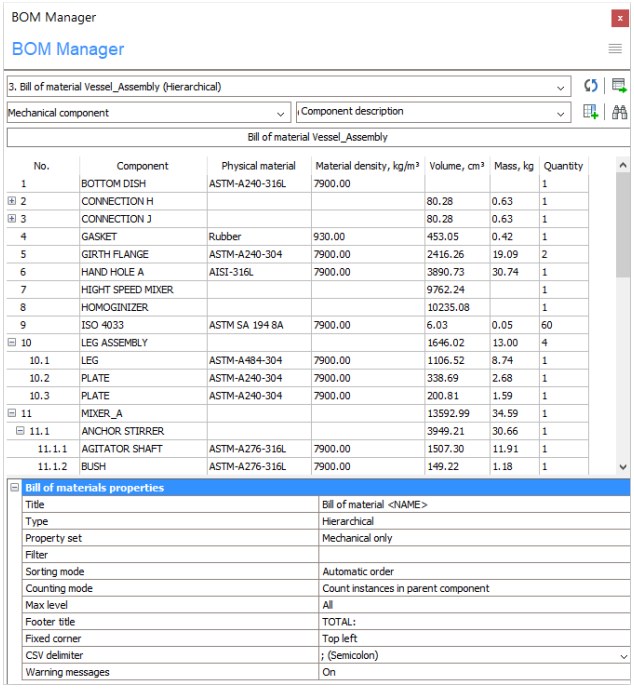
21.4.10 Bill of Materials Manager

21.4.10.1 Overview

Use the *BMBOMPANELOPEN*, *BMBOMPANELCLOSE*, and *BMBOM* commands.

The Bill of Materials Manager allows you to browse, create and edit bills of materials (BOMs) in the current drawing. Use the manager to interactively configure (new and existing) bills of materials. You can save the

configuration of a bill of materials as a template file and reuse it later in other drawings. You can also add a bill of materials that is not linked to a table in the drawing.



Rysunek 35: BOM Manager

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.

21.4.10.2 Open the Bill of Materials Manager

Do one of the following:

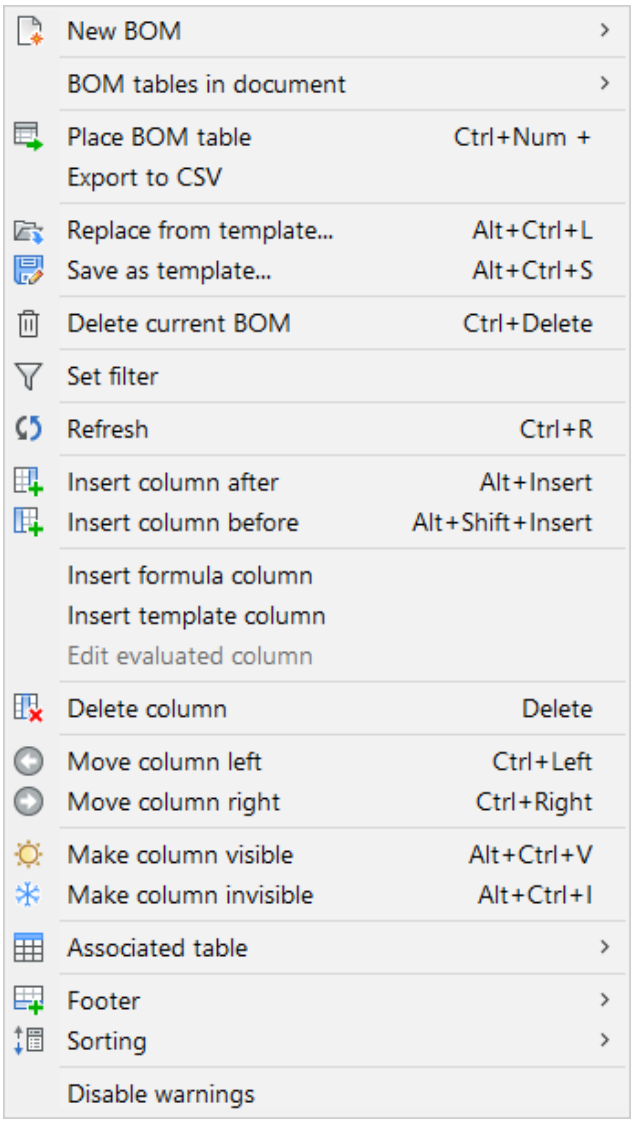
- Place the cursor on a toolbar, right-click, choose **Panels**, and choose **BOM Manager** from the context menu.
- Launch the *BMBOMPANELOPEN* command.

21.4.10.3 Tools

Most of the configuration operations in the manager are available in the main menu. A context menu is available for selected columns or rows. You can also change the properties of the selected columns and the BOM itself in the properties section of the manager. To work with a particular column, select it in the BOM grid. Multiple columns can be selected simultaneously.

Main menu

Click **Menu** (☰) at the top right corner of the BOM Manager panel.



Rysunek 36: Main menu

Tabela 15: Main menu

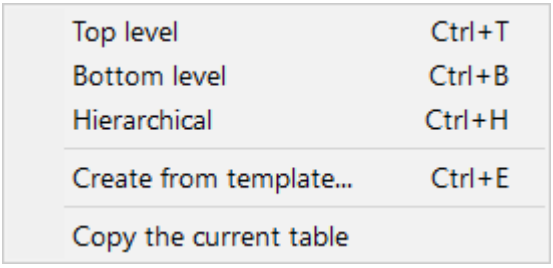
Icon	Tool	Description
	**New BOM**	Open a submenu:  **Rysunek 37:** Submenu • **Top level**: create a new, top-level BOM. • **Bottom level**: create a new bottom-level BOM. • **Hierarchical**: create a new hierarchical BOM. • **Create from template**: open a dialog box to select a BOM configuration template file. • **Copy the current table**: create a new BOM based on the current table.
	BOM tables in the document	Open a submenu:  **Rysunek 38:** Submenu • **Go to first table**: switch to the first BOM table in the document. • **Go to previous table**: switch 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**: switch to the last BOM table in the document. • **<Name of the BOM>**: switch to the corresponding BOM table in the document.

Tabela 15: Main menu

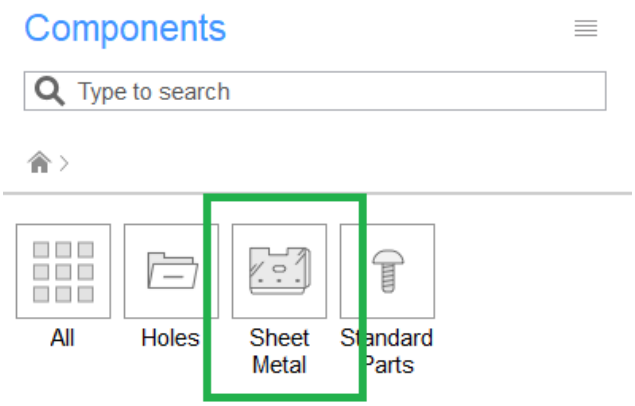
Icon	Tool	Description
	Place/update BOM table	Start the interactive placement of a table in the drawing. If the table is already placed in the drawing, it updates the table. ***Uwaga:*** All editing operations update the associated table in the drawing automatically.
	Export to CSV	Save the content of the current table to a CSV file. Open a dialog box to select the location for the CSV file:  ***Uwaga:*** You can adjust the separator for CSV values using the table properties. **Rysunek 39:** Select file to export BOM table
	Replace from template	Replace the current BOM table configuration with the selected template. Open a dialog box to select the template file.
	Save as template	Save the configuration of the current BOM table as a template. Open a dialog box to select the location for the new template file.
	Delete current BOM	Delete the current BOM table from the document.

Tabela 15: Main menu

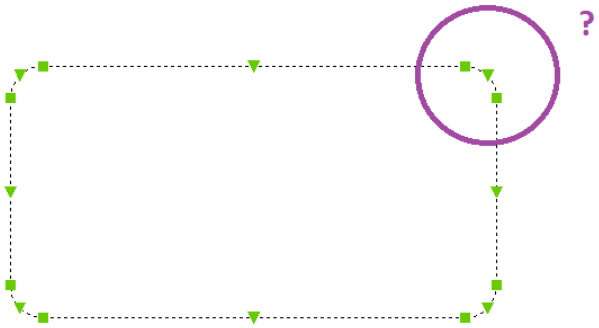
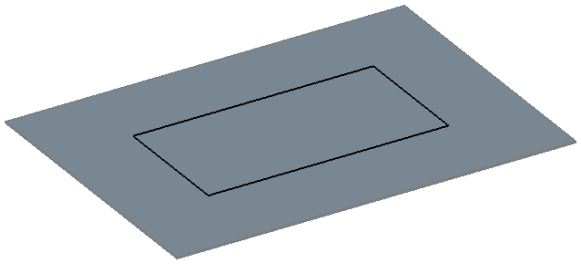
Icon	Tool	Description
	Set filter	Opens a dialog to configure the main filter:  **Rysunek 40:** Filter Configurator
	Refresh	Update the current BOM, and list all BOM tables in the drawing.
	Insert column after	If a column is selected: insert a column from the **Column selector** after the selected column in the table. If no column is selected: add a new column after the last column in the table.
	Insert column before	If a column is selected: insert a column from the **Column selector** before the selected column in the table. If no column is selected: add a new column as the first column in the table.
	Insert formula column	Open a dialog box to configure a formula column:  **Rysunek 41:** Formula column Click **OK** to add a new formula column after the selected one. If no column is selected, a new formula column is added after the last column in the table.

Tabela 15: Main menu

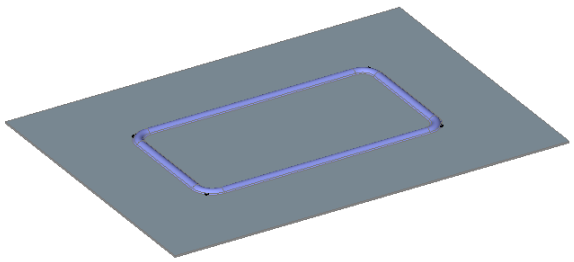
Icon	Tool	Description
	Insert template column	Open a dialog box to configure a template column:  **Rysunek 42:** Template column Click **OK** to add a new template column after the selected one. If no column is selected, a new template column is added after the last column in the table.
	Edit evaluated column	Open a dialog to edit the selected formula or template column.
	Delete column	Removes the selected column from the table. You can also select a column in the **BOM grid** and hit the **Delete** key to remove a column.
	Move column left	Move the selected column(s) one column to the left.
	Move column right	Move the selected column(s) one column to the right.
	Make column visible	Show the selected column in the associated table in the drawing.
	Make column invisible	Remove 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.

Tabela 15: Main menu

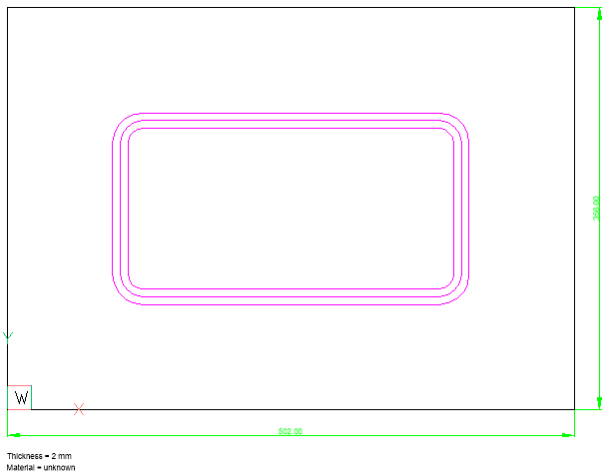
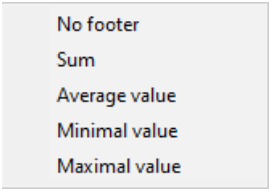
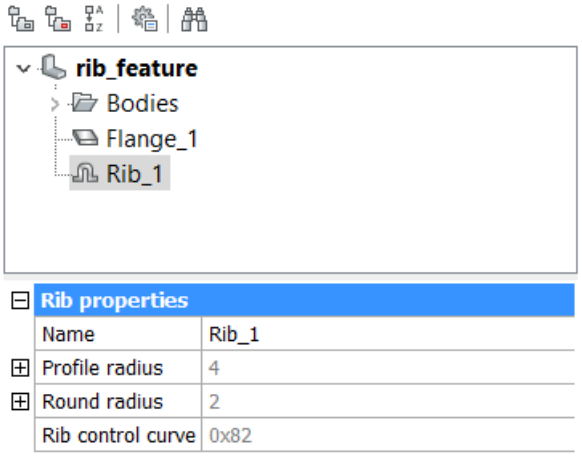
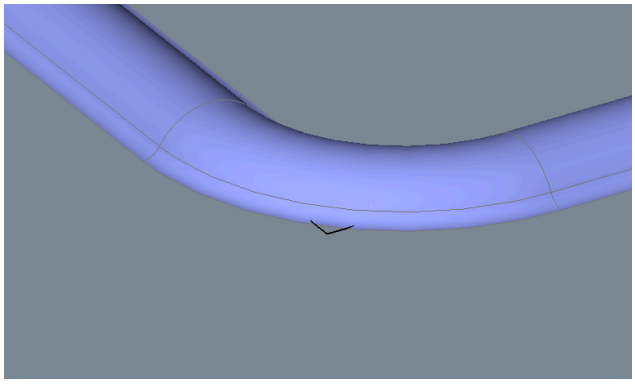
Icon	Tool	Description
	Associated table	Open a submenu:  **Rysunek 43:** Submenu • **Zoom to**: zoom to the table. If necessary, set the table layout as the current one. • **Select**: select the table. If necessary, set the table layout as the current one. • **Delete**: delete the table from the document. • **Detach**: detach the table. The table in the document is no longer updated when the BOM table is modified.
	Footer	Open a submenu:  **Rysunek 44:** Submenu For each of the selected columns: • **No footer**: remove the footer. • **Sum**: display the sum in the footer. • **Average value**: display the average in the footer. • **Minimal value**: display the minimum value in the footer. • **Maximal value**: display the maximum value in the footer. For more details, see section Footer in Bill of Materials.

Tabela 15: Main menu

Icon	Tool	Description
	**Sorting**	Open a submenu:  **Rysunek 45:** Submenu • **Automatic order**: sort the table automatically. • **Natural order**: sort the table by the natural order of component instances in the document. • **Ascending order**: sort the selected columns in ascending order. • **Descending order**: sort the selected columns in descending order. For more details, see section Sorting in Bill of Materials.
	Disable/Enable warnings	Control the display of warning messages during the creation or update of BOM tables.

Toolbars



Rysunek 46: BOM Manager

Tabela 16: BOM Manager

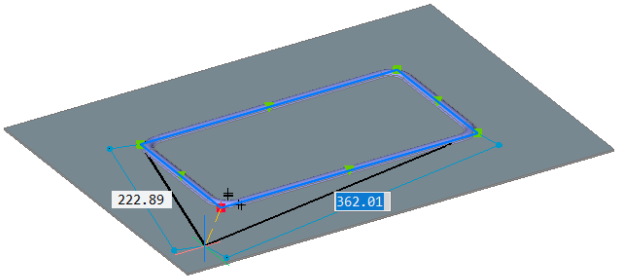
Icon	Tool	Description
	**Main menu**	Open the main menu.
	BOM Selector	Select the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated to the current state of the drawing.
	**Refresh**	Update both the current BOM and the list of all BOM tables in the drawing.
	**Place/update table**	Start the interactive placement of a table in a drawing. Update the table if it already exists. ***Uwaga:*** All edit operations automatically update the associated table in the drawing.
	Category selector	Select a properties category.
	**Column selector**	Select the column to be added to the current BOM. You can also use this to enter a formula for a formula field. To add a formula, type an expression starting with "=", such as "=<Mass>*<Quantity>". The formula may refer to other columns. Use their display names enclosed in <>.
	**Insert column after**	If a column is selected: insert the column from the **Column selector** after the selected column in the table. If no columns are selected: add the new column after the last column in the table.
	**Find**	Toggle the search field on and off. You can also use **Ctrl+F**.
	Search field	Enter a search string; wild cards are supported.

Tabela 16: BOM Manager

Icon	Tool	Description
	Find previous	Select the previous match in the BOM table. You can also use **Ctrl+F3**.
	Find next	Select the next match in the BOM table. You can also use **F3**.
	Select all found	Select all matches in the BOM table.

Column context menu

Right-click a column label to open the context menu. The column is added to the current selection.



Rysunek 47: Column context menu

Tabela 17: Column context menu

Icon	Tool	Description
	Add template column from selected	Open a dialog to configure a new template column. The selected columns will be automatically added to the template expression.
	Delete column	Remove the selected column(s) from the table. You can also select a column in the **BOM grid** and press the **Delete** key.
	Footer	Open a submenu for the selected column(s): • **No footer**: remove the footer. • **Sum**: display the sum in the footer. • **Average value**: display the average values in the footer. • **Minimal value**: display the minimum value in the footer. • **Maximal value**: display the maximum value in the footer.

Tabela 17: Column context menu

Icon	Tool	Description
	**Sorting**	Open a submenu: • **Automatic order**: sort the table automatically. • **Natural order**: sort the table by the natural order of component instances in the document. • **Ascending order**: sort the selected columns by ascending order. • **Descending order**: sort the selected columns by descending order.
	Aggregate function	Open a submenu for the selected column(s): • **No function**: remove the aggregate function. • **Sum**: display the sum of all values in the group. • **Average value**: display the average of all values in the group. • **Minimal value**: display the minimum of all values in the group. • **Maximal value**: display the maximum of all values in the group. • **Concatenation**: display a concatenation of all values in the group in ascending order separated by a specified delimiter ('; ' by default). • **Concatenation with count**: display 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 ('; ' by default).
	Column role	Open a submenu for the selected column: • **Number**: the column is used as a source of numbers by the *BMBALLOON* command. • **Name**: the column is used as a source of names by the *BMBALLOON* command. • **Quantity**: the column is used as a source of quantities by the *BMBALLOON* command. • **Regular**: the column has no specific role.

Tabela 17: Column context menu

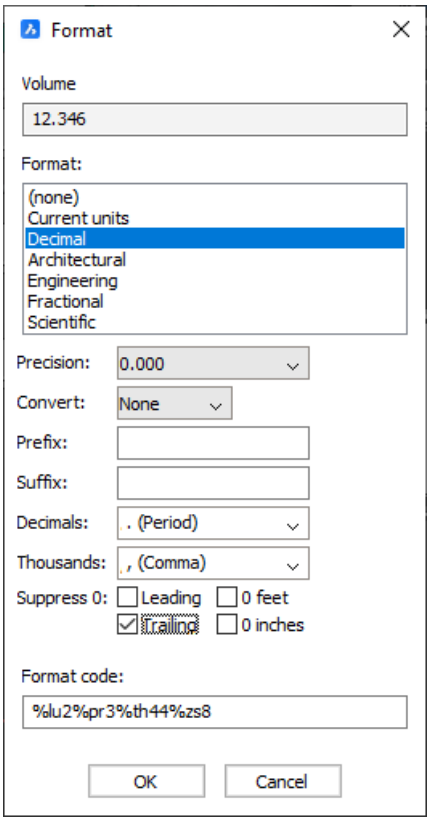
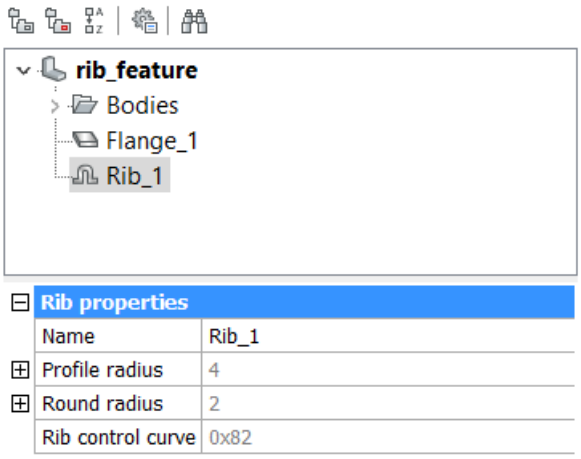
Icon	Tool	Description
%..	**Format**	Set the specified format for selected column(s). Open a dialog box:  **Rysunek 48:** Format

Tabela 17: Column context menu

Icon	Tool	Description
	Set column filter	Open a dialog box to configure the column filter:  **Rysunek 49:** Column Filter
	Make column visible	Display the selected columns in the associated table in the drawing.
	Make column invisible	Remove 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.
	Fit column(s)	Fit the width of the selected column(s) to their content. The column width automatically fits the content unless the width has been changed manually.
	Fit all columns	Fit the width of all columns to their content. The column width automatically fits the content unless the width has been changed manually.

Row context menu

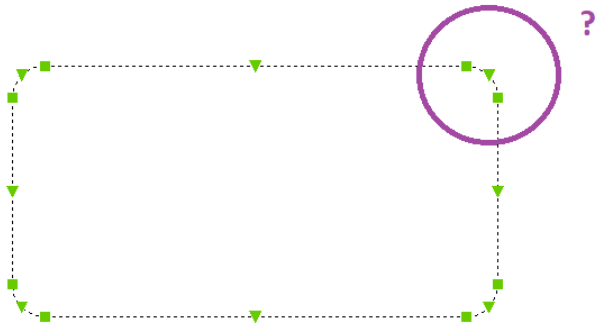
Right-click on any cell to open the context menu.

Rysunek 50: Context menu

Tabela 18: Row context menu

Icon	Tool	Description
	**Zoom to**	Zoom to the select part(s) in the model.
	**Select**	Select the part(s). Replace an existing selection.
	**Add to Selection**	Add part(s) to the selection.
	**Remove from Selection**	Remove the parts from the selection.
	Remove highlighting	Remove highlighting.
	Clear selection	Clear a selection.
	Collapse	Collapse the subassembly group in the BOM grid.
	Expand	Expand the subassembly group in the BOM grid.
	Expand this level	Expand all subassembly groups at the same level of the assembly tree as the current one.
	Collapse this level	Collapse all subassembly groups at the same level of the assembly tree as the current one.
	Expand all	Expand all subassembly groups in the current BOM.
	Collapse all	Collapse all subassembly groups in the current BOM.
	Limit by this level	Set the current level as the maximal level for hierarchical BOM tables.
	Show all levels	Remove the maximal level limit for hierarchical BOM tables.

BOM Title



Rysunek 51: BOM Title

Displays the title of the current BOM, as displayed in the drawing. Left-click on this control to edit the drawing. In this mode, the **BOM title** control shows the **title template** instead. The **title template** is a string that may contain a *<NAME>* placeholder, which will be replaced by the assembly name.

Grid

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

Rysunek 52: Grid

The grid displays the content of the selected BOM table. Each row is linked with its corresponding instances. When a row is selected, the corresponding parts are highlighted in the drawing. Use the context menu, to select corresponding parts, add or remove them from the current selection, and to zoom to those parts. Double-click on any cell in the row to zoom to the corresponding parts.

Moving a column

To move a column in the BOM table drag and drop it to its new position.

Visible and invisible columns



Rysunek 53: Visible and invisible columns

You can make any column in the BOM table invisible, so it does not show in the resulting table. Invisible columns remain visible in the BOM grid with a hatched background. You can change the visibility of columns in the main menu, the column context menu, or by editing the **Visible** property of a column in the **Properties grid**.

Width adjustment

You can adjust the width of any column in the BOM grid, which is independent from the width in the associated table. If the width of the column is less than needed to display the text of the cell, the height of the corresponding row increases to fit the text.

When the column width is adjusted manually, the width remains the same regardless of the content of the table. The width is also stored in the BOM configuration. All adjusted column widths remain the same even when the drawing is closed and reopened.

To fit the column width to the content automatically, do one of the following:

- Select the column(s), right-click the context menu and select **Fit column(s)**.
- Double-click the separator in the column header.

Sorting a BOM

Use the grid to control how the table of content should be sorted. Double-click a column label to change the order between ascending, descending, and unordered. If a column is used to sort the table, its label has a symbol added (▲ for ascending and ? for descending). Double-click the label to revert the sorting order.

You can use multiple columns to sort. The sort symbols of these columns have a subscript to denote the sort order. For example, if you sort by quantity in descending order and then by component name in ascending order, the Quantity column has a ?? symbol and the Component column has a ▲? symbol.

Expand/collapse groups

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

Rysunek 54: Expand/collapse groups

In hierarchical BOMs, each subassembly forms a group that can be collapsed or expanded in the BOM grid.

To collapse or expand a group, do one of the following:

- Click on the node icon in the first column of the BOM grid.
- Right-click on the row, then select **Collapse** or **Expand** from the context menu.

You can also collapse or expand all groups or all groups corresponding to the same assembly level using the context menu.

Uwaga: The subassembly group status (collapsed or expanded) is not stored within the document. When the BOM table is closed and reopened, all subassembly groups are expanded by default.

Properties

Rysunek 55: Bill of materials properties

The properties grid is located at the bottom of the manager's panel. To adjust its size, use the splitter control, located between the grid and the properties grid.

The properties grid contains the properties of the current table; title, type, filter, sorting mode and footer title. If any columns are selected, the grid will also display their common properties. You can change any common property for all the selected columns at once. Column properties change the title, visibility, and width of the column.

For numerical values, you can set the footer type, unit placement, and the units mode (if the column has units that can be changed).

The **unit placement** defines where the units are displayed: in the column title, in the same cell as the value itself, or in its own column.

The **units mode** defines how the displayed unit should be obtained; the best unit for all values, the best unit to display a particular value in the cell, or a specified unit. You can choose the **Units mode** for distance, area, volume, or mass properties.

21.4.10.4 Filter

It is possible to add a custom filter to the current BOM. To apply a filter, do one of the following:

- Use the **Filter** property in the **Bill of materials properties** section of the **Property grid**.
- Use the **Set filter** command from the **Main menu**.

In both cases, the *Filter Configurator* dialog box is displayed:



Rysunek 56: Filter Configurator

Each row in this dialog box represents an elementary condition. It is possible also to add groups to combine several elementary conditions and/or groups. Groups allow to create complex filters.

Adding a new elementary condition

Do the following:

1 If applicable, choose the required category to limit the list of properties to that particular category.

2 Select the required property from the list of properties. When selected, a condition for the property is added automatically.

3 If applicable, the required condition type:
- Not equal to
- Less than
- Greater than
- Not less than
- Not greater than

4 Set the target value. For text values, you can select the target value from the drop-down list of all possible values. For numerical values, you can specify the unit of the target value. The value will be automatically converted to the document unit.

Removing one or more conditions

Do the following:

1 Select the conditions to remove. You can use the Ctrl key to select several conditions, and Shift key to select a range of conditions.

2 Click the **Delete** tool (✕), or right-click the row you want to delete and select **Remove condition** in the context menu.

Adding a new group of conditions

Do the following:

1 Click the **Add** tool (✚).

2 Select **Add "All of" group** to add a new "All of" group, or select **Add "Any of" group** to add a new "Any of" group.

3 Add the necessary conditions.

Grouping selected conditions

Do the following:

1 Select the conditions to remove. You may use the Ctrl key to select several conditions, and Shift key to select a range of conditions.

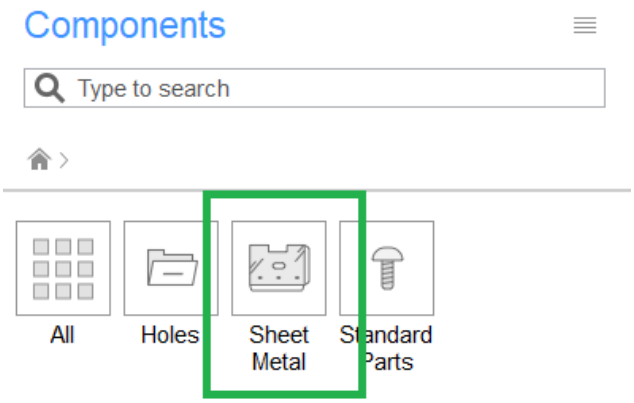
2 Right-click the row you want to group.

3 Select **Form "All of" group from selected** to form a new "All of" group, or select **Form "Any of" group from selected** to form a new "Any of" group.

To apply the filter, click **OK**.

21.4.10.5 Column filters

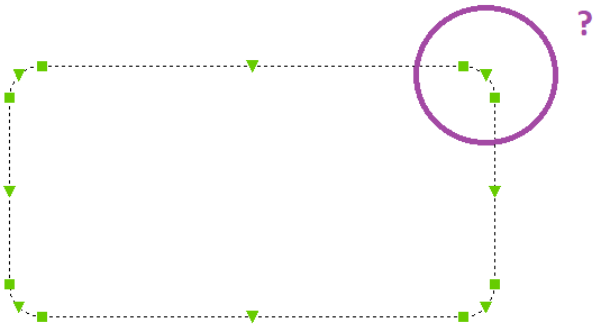
Column filters allow you to set the conditions on which rows to include in the table. Each column filter sets a condition that column values should meet in order to be included. For example, you may set a column filter to include only parts with quantities larger than a certain number, or parts whose names contain **Nozzle**. A column can have no more than one filter. However, it is possible to simultaneously set filters for as many columns as you need. Columns with active filters are marked with a filter icon ( ) in their title.



Rysunek 57: Example of columns with active filters

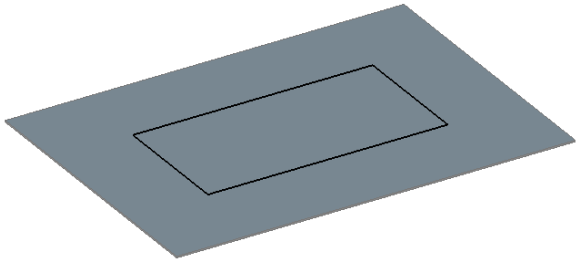
To add a column filter, right-click the column title and select **Set column filter** in the context menu.

A *Column filter* dialog box is displayed.



Rysunek 58: Column filter

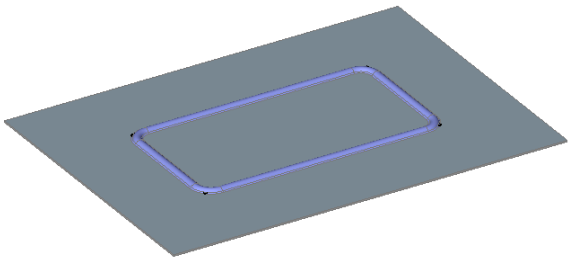
Select the required filter type. Next, you will need to set the target value(s) for the filter, depending on the filter type and the column content.



Rysunek 59: Target values for the filter

For columns with numerical content, you can set the unit for the target value. The target value will be automatically converted to the document unit internally.

For text filters, you can choose between **ignore case** and **match case** options. **In** and **Not in** filters show you the list of all possible values for this column (with respect to already applied filters). Check the values that you want to be included or excluded correspondingly. You can also check all values, uncheck all values, or invert the current selection by using the corresponding buttons on toolbars or commands from the context menu.



Rysunek 60: Column Filter

To apply the column filter, click **OK**.

To remove a column filter, right-click on the column title and select **Remove column filter** from the context menu.

For more details, see section Filtering in Bill of Materials.

21.4.10.6 Sort

BOM tables support several sort modes (for more details, see section Sorting in Bill of Materials).

To switch between these modes, use the **Sorting mode** property in the **Bill of materials properties** section of the **Properties grid**. Use the **Sorting mode** drop-down menu or double-click the column labels in the **Grid** to toggle between ascending, descending or natural order for the respective column.

21.4.10.7 Column units

It is possible to change the **Unit placement** and **Units mode** for most numerical properties with units. Select the required columns in the **Grid** and use the **Column properties** section of the **Properties grid** to change the **Unit placement** and the **Units mode**.

For more details, see section **Column units** in Bill of Materials.

See also section **Changing the units and unit placement modes for a column**.

21.4.10.8 Evaluated columns

A BOM supports evaluated columns. There are 2 types of evaluated columns: formula and template columns. For more details about formula fields, such as expression syntax, see section **Evaluated columns** in Bill of Materials.

See also section **Adding an evaluated column**.

Formula columns

To add a formula column, do the following:

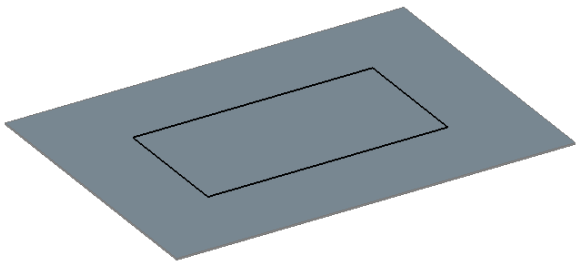
1 Select the **Evaluated column** category using the **Category selector**.

2 Select **Formula column** in the **Column selector**.

3 Use **Insert column after** or **Insert column before** tools.

As an alternative, you can use the **Insert formula column** option from the **Main menu**.

In both cases, an *Expression configurator* dialog box is displayed:



Rysunek 61: Expression configurator

This dialog box allows you to configure the title, unit and expression for the formula column. It also displays a preview of the formula results. The expression field supports autocompletion.

To add a reference to a column or a property to the expression at the cursor position, select the required property from the list of properties. You can also click the **Add column reference** tool ( ) and select the property from the menu, or click the **Add function** tool ( ) to select the required function.

To add a formula column to the BOM table, click **OK**.

Template columns

To add a template column, do the following:

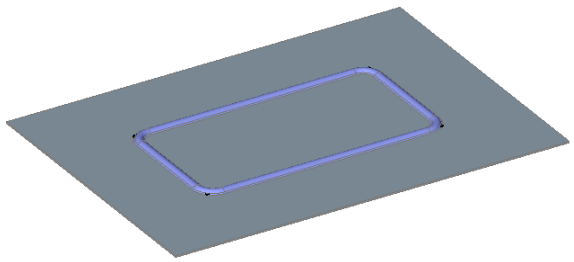
1 Select *<Evaluated column>* category using the **Category selector**.

2 Select *<Template column>* in the **Column selector**.

3 Use **Insert column after** or **Insert column before** tools.

As an alternative, you can use the **Insert template column** option from the **Main menu**.

In both cases, a *Template configurator* dialog box is displayed.



Rysunek 62: Template configurator

This dialog box allows you to configure title, expression and, if necessary, unit for your template column. It also displays a preview of the results. The expression field supports autocompletion.

Template columns may contain more than one template subexpression. In that case, they are evaluated from first to last, and the first subexpression yielding in non-empty result is used. The **Template configurator** allows you to add, delete and switch between template subexpressions using the toolbar. It also allows to change the order of template subexpressions.

Uwaga: In the expression field, all subexpressions are shown at once, where each subexpression is enclosed in {} brackets.

To add a subexpression, click .

To delete the current subexpression, click .

Uwaga: When the last subexpression is removed, it will be replaced with an empty one.

To change the current subexpression, select the number of the required subexpression from the drop-down list.

Use and to change the order of the subexpressions.

The currently selected subexpression can be configured using the **Template** section. Each column in this section represents an item of the template expression, which you can change.

The types of template items are:

- **Column reference**: represents a reference to a BOM column or a property. To add a column reference, select the required column from the list of properties. It will be automatically added after the last selected column, or, if no column is selected, after the last column. If necessary, a **Delimiter** will be added in front of the **Column reference** automatically.
- **Delimiter**: represents a separator between 2 items. The delimiter text will appear in the final result only if the text on both sides of it is not empty. For example, if you have PartNumber and Length column references with the "X" delimiter between them, the "X" will be added to the final text only if both PartNumber and Length are not empty. In most cases, delimiters are added automatically. However, you can add delimiters by using the button and selecting **Add delimiter** from the drop-down list. Alternatively, you can add a delimiter from the context menu.
- **Text**: represents a custom text. You can add custom text by clicking and selecting **Add text** from the drop-down list. Alternatively, you can add a custom text from the context menu.
- **Evaluated text**: represents an item that must be added to the final expression as is. Evaluated text may contain functions or other special symbols that will modify the result. You can add an evaluated text by using the button and selecting **Add expression** from the drop-down list. Alternatively, you can add an evaluated text from the context menu.

At any moment, you can change the content of any item in the template. You can also delete items by selecting them and using the button, or by choosing **Delete column** in the context menu. To move an item, select a column in the template and then drag and drop it at the desired position. Any modifications of the template will be reflected in the expression field. Similarly, any changes in the expression field will be reflected in the template area.

To add a template column to the BOM table, click **OK**.

21.4.10.9 Footer row

A BOM table can include a footer row. To add a footer row, specify the footer type for one or more columns. By default, the title of the footer row is *TOTAL:*. Change the footer title with the **Footer title** property in the **Bill of materials property** section of the **Properties grid**.

For more information about a footer row in a BOM, see section **Footer** in Bill of Materials.

See also section **Add a footer for the column**.

21.4.10.1 Aggregated columns

0

In top-level and bottom-level BOM tables, almost any column can be assigned with an aggregate function. Aggregate functions group values of several rows in a BOM (that differ only in those values) into a single

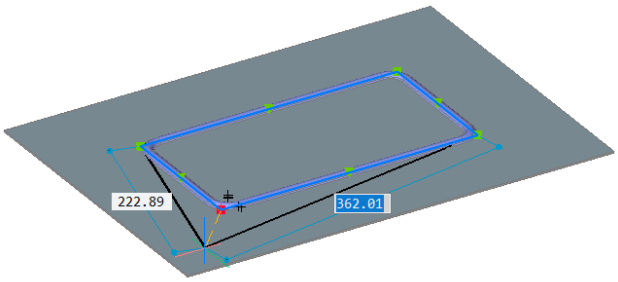
value. For example, aggregate functions can be used to compute the total length of pipes with respect to their size and thickness, or to concatenate all sizes of bolts and nuts used in the assembly in a single cell.

For more details, see section **Columns** in Bill of Materials.

To set an aggregate function, do one of the following:

- Right-click on the column title and select the required aggregate function from the **Aggregate function** submenu.
- Select the column and change the aggregate function in the corresponding column property.

Aggregated columns are marked with the ∑ symbol in their title.



Rysunek 63: Example of aggregated columns

21.4.10.1 Templates

1

You can save BOM configurations as templates. Use the BOM manager to create a new table from a template and to replace the current table configuration with one from a template. You can save the configuration as a template at any moment. You can change a loaded template configuration with the regular tools and save it as another template.

For more information about BOM templates, see section Templates in Bill of Materials.

21.4.10.1 Searching

2



Rysunek 64: Search

The BOM manager has a search field that allows you to search for a particular value in the table. You can also use wild cards in the search field. By default, BricsCAD searches the entire table. To limit the search

area, select the required columns. If the value is found, the cursor moves to that cell. If no columns are selected, the row is selected. The **Grid** automatically scrolls to make the cell visible. To find another value, use the  or  buttons. You can also use **F3** or **Ctrl+F3**. To select all found results, use the  button.
By default, the text in the search field is searched for. However, you can also use **wild cards**.
The following special symbols are supported:

Tabela 19: Wild cards

Name	Character	Description
Back quote	`	Escapes the next character (takes it literally).
Comma	,	Separates patterns.
Asterisk	*	Matches any character sequence, including a null sequence, anywhere.
Question mark	?	Matches a single character.
Period	.	Matches a single non-alphanumeric character.
At sign	@	Matches a single alphabetic character.
Pound sign	#	Matches a single numeric character.
Tilde	~	If the first character in a pattern matches anything but the pattern.
Brackets	[...]	Matches a single enclosed character.
Tilde brackets	[~...]	Matches a single non-enclosed character.
Hyphen	-	In brackets, matches a single character in the specified range.

To open the search field, use the **Search** tool on the toolbar or use **Ctrl+F**.

21.4.10.1 Associated table 3

Each BOM table in the manager can have an associated table in the document, which updates when the BOM configuration is changed. A BOM table can have one associated table only. However, you can create BOM tables that do not have an associated table in the document.

To create an associated table, use **Place/Update table**. If the BOM table was created with *BMBOM*, the associated table is created automatically.

You can locate the associated table in the document with **Associated table - Zoom to** or **Associated table - Select**. If necessary, the active layout switches to the associated table layout.

You can delete the associated table with **Associated table - Delete**. As an alternative, you can detach the associated table from the BOM table with **Associated table - Detach**. After it is detached, the associated table loses its link to the BOM table and will not update with the BOM.

21.4.10.1 Updating the BOM

4

From BricsCAD V20.2, the BOM manager automatically updates the table content in the panel, when the assembly is modified or updated. For large assemblies, this may be inconvenient. The BOM manager automatically updates the table when active.

To avoid the automatic update, close the BOM manager to make the BOM Manager panel inactive.

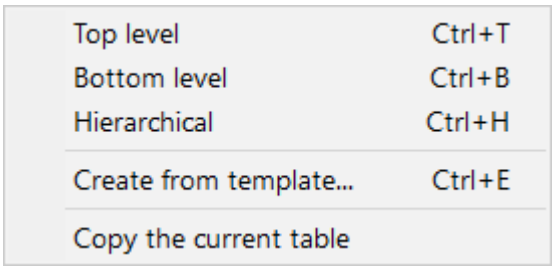
Use **Refresh** to manually refresh the BOM manager content.

Uwaga: The associated table is not updated automatically when the assembly is changed or modified. Use **Place/Update table** to update the associated table.

21.4.10.1 Adding a new bill of materials

5

1 Use the **New BOM** submenu of the **Main menu**.



Rysunek 65: New BOM submenu

2 Select the BOM type or select **From template** to choose the template file for a BOM.

21.4.10.1 Selecting the current BOM

6

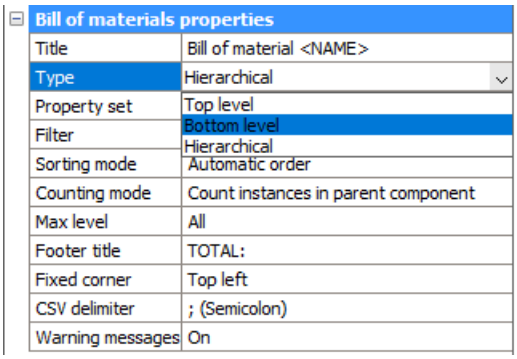
Use the **BOM selector** to select an existing bill of materials.

Uwaga: The table will automatically update to the current state of the assembly and the associated table in the drawing (if it exists).

21.4.10.1 Changing the type of the current BOM

7

Change the **Type** property in the **Bill of materials properties** section of the **Properties grid**.

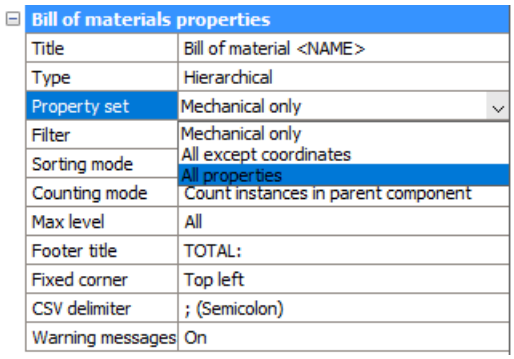


Rysunek 66: Bill of materials properties

21.4.10.1 Changing the property set of the current BOM

8

Change the **Property set** property in the **Bill of materials properties** section of the **Properties grid**.

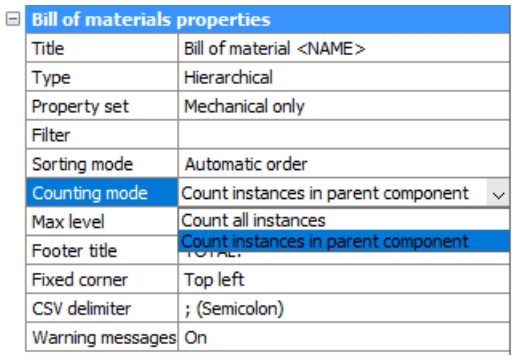


Rysunek 67: All properties

21.4.10.1 Changing the counting mode of a hierarchical BOM

9

Change the **Counting mode** property in the **Bill of materials properties** section of the **Properties grid**.

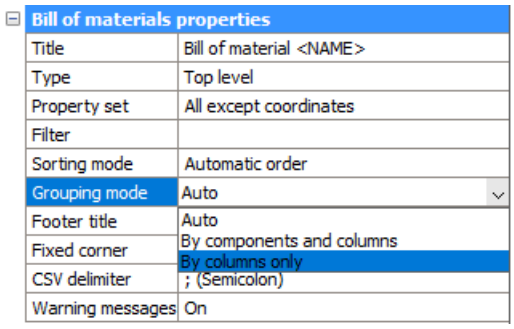


Rysunek 68: Bill of materials properties

21.4.10.2 Changing the grouping mode of top- or bottom-level BOM

0

Change the **Grouping mode** property in the **Bill of materials properties** section of the **Properties grid**.



Rysunek 69: Bill of materials properties

21.4.10.2 Setting a custom filter for the current BOM

1

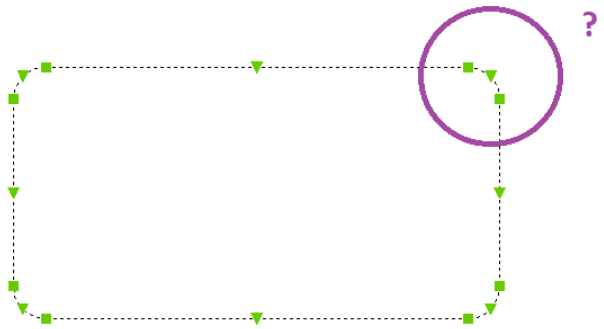
1 Select the **Set filter** option from the **Main menu**.

A *Filter Configurator* dialog box is displayed.

2 Configure the filter using the dialog tool.

See section **Filter** for details.

3 Click **OK** to apply the filter.

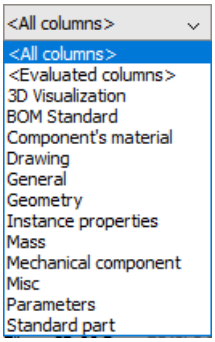


Rysunek 70: Filter Configurator

21.4.10.2 Adding new columns to the current BOM

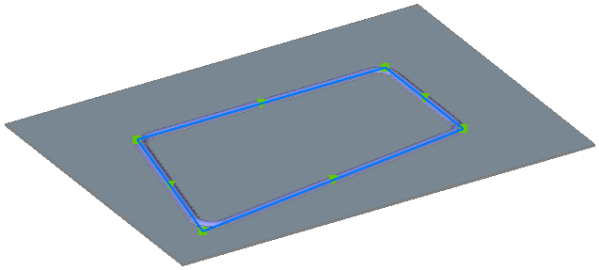
2

1 If applicable, use the **Category** selector to select the category of the properties you want to add.



Rysunek 71: All columns

2 Select the property to add from the list of the available columns. This list contains all columns from the selected category which are not yet added to the BOM. If you start to type the name of the field, BricsCAD automatically suggests names to you. Click to select it from the list.



Rysunek 72: Component description

3 If applicable, select the column in the BOM you want to place the new column next to.

4 Click **Insert the column after** ( ) to place the new column after the selected one, or **Insert the column before** ( ) to place the new column before the selected one.

If no column is selected, the new column is placed as the last or the first position respectively.

21.4.10.2 Adding an evaluated column

3

1 Select the **Evaluated columns** category using the **Category selector**.

2 Select **Formula column** to create a formula column, or select **Template column** to create a template column.

3 Click **Insert the column after** ( ) to place the new column after the selected one, or **Insert the column before** ( ) to place the new column before the selected one.

If no column is selected, the new column is placed as the last or the first position respectively.

A dialog box to configure the evaluated column is displayed.

4 Change the **Title** of the column.

5 If applicable, change the **Unit**.

6 Configure the expression using the dialog tools.

7 Click **OK** to add the column to the BOM table.

21.4.10.2 Adding a template column from selected columns

4

1 Select the columns to be combined in a template column. Use the Ctrl key to select multiple columns or the Shift key to select a range of columns.

2 Right-click on the title of one of the selected columns.

3 Select **Add template column from selected** from the context menu.

A dialog box to configure the evaluated column is displayed.

4 Change the **Title** of the column.

5 If applicable, change the **Unit**.

6 Configure the expression using the dialog tools.

7 Click **OK** to add the column to the BOM table.

8 If applicable, remove or hide columns that are no longer necessary.

21.4.10.2 Changing an evaluated column

5

1 Right-click on the column title in the **BOM grid**.

2 Select **Edit column expression** from the context menu.

A dialog box to configure the evaluated column is displayed.

3 If applicable, modify the column title, unit and expression.

4 Click **OK** to apply the changes.

21.4.10.2 Removing a column from the current BOM

6

1 Select the column in the grid.

2 Click the **Remove the column** tool (), or press **Delete**.

21.4.10.2 Changing the BOM title

7

Use **BOM Title** to edit the title, or use the **Title** property in the **Bill of materials properties** section of the **Properties grid**.

You can use the *<NAME>* placeholder to refer to the assembly name.

21.4.10.2 Changing the footer row title

8

Use the **Footer title** property in the **Bill of materials properties** section of the **Properties grid**.

21.4.10.2 Adding a prefix or suffix to numbers

9

1 Select the **Number** column.

2 Select **Number prefix** property in the **Column properties** section.

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

Rysunek 73: Column properties

3 Enter the new prefix string if necessary.

4 Select **Number suffix** property in the **Column properties** section.

5 Enter the new suffix string if necessary.

21.4.10.3 Changing the delimiter between numbers in a hierarchical BOM 0

1 Select the **Number** column.

2 Select **Delimiter** property in the **Column properties** section.

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

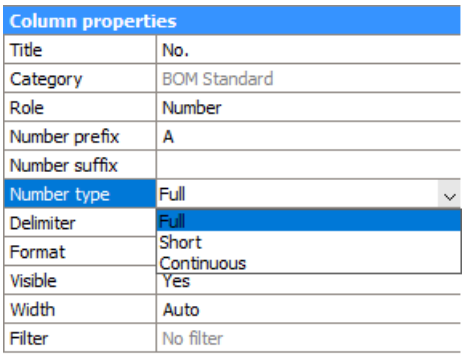
Rysunek 74: Column properties

3 Enter the new delimiter string.

21.4.10.3 Changing the number type in hierarchical BOM 1

1 Select the **Number** column.

2 Select **Number type** property in the **Column properties** section.



Rysunek 75: Column properties

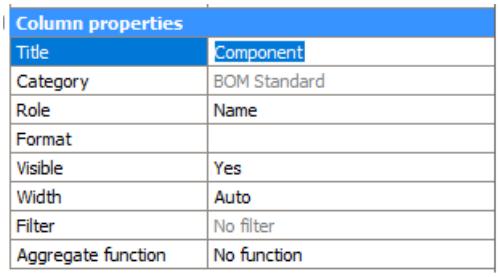
3 Select the required number type.

21.4.10.3 Change the column title

2

1 Select the column in the **BOM grid**.

2 Select **Title** property in the **Column properties** section.



Rysunek 76: Column properties

3 Change the **Title** property value to the new title of the column.

21.4.10.3 Change the column format

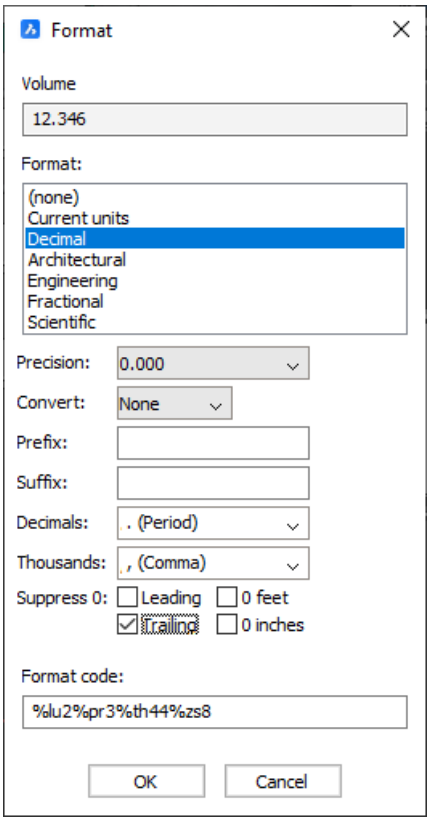
3

1 Select the column(s).

2 Right-click on the title of a selected column.

3 Select **Format** in the context menu.

The *Format* dialog box displays.



Rysunek 77: Format

4 Define the desired format.

5 Click **OK** to apply the changes.

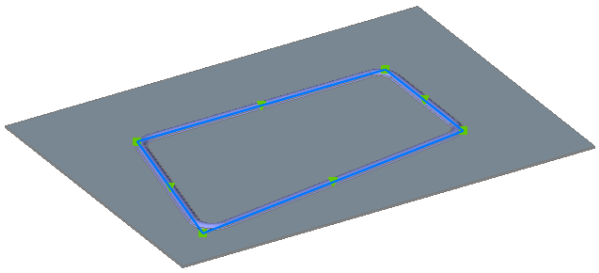
21.4.10.3 Changing the units and unit placement modes for a column

4

1 Select the column(s).

2 If applicable, change the **Unit placement** property in the **Column properties** section.

3 If applicable, change the **Units mode** property in the **Column properties** section.



Rysunek 78: Column properties

4 If applicable, change the **Title format**.

- Use *<Title>* to add the column title.
- Use *<Unit>* to add the column unit.

- Use \n to add a line break.

21.4.10.3 Adding a footer to a column

5

1 Select the column(s).

2 Use the **Footer** submenu of the **Main menu** to select the type of footer value, or right-click on the column label and then select **Footer** submenu.

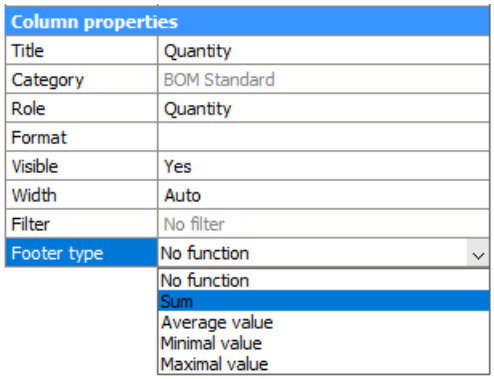
The following types are available:

- **No footer** (the column will be removed from the footer. If there is no column with another footer type, the footer row will be removed completely)
- **Sum**
- **Average value**
- **Minimal value**
- **Maximal value**

As an alternative:

1 Select the column(s).

2 In the **Properties grid**, select the **Footer type** property and choose the footer type.



Rysunek 79: Column properties

Uwaga: To toggle between **Sum** and **No footer**, click the **Footer** button ( ).

21.4.10.3 Saving the current configuration as a template

6

1 Choose **Save as template** ( ) in the main menu ( ).

The dialog box to select the name and location for a template file displays.



Rysunek 80: Select file to save BOM table template

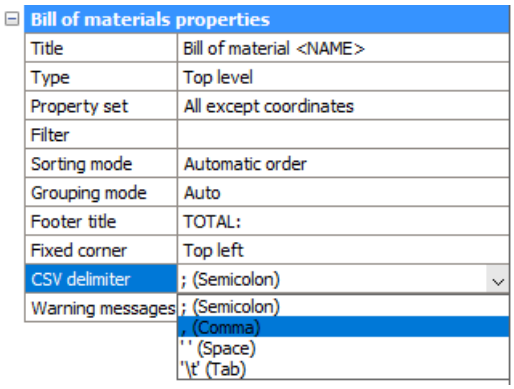
2 Specify the name and location of the template file.

3 Click **Save**.

21.4.10.3 Exporting the current table to CSV

7

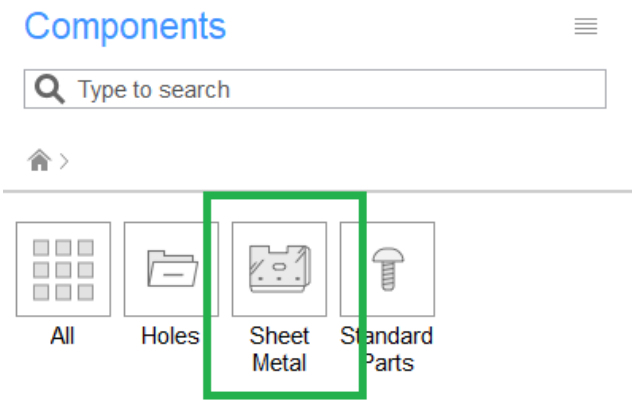
1 If applicable, adjust the **CSV delimiter** property of the table.



Rysunek 81: CSV delimiter

2 Choose **Export to CSV** in the main menu ( ).

The dialog box to select the name and location for a CSV file displays.



Rysunek 82: Select file to export BOM table

3 Specify the name and location of the CSV file.

4 Click **Save**.

21.4.11 Bill of Materials

21.4.11.1 Overview

You can use the *BMBOM*, *BMBOMPANELOPEN* and *BMBOMPANELCLOSE* commands.

An engineering Bill of Materials (BOM) is a list of unique mechanical components used in an assembly. In BricsCAD, the list is represented as a table entity and can be placed anywhere in the drawing. Each entity has its own serial number (starting with 1), name, and number of occurrences.

You can view and configure Bills of Materials in the Bills of Materials Manager.

21.4.11.2 Bill of Materials types

There are 3 BOM types in BricsCAD:

- **Top level**

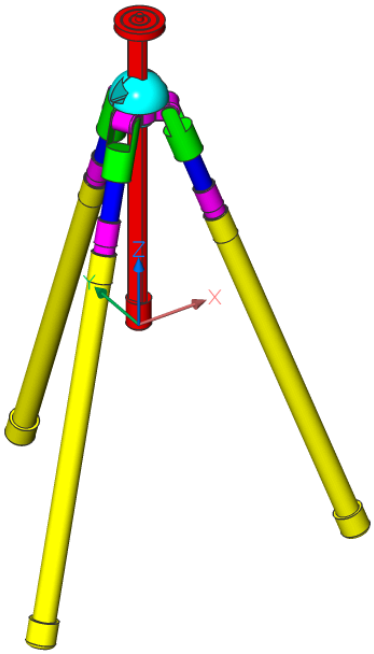
  Contains top-level components only. Top-level components are components inserted directly into the main assembly. This type of BOM lists the main components only, without details of subcomponents.
- **Bottom level**

  Contains bottom-level components (parts) only. Bottom-level components are parts that do not contain other parts or subassemblies. This type of BOM lists all parts in the main assembly, regardless of which component the parts belong to.
- **Hierarchical**

  Contains all components with their subcomponents listed directly below. The serial numbers of subcomponents is a sequence of numbers separated by dots (for example, "1.2.3"). This BOM type shows the hierarchical structure of the assembly.

Uwaga: The **BOM status** property controls which parts are included in a Bill of Material. The level of a component in a BOM table is set with respect to the **BOM status** of all related components. For example,

a component with a Terminal BOM status is always considered as bottom-level, even if it contains other parts or subassemblies.



Rysunek 83: Example: Tripod assembly

Tabela 20: BOM types

Type	Bill of Material
Top level	Bill of Materials tripod_main_assy No. | Component | Quantity 1 | tripod_axis_assy | 1 2 | tripod_foot_assy | 3 **Rysunek 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 **Rysunek 85:** Bottom level

Tabela 20: BOM types

Type	Bill of Material
Hierarchical	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

Rysunek 86: Hierarchical

21.4.11.3 Counting mode and maximal level

For hierarchical BOM tables, 2 counting modes are supported:

- **Count all instances**

  The total number of subassemblies and parts in the entire assembly.
- **Count instances in parent component**

  The total number of subassemblies and parts in the parent component. The parent component is considered as the root component.

For example, the main Tripod assembly contains 3 Tripod foot assemblies. Each instance of a Tripod foot assembly contains 1 Bond assembly and 1 Tube assembly.

If the **counting mode** is set to **Count all instances**, the result is:

Tabela 21: BOM Tripod Assembly (count all instances)

#	Component	Quantity
1	Tripod assembly	1
1.1	Tripod foot assembly	3
1.1.1	Bond assembly	3
1.1.2	Tubes assembly	3

If the **counting mode** is set to **Count instances in parent component**, the result is:

Tabela 22: BOM Tripod Assembly (count instances per component)

#	Component	Quantity
1	Tripod assembly	1
1.1	Tripod foot assembly	3
1.1.1	Bond assembly	1
1.1.2	Tubes assembly	1

The default value for the **counting mode** is **Count instances in parent component**. You can change the counting mode in the BOM Manager, or with the *BMBOM* command.

Uwaga: Before BricsCAD V20.2, all hierarchical BOM tables used the **Count all instances** mode by default. When such tables are updated in BricsCAD V20.2 or later, the default mode is set to **Count instances in parent component**. This may result in different quantities for some parts. If necessary, manually set the counting mode in the BOM Manager.

By default, hierarchical bills of materials show all levels in the model. It is possible to limit a hierarchical BOM to a certain level using **Max level** property. When the maximal level is set, only parts up to that level will be included. For example, if **Max level** = 2, only top level components and subcomponents directly included in top level components will be listed.

21.4.11.4 BOM columns

By default, the BOM table contains the Number, Name and Quantity columns.

Using the Configure option of the *BMBOM* command you can add more columns, such as Description, Density, Volume, Mass, Material, Thickness (for sheet metal parts), and Parameters (for parametric components).

You can add any built-in property of components, instances or block references to the BOM table. Custom evaluated columns are also supported, using the DataExtraction technology.

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

Rysunek 87: BOM with density, volume, and mass

Grouping modes

BOM tables list unique components only; multiple inserts of the same components are grouped, according to the Grouping mode.

Starting from BricsCAD V21, the following grouping modes are supported:

- **Auto** (default mode)

  Multiple inserts of the same components are grouped together with respect to their parameters and added properties. It means that multiple inserts of the same parametric component are always grouped in separate BOM rows with respect to their parameters, regardless if those parameters are added to BOM as columns.

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

Rysunek 88: BOM Bookshelf

- **By components and columns**

  In this mode, multiple inserts of the same components are grouped together with respect to added properties only. It means that multiple inserts of the same parametric component could be grouped in the same row even if their parameters are different, depending on what columns are added to BOM. However, instances of different components are always grouped in different rows. This mode is useful, if some parameters correspond to different states of components but not result in being different parts.

- **By columns only**

  In this mode, inserts of components are grouped with respect to their properties added to the BOM, regardless if they are inserts of the same component. This mode is useful to get a summary table, such as a list of all physical materials or a list of all sizes of pipes.

Visible and invisible columns

You can make any column in the BOM table invisible. Invisible columns do not appear in the table in the document. However, you can still use these columns in formulas. Invisible columns are also displayed in the BOM Manager.

Aggregated columns

Starting from BricsCAD V21, it is possible to set an aggregate function to almost every column in top- or bottom-level BOM tables. Aggregate functions group values of several rows in BOM (that differ only in those values) into one single value. Following aggregate functions are supported:

- **Sum**: Sum the grouped values.
- **Average**: Find the average value of grouped values.
- **Minimum**: Find the minimal value of grouped values.
- **Maximum**: Find the maximal value of grouped values.

- **Concatenate**: Concatenate all unique values in the group in ascending order.
- **Concatenate with counts**: Concatenate all unique values with their number of counts in the group.

For example, take the following BOM table:

Tabela 23: BOM table

Component	Width	Length	Height	Quantity
Beam	16	250	10	1
Beam	16	200	10	2
Beam	16	220	20	1
Beam	20	215	20	3
Beam	20	225	20	1

Apply an aggregate function to the **Length** column. The results are combined in the following table:

Tabela 24: BOM table

Component	Width	Height	Length (sum)	Length (average)	Length (minimal)	Length (maximal)	Length (concatenate)	Length (concatenate with counts)
Beam	16	10	650	216.6667	200	250	200; 250	2x200; 250
Beam	16	20	220	220	220	220	220	220
Beam	20	20	870	217.5	215	225	215; 225	3x215; 225

For **Concatenated with counts** columns, you can:

- Adjust the position of the number of counts (in front or behind the value).
- Add the delimiter between values.
- Add the delimiter between values and their number of counts.
- Define the postfix and/or prefix for the number of counts.

If an aggregated column is used in an evaluated column, the final values are used by default. However, it is possible to apply an aggregate function to an evaluated column. In this case, the results for such

an evaluated column is first computed for all original values (before aggregation), and then the resulted values are aggregated.

Number column and column roles

A number column is a special column to enumerate rows. It assigns numbers automatically for all rows in the BOM table, without skips and independently of other tables. Rows are numbered starting from 1.

Following settings can be changed for the number column:

- **Prefix**: A string that is added in front of any number. For example, you want all numbers to start with "A".
- **Suffix**: A string that is added after the number. For example, you want to add "-1" to all numbers.

For a hierarchical BOM, the following settings are available:

- **Number type**: Specifies the number format:
  - **Full**: Numbers are formatted as "1.2.3".
  - **Short**: Only numbers for the current level show. It is used together with the Level column, which shows the level of the current part in the assembly.
  - **Continuous**: All components are numbered continuously starting from 1, regardless of their level.
- **Delimiter**: Specifies the symbol to separate numbers for different levels when the number type is set to *Full*.

In general, the number column is used as the source to annotate the drawing by the *BMBALLOON* command. However, any column in the table can be used as the source for the annotation balloons, by changing its role. The following column roles are available:

- **Number**: The column contains numbers for annotations.
- **Name**: The column contains part names for annotations.
- **Quantity**: The column contains quantities of parts for annotations.
- **Regular**: The column has no special role.

Only one column can have a **Number**, **Name** or **Quantity** role. If no column has one of these roles assigned, automatic annotations are not possible.

Column width



Rysunek 89: Column properties

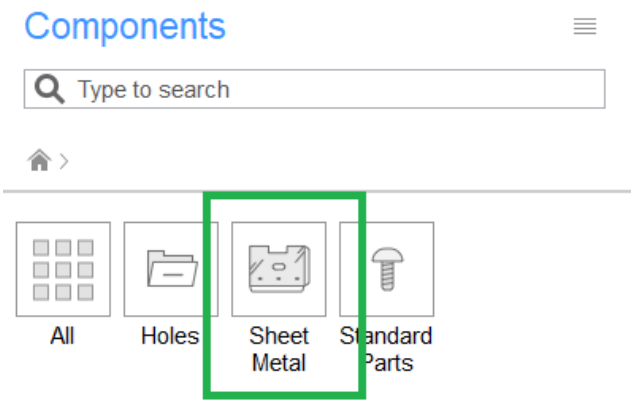
By default, the width of each column is set automatically, so its content fits without additional line breaks. However, it is possible to set a fixed column width.

To set the column width, perform one of the following:

- Select the required column in the BOM manager, then type a value for the **Width** property in layout units.
- Change the width of the required column in the associated table. Starting from BricsCAD V20.2, BOM tables automatically keep the width of columns that have been changed manually.

To restore the automatic width mode, select the column in the BOM manager and select **Auto** in the drop-down list for the **Width** property.

Column format



Rysunek 90: Format

You can format column values:

1 Select the required column in the BOM manager.

2 Choose the **Format** option in the context menu.

3 Define the desired format in the **Format** dialog box.

Uwaga: The available settings depend on the data type of the selected column. You can select multiple columns of the same type to format them equally.

Property sets

Because BOM tables use DataExtraction technology, you can list properties of mechanical components and entity properties, for example, block references in BOM tables.

Uwaga: These properties are not always necessary. In large assemblies, these properties could negatively affect performance. This is because BricsCAD caches properties during BOM table creation.

Property sets are built-in sets of properties. You can choose the most optimal set of properties for your needs. The following property sets are supported:

- **Mechanical only**(default property set): Only properties of mechanical components are listed.

- **All except coordinates**: Lists all properties of components, except coordinate ones. Coordinate properties are typically used only for component instances.
- **All properties**: Lists all possible properties of components and component instances.

You may change the property set for a BOM table in the BOM Manager or use *BMBOM* when you add a new table.

21.4.11.5 Filter a BOM table

You can filter parts that will form the Bill of Material. Starting with BricsCAD V21, there are 2 filter types: **main** and **column**.

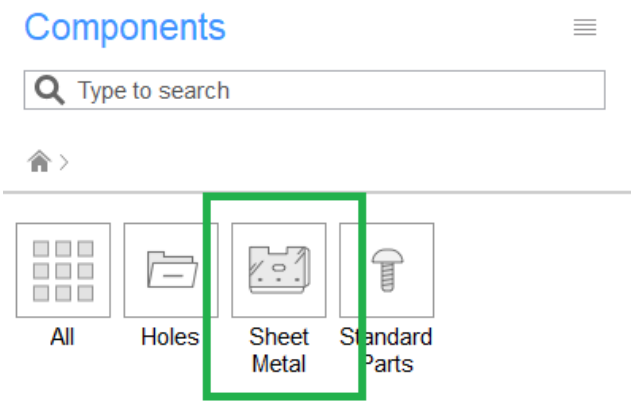
Main filter

A Main filter works in the same way as the *DATAEXTRACTION* command. First, all parts in the assembly, at the required level (top, bottom or hierarchical), are matched against the filter expression. Then, only the parts that have passed the filter are included in the BOM.

The filter expression may not refer to BOM columns because filtering is done before the BOM is generated. For example, it is possible to create a filter for all standard parts because **Is standard** is a normal property of the component. However, it is not possible to set the main filter to filter out all parts with a quantity greater than 1. You must use Column filters for that.

Filter configurator

The easiest way to configure the main filter is to use the **Filter Configurator** in the BOM manager.



Rysunek 91: Filter Configurator

The Filter Configurator allows to define a set of conditions. Each condition defines a relation between a selected property and a target value.

Possible relations are:

- Equal to
- Not equal to
- Less than
- Greater than

- Less than or equal to
- Greater than or equal to

Conditions can be added to a group, which defines how those conditions are treated. There are 2 types of groups:

- **All of** group: is "true" if all conditions in that group are "true". This is the same as the "and" operator.
- **Any of** group: is "true" if at least one condition in that group results in "true". This is the same as the "or" operator.

A group can contain an arbitrary number of conditions. It can also contain nested groups, thus making complex filters possible.

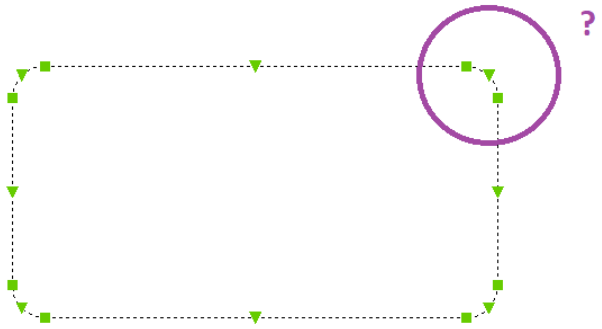
To add a condition:

- Select a property from the list of properties.
- Select a required relation operator and specify the target value.

  For text values, a target value can be selected from the list of all possible values also. For numerical values, their target unit can be specified also in order to automatically convert the value with respect to the default unit of the document.

For text values, "Equal to" and "Not equal to" relations support wild cards.

The Filter Configurator automatically converts the set of conditions into a filter expression, which can be made visible by clicking :



Rysunek 92: Expression

This expression is fully editable and two-way associative with the set of conditions.

Syntax description:

Tabela 25: Relational operators

Operator	Meaning
==	equal to
!=	not equal to

Tabela 25: Relational operators

Operator	Meaning
<	less than
>	greater than
<=	less than or equal to
>=	greater than or equal to

- Property names must be enclosed in quotes.
- Relational operators can be combined using "and" and "or" operators. Operator "and" results in "true" if both sides of the operator are true. Operator "or" results in "false" if either left or right side of the operator is true.
- Logical operators "and" and "or" are case sensitive.
- Use "(" and ")" to combine two operators into one.
- String values are enclosed in double quotation marks "".
- Relational operators for strings are case sensitive.
- "==" and "!=" operators support wild cards.

For example, to get a BOM of all standard parts, use the following expression:

"Is standard"=="Yes"

To get a BOM of all ISO standard parts, use the following expression:

"Is standard"=="Yes" and "Part standard"=="ISO"

To get a BOM of all ISO or DIN standard parts, use the following expression:

"Is standard"=="Yes" and ("Part standard"=="ISO" or "Part standard"=="DIN")

Wild cards

Wild cards allow you to match property values against a specific template. For example, if you want to find all components containing "nozzle" in their names, the following expression could be used: "Component name"=="*nozzle*".

Wild cards can also be used as target values in the Filter Configurator.

Only "Equal to" and "Not equal to" relational operators support wild cards.

Uwaga: Wild cards in the main filter are case-sensitive. For Column filters, you can specify if you need case-sensitive or case-insensitive matching.

The following special symbols are supported:

Tabela 26: Special symbols

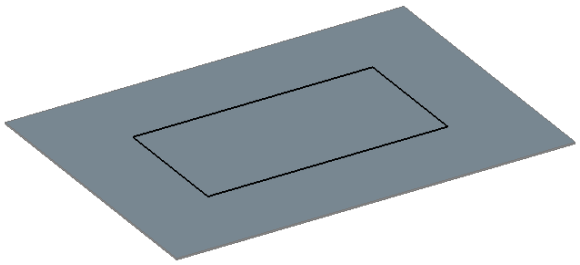
Name	Character	Description
Back quote	`	Escape the next character (takes it literally).
Comma	,	Separate patterns.
Asterisk	*	Match any character sequence, including a null sequence, anywhere.
Question mark	?	Match a single character.
Period	.	Match a single non-alphanumeric character.
At sign	@	Match a single alphabetic character.
Hash tag	#	Match a single numeric character.
Tilde	~	If the first character in a pattern matches anything but the pattern.
Brackets	[...]	Match a single enclosed character.
Tilde brackets	[~...]	Match a single non-enclosed character.
Hyphen	-	In brackets, match a single character in the specified range.

Column filter

Column filters have been introduced in BricsCAD V21. This allows you to quickly set the conditions which rows display, depending on the column content (somewhat similar to filters in some spreadsheet processors). Unlike the main filter, they are applied after the BOM table has been generated. Therefore, it is possible to use column filters for columns with aggregated content, such as quantities.

Each column can have one filter only. However, you can add column filters to as many columns as you need. Only rows that pass all column filters display in the final BOM.

You can configure column filters using the **Column Filter** dialog box in the BOM manager:



Rysunek 93: Quick Filter

In this dialog box, you can select the type of filter from the list and configure it accordingly. The list of possible filters depends on the column content type, as shown in the table:

Tabela 27: Filters

Filter type	Column type	Description
always accepted	any	all column values are accepted
equal to	numerical	value should be equal to the target one
not equal to	numerical	value should be not equal to the target one
less than	numerical	value should be less than the target one
greater than	numerical	value should be greater than the target one
less than or equal to	numerical	value should be less than or equal to the target one
greater than or equal to	numerical	value should be greater than or equal to the target one
between	numerical	value should be in the given range, both ends included
not in range	numerical	value should not be in the given range, both ends excluded
exactly matches	text	value should exactly match the target one
does not match	text	value should be different from the target one
starts with	text	value should start with the given text

Tabela 27: Filters

Filter type	Column type	Description
does not start with	text	value should not start with the given text
ends with	text	value should end with the given text
does not end with	text	value should not end with the given text
contains	text	value should contain the given text
does not contain	text	value should not contain the given text
matches pattern	text	value should match the given wild card template
does not match pattern	text	value should not match the given wild card template
in	text and numerical	value should be in the given set of values
not in	text and numerical	value should not be in the given set of values

For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

21.4.11.6 Sort a BOM table

There are 3 sorting modes for BOM tables:

- **Automatic order**

  The table content is sorted as follows: component name, physical material name, thickness (if applicable), parameters (in alphabetic order), and quantity.
- **Natural order**

  The table content is not sorted. Instead, all components are in the order of their appearance in the document. In most cases, parts added earlier will precede parts added later.
- **Custom order**

  The table content is sorted according to the columns you select. If no columns are specified, the natural order is used.

Starting from BricsCAD V21, BOM tables use so-called "natural sorting" of text. It means that if text contains a mix of letters and numbers, it will be sorted with respect to those numbers. For example, strings "N1", "M2", "N11", "N2", "AA1" will be sorted as "M2", "N1", "N2", "N11", "AA1." Therefore, it is possible to

assign custom numbers with various prefixes to components using custom parameters, and then sort the BOM tables by those numbers in the correct order.

21.4.11.7 Evaluated columns

Evaluated columns are special columns, of which results are depending on other columns using user-defined expressions. Starting from BricsCAD V21, 2 types of evaluated columns are supported: **formula** and **template** columns.

Formula columns

A Bill of Materials supports formula fields. The same syntax is used in the *DATAEXTRACTION* command. **FormulaExpression** is the expression in the format of **\AcExpr** formula fields. This can contain constants such as PI or functions such as SIN. An operand in an expression can be a **ColumnTitle** enclosed in arrow brackets (<>).

For example, formula *=<Mass>/<Volume>* calculates the average density of the component.

If a row does not contain a value, a default value will be used instead. It is possible to specify the default value for an operand using the following syntax: *<"ColumnTitle",DefaultValue>* where **ColumnTitle** is the title of the required column, and **DefaultValue** is the default value. **ColumnTitle** must be between double quotes ("). If no default value is specified, an empty value is used, which in most cases results in an empty output.

A formula column can contain multiple expressions. Each expression must be enclosed by sum brackets ({}). These expressions are evaluated from first to last, and the first one yielding a non-empty output is used.

Starting from BricsCAD V21, a formula can refer to other formula columns, provided that they do not form a loop. In a hierarchical BOM, a formula can also refer to values of the parent row by adding *Parent:* before the column name. It also can refer to properties that are not explicitly added to the table, but all other formula columns referred to in the formula must exist in the table. You can still use such columns displayed as invisible in the BOM table in formula expressions.

For distance, area, volume and mass properties, the values will be converted to the **user-defined** unit before using them in the formula expression. If the **user-defined** unit is not set, the document unit is used. For the mass property, kilogram is used in metric documents and lbs (pound) in imperial documents.

Template columns

Template columns allow you to combine several properties, as well as a custom text, into a single column. A special syntax, based on Diesel expressions, is used to define template columns. Expressions for template columns must start with the **$** sign. An operand in an expression can be a **ColumnTitle** enclosed in arrow brackets (<>). It is also possible to use special functions:

Tabela 28: Special functions

Function	Description
$(+, arg1, arg2[, ... , arg9])	Add the provided arguments to the first one.

Tabela 28: Special functions

Function	Description
$(-, arg1, arg2[, ... , arg9])	Subtract the provided arguments from the first one.
$(*, arg1 , arg2[, ... , arg9])	Multiply the first argument by the provided arguments.
$(/, arg1, arg2[, ..., arg9])	Divides the first argument by the provided arguments.
$(=, arg1, arg2)	Return 1 if arg1 is equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(<, arg1, arg2)	Return 1 if arg1 is less than arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(>, arg1, arg2)	Return 1 if arg1 is greater than arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(!=, arg1, arg2)	Return 1 if arg1 is not equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(<=, arg1, arg2)	Return 1 if arg1 is less than or equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(>=, arg1, arg2)	Return 1 if arg1 is greater than or equal to arg2; otherwise, return 0. Both arg1 and arg2 must be numbers.
$(and, arg1, arg2[, ... , arg9])	Apply bitwise AND operation to arguments.
$(angtos, angle[, mode, precision])	Convert the angle to string with respect to given mode (0 - degrees, 1 - d/m/s, 2 - grads, 3 - radians) and precision.
$(delim, delimiter_string)	Insert a smart delimiter. The delimiter string is added to the output only if the output before and after this delimiter is not empty.
$(edtime, time, format)	Format the date according to the given format string.
$(eq, arg1, arg2)	Return 1 if arg1 is equal to arg2; otherwise, return 0. Both arg1 and arg2 must be text.
$(eval, expression)	Evaluate the given expression as a Diesel expression.
$(fix, value)	Truncate the real value to the integer one.

Tabela 28: Special functions

Function	Description
$(getvar, variable)	Return the value of the given system variable.
$(if, arg, then[, else])	Return 'then' if 'arg' is not 0; otherwise, return 'else' or empty string if 'else' is not provided.
$(index, position, comma_separated_string)	Return the substring at the given position in the given comma-separated string.
$(nth, position, arg1[, arg2, ... , arg9])	Return the argument at the given position.
$(or, arg1, arg2[, ..., arg9])	Apply bitwise OR operation to arguments.
$(rtos, value[, mode, precision])	Format the given value with respect to the required mode and precision.
$(strlen, string)	Return the length of the given string.
$(substr, string, start[, length])	Return the substring of the 'string' from 'start' of the given 'length'. Numeration starts from 1.
$(upper, string)	Convert 'string' to uppercase.
$(xor, arg1, arg2[, ... , arg9])	Apply bitwise XOR to all arguments.

One template column can contain multiple expressions. Each expression must be enclosed by sum brackets ({}). These expressions are evaluated from first to last, and the first one yielding a non-empty output is used.

Starting from BricsCAD V21, a template column can refer to other evaluated columns, provided that they do not form a loop. In a hierarchical BOM, a template column can also refer to values in the parent row by adding *Parent:* before the column name. It can also refer to properties that are not explicitly added to the table, but all other evaluated columns referred to in any of the expressions must be present in the table. You can still use columns displayed as invisible in the BOM table in template expressions.

You can use "\n" string to add a line break.

For example, the following expression combines PartNumber, Width, Height and Length properties:

${<PartNumber>$(delim," X ")<Width>$(delim," X ")<Height>$(delim," X ")<Lenght>}

Expression configurator

You can use the *Expression configurator* to configure an evaluated column. This dialog box is available in Bill of Materials Manager.



Rysunek 94: Template editor

This dialog box allows you to set the title and unit for the column, edit the expression using an editor with syntax highlight and autocompletion, and see a preview of the BOM table with the evaluated column. For a template column, the **Template** section allows you to easily insert, move and delete properties, as well as delimiters and custom texts.

21.4.11.8 Column units

You can adjust the **units placement** mode for columns that have associated units. The following options are available:

- **Don't show units**

  The BOM table does not display units for this column.
- **In column title**

  if all values in the table use the same unit or can be converted to a common unit, all values are converted to that common unit. The unit name is added to the column title. You can configure the position of the title and the unit using the **Title format** property. Use *<Title>* to denote the title, and *<Unit>* to denote the unit. For example, the default title format is "*<Title>*, **<Unit>**." You can use "\n" to add a line break.

  The common unit is picked in the following order:
  a. A unit set by you.
  b. If the property is a distance, an area, a volume or a mass, and the unit mode is **Best for all values**, the best representation for the majority of the values is used.
  c. If all values have the same unit, this will be used.
- **In cell**

  The unit name is added to the value. Value and unit display in the same cell.
- **In additional column**

  The unit name is placed in a separate column.

You can set the **Unit mode** for distance, area, volume and mass values to define the units used to display a particular value. For example, if there are masses going from 1 gram to several tons, choose **Best for each value** to display each mass converted to the most suitable mass unit. To see all masses in kilograms, set kilograms as the **user-defined** unit for the column. **Best for all values** is the default.

Units mode options:

- **Don't show units**

  Units are not displayed.
- **Best for all values**

  The unit with the best representation for the majority of the values is displayed. Only dimensions, area, volume and mass properties are supported.
- **User-defined**

  You choose the unit to display. In addition to dimensions, area, volume, and mass properties, it is also possible to select a required unit for material properties, such as density, thermal conductivity and specific heat.
- **Best for each value**

  The unit that is most suitable for each value is used. The **In column title** mode of the **Unit placement** automatically switches to the **In additional column** mode. Only dimensions, area, volume, and mass properties are supported.

Uwaga: If a user-defined unit is set for a column and this column is used in a formula field, the values converted to that unit will be substituted in the formula. Otherwise, the document unit will be used.

21.4.11.9 Footer

The Bill of Materials table supports a footer. To define which columns will form the footer, set the **Footer type** for the required columns:

- **No footer**

  The column ID will not be included in the footer row.
- **Sum**

  The sum of all values for each individual component instance displays in the footer.

  Uwaga: This is not the equivalent of the sum of all values displayed in the table, because the number of each component is used. For formula fields, the sum of values for Quantity=1 is used.
- **Average**

  The average value, of all values, for each individual component, displays in the footer. For formula fields, the sum of values for Quantity=1 is used.
- **Minimal**

  The smallest value for each individual component displays in the footer. For formula fields, the sum of values for Quantity=1 is used.
- **Maximal**

  The largest value for each individual component is displayed in the footer. For formula fields, the sum of values for Quantity=1 is used.

If all columns have the footer type **No footer**, the footer row does not display.

21.4.11.1 Templates

0

You can save a BOM configuration as a template and use it to create new BOM tables. The template file contains all necessary data. It includes table type, title, filter, and set of columns with all the associated properties. You can use <NAME> in the title as a placeholder for the actual name of the assembly. For example, if your assembly name is 'My Assembly' and the title of the BOM is specified as Bill of Materials for <NAME>, the BOM title will be 'Bill of Materials for My Assembly', because 'My Assembly' replaces <NAME>.

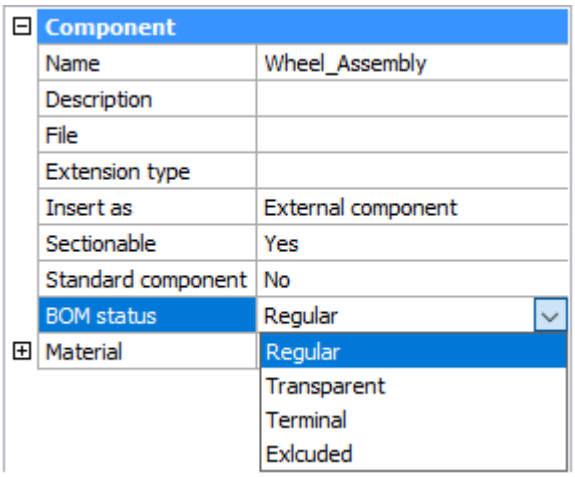
By default, BOM template files have a .BOM extension.

A BOM template file is an extension of the Data Extraction Definition file (.DXD). However, you cannot use a .DXD file as a BOM template, because some BOM-specific data will be missing. You can not use a .BOM file to define a DataExtraction template.

You can edit template files in any text editor. However, Bricsys recommends to use the *BMBOM* command or the BOM manager to create and edit template files.

21.4.11.1 BOM status

1



Rysunek 95: Component

Each component has a **BOM status** property. You can use this to control the behavior of the component in **Bill of Materials** tables. This property can have one of the following values:

- **Regular**

  The component appears in a Bill of Materials at the same position as the assembly structure and the default value of the selected mode for a Bill of Materials.

- **Transparent**

  The component acts as a transparent container for its subcomponents. The component does not appear in the Bill of Material, but subcomponents can appear in the BOM, depending on their BOM status, position in the assembly structure, and the mode of the BOM table.

- **Terminal**

The component is treated as a leaf component (has no subcomponents). This is useful for complex parts or standard components.

- **Excluded**

  The component and its subcomponents do not appear in the Bill of Materials.

To set the BOM Status of an external or root component

1 Open the file that contains the component.

2 Select the root component in the Mechanical Browser.

3 Set the **BOM status** property.

To set the BOM Status of a local component

1 Select the component in the Mechanical Browser.

2 Right-click on the selected node to open the context menu.

3 Select a BOM status from the **BOM status** submenu.



Rysunek 96: BOM status submenu

Uwaga: It is not possible to set the BOM status via the context menu for local components that are inserted into an external component. To do this, you must open the component that contains the local component.

21.4.11.1 Fixed corners

2

Starting from BricsCAD V21, you can set any corner of the table as fixed. The fixed corner will always keep its initial position during BOM update. When the table has been moved manually, the new position of this corner will be kept. Fixed corners allow to align BOM tables with other objects.

You may choose which corner to fix during the table placement by either the *BMBOM* command or the BOM manager; use the Tab key to cycle the corners. You can change the fixed corner of a BOM table in the BOM manager.

21.4.12 Multi-parametric 3D animations

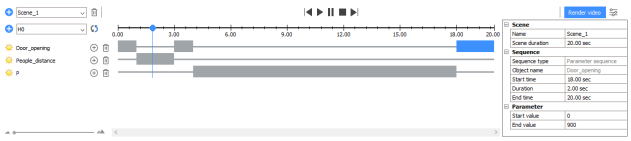
21.4.12.1 Overview

Use commands: *ANIMATIONEDITOROPEN* and *ANIMATIONEDITORCLOSE*.

You can animate several parameters and program camera movements with the Animation Editor panel. Once the animation is programmed, you can play it in BricsCAD or render it as a 2D video.

Each parameter animation has its own "timeline". You can add as many animation sequences to a parameter as you like, as long as these sequences do not overlap.

There is also a special timeline for camera movements. You can add as many camera sequences as you like. You can set both the point of focus and the position of the camera with either a polyline or a point. This is similar to the *ANIPATH* command.



Rysunek 97: Timeline

21.4.12.2 Scenes, Timelines, and Animation Sequences

Scenes

A scene consists of one or more **timelines** and each of these timelines has one or more **animation sequences**. When the editor is launched for the first time it automatically creates a scene. A scene sets the duration of the animation. Together, they define the **animation schedule** used for playback and render. Currently, you can only render the current scene with the Animation Editor. However, you can have many scenes in the same document and you can switch between them.

Animation sequences

An animation sequence is the basis of an animation. It represents the single, continuous animation of an object. An animation sequence has an associated start time and duration. It also has values specific to the animated object, such as start and end values for parameters, or camera and viewpoint paths for camera sequences:

- **Parameter**

  An animation sequence represents the continuous change of a parameter from a start to an end value. For example, a distance constraint between two faces: the animation will move one face to a maximum and minimum distance value.

- **Camera**

  An animation sequence corresponds to a continuous camera movement on a path. You can adjust various camera movements with the camera timeline.

Timelines

A timeline is a visual representation of an animation sequence for an object. The timeline displays how and when the parameters will animate. Each parameter has its own timeline in a scene. Animation

sequences cannot overlap. A timeline can be temporarily disabled to exclude it from an animation. All timelines are independent. This means that you can animate many parameters at the same time and see their combined effects.

21.4.12.3 To open the Animation Editor

Perform one of the following:

- Place the cursor on a toolbar, right-click, and choose **Animation Editor** from the context menu.
- Click the **Animation Editor** tool ( ) on the **View** section of the ribbon.
- Choose **Animation Editor** from the **View** menu.
- Enter *ANIMATIONEDITOROPEN* into the Command line.

21.4.12.4 The Animation Editor Interface

The Animation Editor is a panel. It has 3 main areas:

- **Toolbar**: Contains tools to select a scene, adjust an animation and render a video.
- **Timelines area**: Add and edit the timelines and animation sequences.
- **Configurator**: Configure the parameters of the current scene and the selected animation sequence.

Toolbar

Rysunek 98: Toolbar

The toolbar contains tools to:

- Add and remove scenes.
- Control the playback of the animation.
- Set the duration of the current scene.
- Add a camera timeline.
- Start a render.

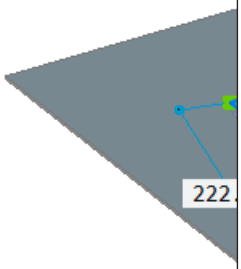
Tabela 29: Toolbar

Tool	Name	Description
	Add Scene	Add an empty scene with a default duration.

Tabela 29: Toolbar

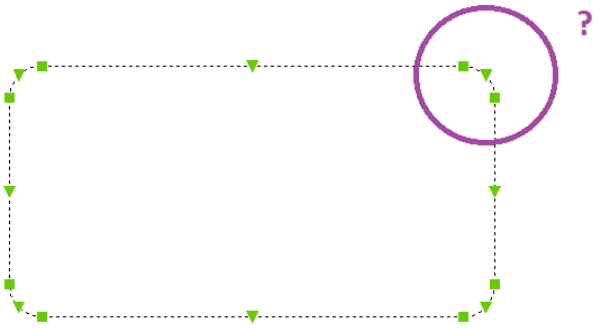
Tool	Name	Description
	**Scene Selector**	Select the current scene.
	Remove Scene	Remove the current scene from the document.
⏮	**To previous sequence**	Jump to the start of the previous animation sequence.
▶	**Play**	Start the animation.
⏸	**Pause**	Pause the animation.
■	**Stop**	Stop the animation and return the cursor to the start position.
⏭	**To next sequence**	Jump to the start of the next animation sequence.

Tabela 29: Toolbar

Tool	Name	Description
	**Render video**	Open a dialog box to start the render of a 2D video.
	Toggle configurator	Toggle the configurator panel.

Timelines Area

Use the **Timelines Area** to add and remove timelines, to add and remove animation sequences, and to display the animation schedule. Click on an animation sequence to select it, or drag the start or end of the bar to adjust the duration. The selected animation sequence displays in blue, other sequences display in gray. Each timeline in the timelines area can be temporarily disabled to exclude it from the animation.



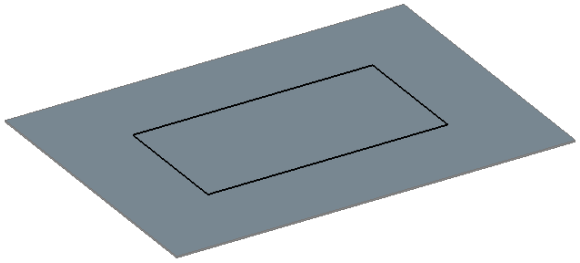
Rysunek 99: Timelines Area

The **Timelines Area** has a cursor that represents the current position in the animation. When the animation is played, the cursor represents the current playback position. You can also manually move the cursor to a new position at any time. When the cursor is moved, all parameters change to match the current frame. The camera position is also updated.

To make sure that the model state corresponds to the current animation position, click .

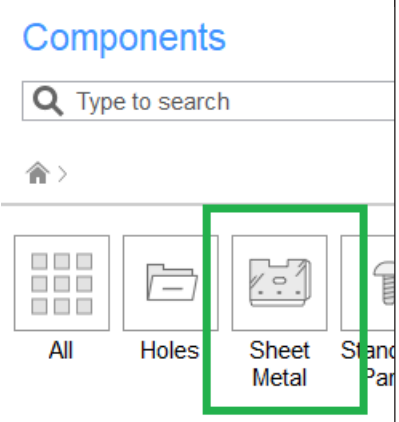
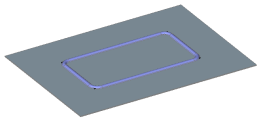
Uwaga: You can adjust this manually at any moment but this will not affect the animation.

Time Scale

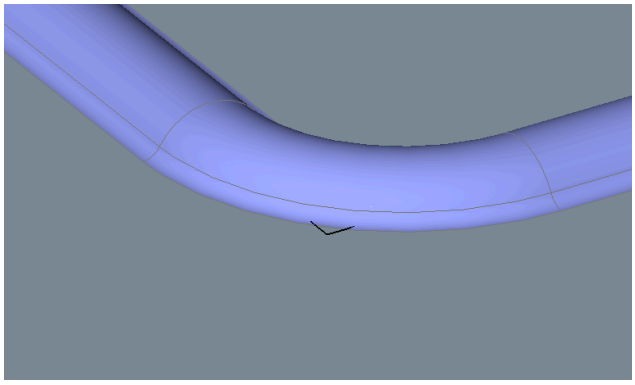


Rysunek 100: Time Scale

Tabela 30: Time Scale

Tool	Name	Description
	**Add Timeline**	Add the selected parameter to the timeline.
	**Object Selector**	Select the object to add to the timeline.
	Update model	Update the model state with respect to the current cursor position.
	**Time Scale**	Display the time marks along the timeline.
	**Cursor**	Display the current position of the playback.

Timeline



Rysunek 101: Timeline

Tabela 31: Timeline

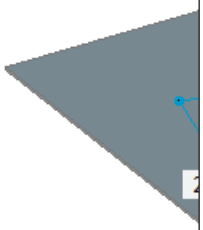
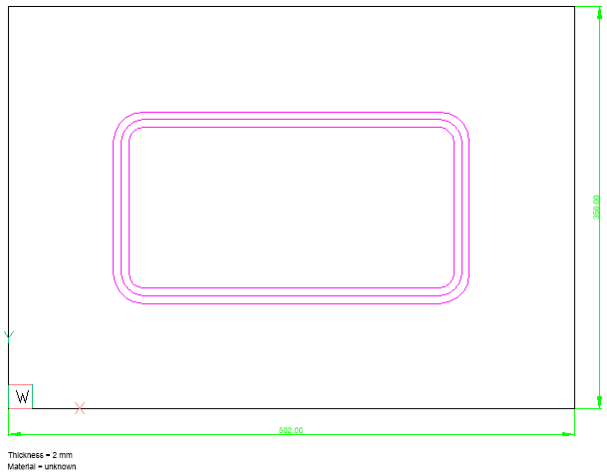
Tool	Name	Description
	**Timeline Enabled**	Indicates that the timeline is enabled. Click to disable the timeline.
	**Timeline Disabled**	Indicates that the timeline is disabled. Click to enable the timeline.
	**Timeline Object**	Displays the name of the associated object.
	**Add Sequence**	Add a new animation sequence, at the cursor position.
	Remove Timeline	Remove a timeline from the animation.

Tabela 31: Timeline

Tool	Name	Description
	Animation Sequence	Represents the duration and position of an animation sequence. Left-click to select the sequence.
	**Selected Animation Sequence**	Represents the duration and position of the selected animation sequence. You can see and change the properties in the **Configurator**. Move the mouse to the end of the sequence, until the cursor changes to a double-sided arrow. Then left-click and drag to change the start time, end time and duration. Left-click at the middle of the sequence and drag to a new position at the timeline, without changing the duration.

Configurator

Access all the properties of the animation scene and the selected sequence via the **Configurator**. All properties are grouped by object.



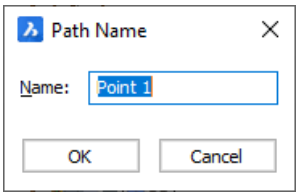
Rysunek 102: Configurator

- **Scene**: Properties of the current scene.
  - **Name**: Name of the scene.
  - **Duration**: Duration of the scene, in seconds.
- **Sequence**: Properties of the selected sequence.
  - **Sequence type** (read only): The type of the sequence.
  - **Object name** (read only): The name of the object controlled by that sequence.
  - **Start time**: Start time of the sequence, in seconds. The duration will be adjusted, the end time will remain unchanged.

- **Duration**: Duration of the sequence, in seconds. When changed, the end time will be adjusted with respect to the start time and duration.
- **End time**: End time of the sequence, in seconds. When changed, the duration will be adjusted, the start time will remain unchanged

- **Parameter** (for parametric sequences)
  - **Start value**: Start value of the parameter.
  - **End value**: End value of the parameter.
- **Camera** (for camera sequence)
  - **Camera path type**: Camera path entity type. May take one of the following values:
    - **Point**: The camera is fixed in the given point.
    - **Path**: The camera moves along the given path.
  - **Camera point** (when the **Camera path type** is set to **Point**). The following controls are displayed:

Tabela 32: Camera point

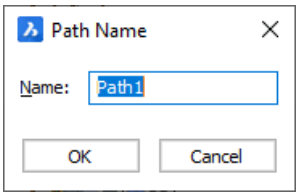
	List of previously used points. Select the point from the drop-down list to use one.
	Click to select the point in the model. After the point is selected, a dialog box is displayed:  **Rysunek 103:** Path Name • **Name**: Enter the point name. • **OK**: Click to accept the point selection. • **Cancel**: Click to cancel the point selection.
	Click to remove the point from the list.

  - **Camera path** (when the **Camera path type** is set to **Path**). The following controls are displayed:

Tabela 33: Camera path

	List of previously used paths. Select the point from the drop-down list to use one.

Tabela 33: Camera path

	Click to select the path in the model. After the path is selected, a dialog box is displayed:  **Rysunek 104:** Path Name • **Name**: Enter the path name. • **OK**: Click to accept the point selection. • **Cancel**: Click to cancel the point selection.
	Click to remove the path from the list.

- **Target path type**: Type of the target path. Target defines the point the camera will look at the given moment. Can take one of the following values:
  - **Point**: The target is fixed to a given point.
  - **Path**: The target moves along a given path.
- **Target point** (when the **Target path type** is set to **Point**). The following controls are displayed:

Tabela 34: Target point

	List of previously used points. Select the point from the drop-down list to use one.
	Click to select the point in the model. After the point is selected, a dialog box is displayed: **Rysunek 105:** Path Name • **Name**: Enter the point name. • **OK**: Click to accept the point selection. • **Cancel**: Click to cancel the point selection.
	Click to remove the point from the list.

- **Target path** (when the **Target path type** is set to **Path**). The following controls are displayed:

Tabela 35: Target path

	List of previously used paths. Select the point from the drop-down list to use one.

Tabela 35: Target path

	Click to select the path in the model. After the path is selected, a dialog box is displayed: **Rysunek 106:** Path Name • **Name**: Enter the path name. • **OK**: Click to accept the point selection. • **Cancel**: Click to cancel the point selection.
	Click to remove the path from the list.

Uwaga: Similar to *ANIPATH*, all selected points and paths are stored in the drawing.

21.4.12.5 Animation Playback

Use the animation playback controls to preview an animation.

- To **start** an animation, click the **Play** button.

  The playback starts from the cursor position.
- To **pause** the animation, click the **Pause** button.
- To **stop** the animation and move the cursor to the start of the animation, click the **Stop** button.

You can jump to the next or the previous animation sequence (in chronological order) with the **Jump to Next Sequence** and **Jump to Previous Sequence** tools respectively.

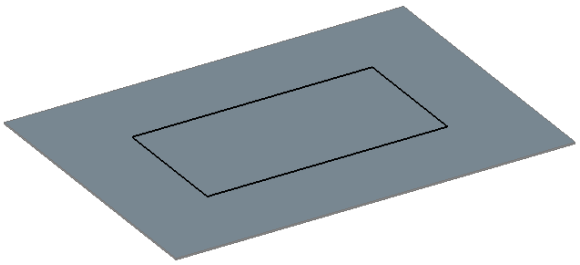
Uwaga:

- During playback, the parameters involved in the scene and the camera position are changed directly in the drawing. For complex models, the recalculation of the model may take a significant amount of time. You can use the cursor to check the key moments of the animation instead.
- During playback, the camera will move with any set camera sequences. If no camera sequences are present, the current view is used. The output video is the same as the current viewport.
- After the playback finishes, the model will automatically revert to the state before the animation was started. In some cases, it may cause the model to be in a different state than the cursor position would dictate. To update the model with respect to the current cursor position, click  .

21.4.12.6 Render

1 Click the **Render video** button to render the animation as a 2D video.

  The *Render video* dialog box displays:

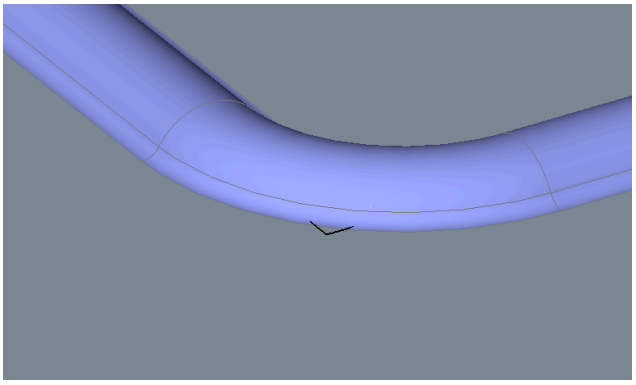


Rysunek 107: Render video

2 Adjust the render parameters:
   - **Render frame rate (fps)**: The frame rate.
   - **Resolution**: The resolution of the video.
   - **File format**: The format of the video (AVI, MPG and WMV formats are supported).
   - **Start time**: The time position the render will start.
   - **End time**: The time position the render will end.

3 Click the **Render** button to start the render.

   A file dialog box displays:



Rysunek 108: Save Video As

4 Specify the location and the name of the output file.

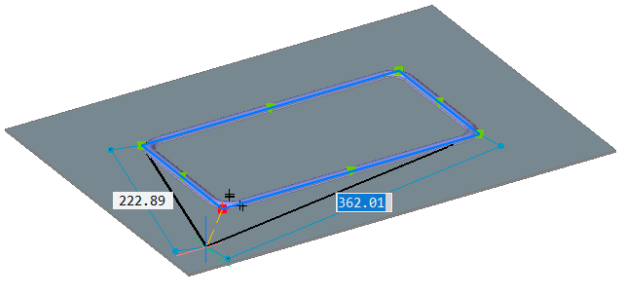
5 Click **Save** to start the render.

   A preview shows the progress.

21.4.12.7 To add a new scene

1 Click the **Add Scene** button.

   The Add scene dialog box displays:



Rysunek 109: Add scene

2 Enter the scene name.

3 Enter the scene duration in seconds.

4 Click **Add** to add the scene.

21.4.12.8 To remove a scene

1 Select the scene with the **Scene Selector**.

2 Use the **Remove Scene** tool.

21.4.12.9 To add an animation of a parameter

1 Add the desired parameter to the timeline:
- Select the parameter with the **Parameter Selector**.
- Use the **Add Timeline** tool to add a new timeline.

2 Place the cursor at the desired start position on the **Time Scale**.

3 Use the **Add Sequence** tool to add a new animation sequence,
or double-click at the timeline where you want to add a new sequence.

4 Drag to adjust the desired duration of the sequence,
or use the **Configurator** to set the desired start time, end time or duration, in seconds.

5 Use the **Configurator** to set the desired start and end values for the parameter.

21.4.12.1 To add a camera movement
0

1 If applicable, add a camera timeline by selecting **Camera** from the **Object selector**.

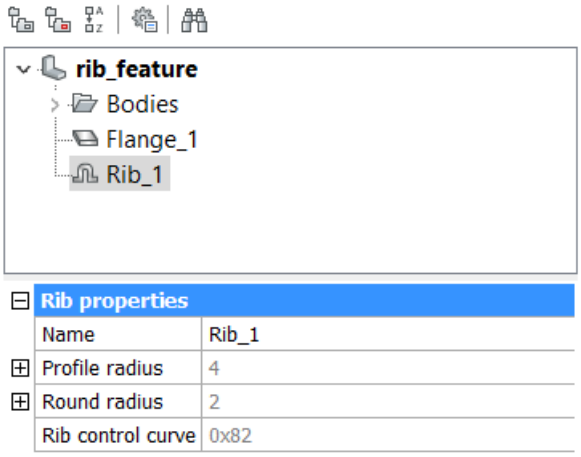
2 Place the cursor at the desired start position on the **Time Scale**.

3 Use the **Add Sequence** tool on the camera timeline to add a new camera sequence,
or double-click at the timeline where you want to add a new camera sequence.

4 Click and drag to adjust the desired duration of the sequence,
or use the **Configurator** to set the desired start time, end time or duration, in seconds.

5 Use the **Configurator** to set the camera path:
- Choose the **Camera path type** from the following:
  - To specify that the camera position should be fixed, or

- **Path** to specify that the camera should move along the given path.
- Use the Camera point property to select the point or the **Camera path** to select the path.



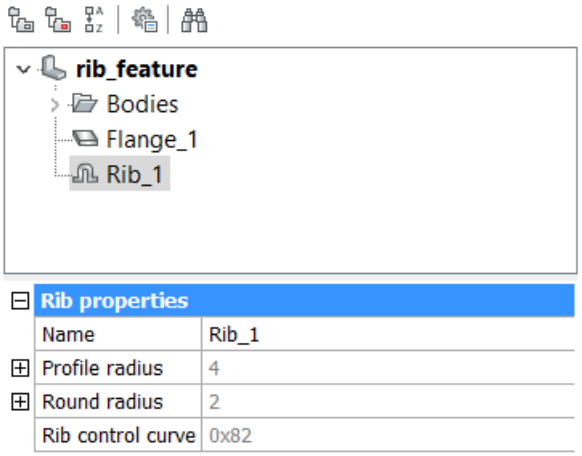
- Select the property for editing, then click to select the path or the point on the model,
  or select one of the previous points or paths from the drop-down list.

6 Use the **Configurator** to set the target path. The **target** is the point where the camera will focus on.

To choose the **Target path type** do one of the following:

- Specify that the target should be fixed.
- **Path** to specify that the target should move along the given path.

Use the **Target point** property to select the point or the **Target path** to select the path:



Select the property for editing, then click to select the path or the point on the model,
or select one of the previous points or paths from the drop-down list.

21.4.12.1 To remove an animation sequence

1

1 Right-click the sequence.

2 Select **Delete** from the context menu.

As an alternative you can:

1 Left-click the sequence.

2 Hit the **Delete** key.

21.4.12.1 To disable or enable a timeline

2

1 Check the timeline status icon. If the icon is , then the timeline is disabled. If the icon is , the timeline is enabled.

2 Click the timeline status icon to toggle the status.

Animation sequences on disabled timelines are not included in the animation.

21.4.12.1 To animate an exploded view

3

As the animation editor is not supported in Block editor or reference editing modes, only exploded views inserted in the model space can be animated.

1 Insert the exploded view in model space.

2 If applicable, suppress the exploded view and the model, if you want to hide or show them during animation.

3 Select the exploded view object in the **Object selector**.

4 Click to add a timeline.

5 Use the **Add Sequence** tool on the exploded view timeline to add a new exploded view sequence, or double-click at the timeline where you want to add a new sequence.

6 Click and drag to adjust the desired duration of the sequence,
or use the **Configurator** to set the desired start time, end time or duration, in seconds.

7 If applicable, use the **Configurator** to set the start step and the end step of the exploded view, or to enable the autohide option.

21.4.12.1 To render a video

4

See the section Render.

21.5 Arkusz Blachy

21.5.1 Metody projektowania konstrukcji blachowych

Projektowanie arkusza blachy umożliwia modelowanie części arkusza blachy i generowanie ich rozwiniętych reprezentacji z informacjami produkcyjnymi.

Możesz łatwo i szybko tworzyć złożone części blaszane za pomocą BricsCAD, ponieważ proces projektowania różni się od procesu produkcyjnego. Nie myśl w kategoriach płaskiego arkusza, który powinien zostać wycięty i zgięty, ale modeluj swoją część bezpośrednio podczas tworzenia brył za pomocą narzędzi do modelowania bezpośredniego.

Samouczek: Przedstawiamy BricsCAD Sheet Metal

Samouczek: Projektowanie od podstaw w BricsCAD Arkusz Blachy

Możesz tworzyć części arkusza blachy w BricsCAD na cztery różne sposoby (lub w dowolnej ich kombinacji):

1 Utwórz kołnierz bazowy z zamkniętego profilu płaskiego, a następnie dodaj kołnierze:
   - Pociągnij za krawędzie, aby utworzyć dodatkowe kołnierze krawędzi z zagięciami.
   - Twórz częściowe kołnierze z automatycznym tworzeniem odpowiednich podcięć gięcia.
   - Wyciągnij kilka sąsiednich krawędzi, aby utworzyć kilka kołnierzy jednocześnie z automatycznym tworzeniem podcięć narożnych i połączeń między nimi.
   - Zegnij kołnierz, aby utworzyć nowy kołnierz z zagięciem z jego części.
2 Utwórz kombinację kołnierzy i wyciągniętych zagięć łączących dwa profile 3D.

   Użyj narzędzia Wyciągnięcie złożone arkusza blachy.
3 Utwórz kombinację kołnierzy i zagięć z profilu płaskiego.

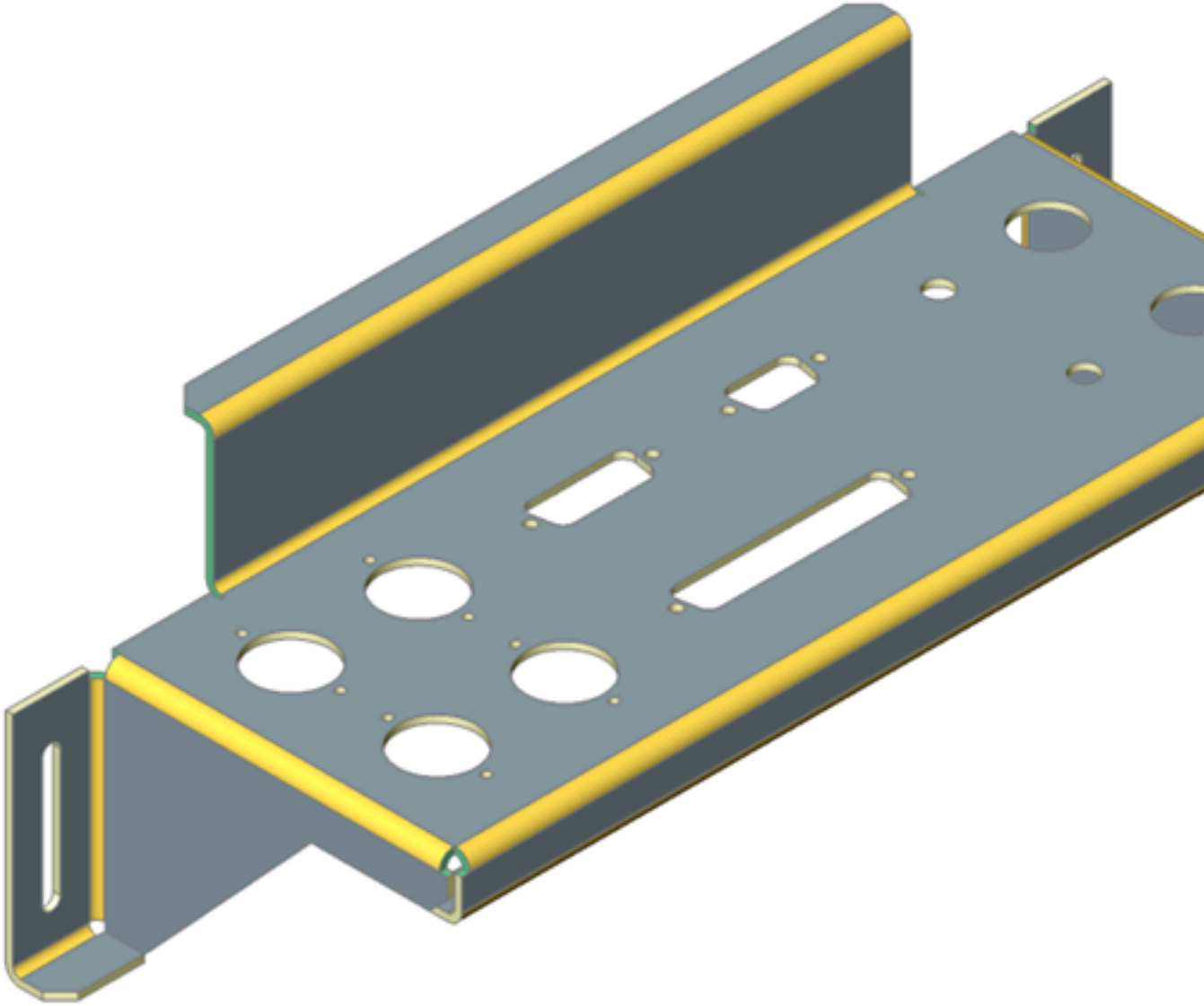
   Użyj narzędzia Wyciągnięcie arkusza blachy.
4 Utwórz korpus skorupowy z bryły 3D.

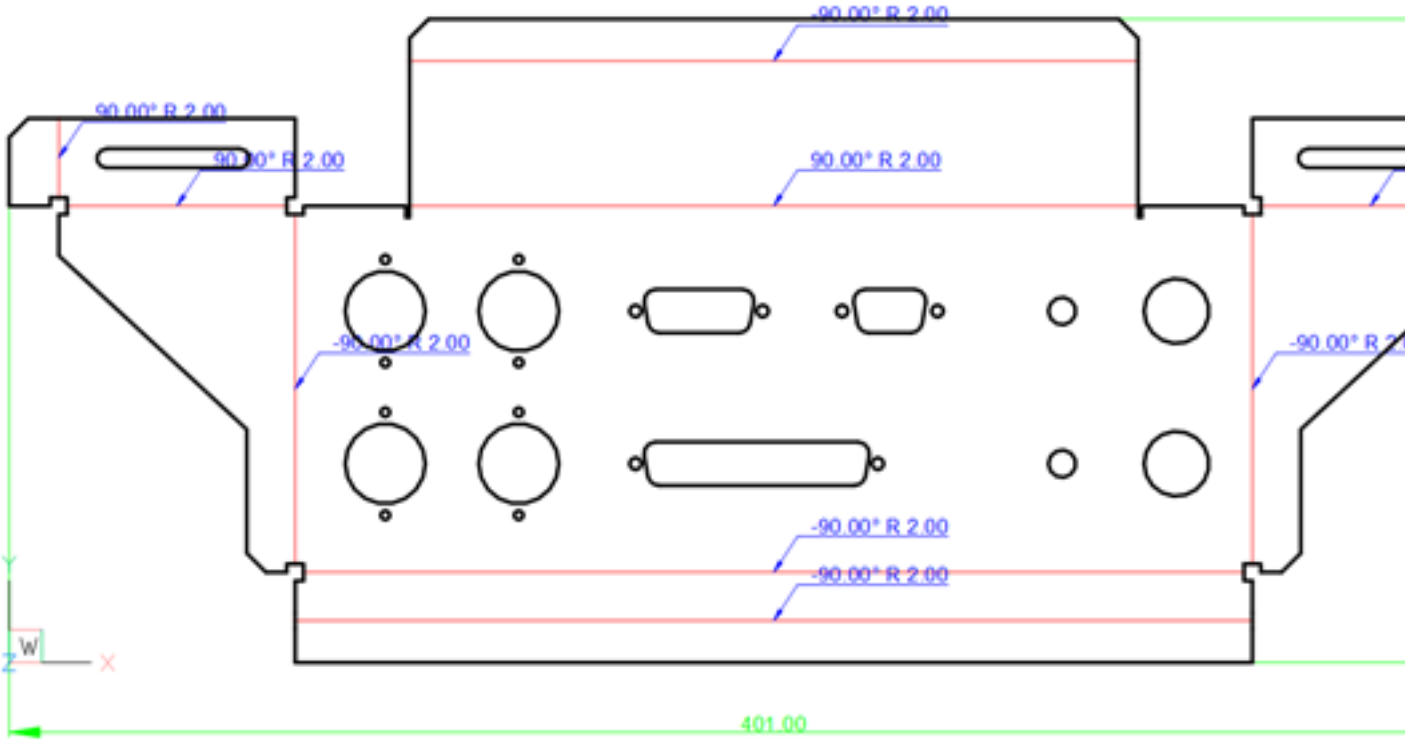
   Posługiwać się *Skorupa* opcja *SOLIDED* polecenie o żądanej grubości, a następnie przekonwertuj je na część arkusza blachy za pomocą *SMKONWERSJA* polecenie i utwórz zagięcia i połączenia z jego twardych krawędzi.
   - Wykonaj otwory rysując zamknięte profile na czołach kołnierzy i przepychając je przez kołnierze.
   - Dostosuj grubość materiału i promień gięcia za pomocą wstępnie zdefiniowanych parametrów.
   - Zastosuj operacje modelowania bezpośredniego i wiązania 3D, aby jeszcze bardziej dostosować swój projekt, zawsze zachowując intencje projektowe w zakresie elementów arkusza blachy.
   - Automatycznie generuj rozwiniętą reprezentację części i wyślij ją do systemu CAM, eksportując odpowiedni rysunek 2D z adnotacjami zgięć w formacie .dwg lub .dxf plik.

Możesz uszczegółowić część arkusza blachy, korzystając z następujących operacji:

- Wykonaj otwory rysując zamknięte profile na czołach kołnierzy i przepychając je przez kołnierze.
- Wstaw operacje formularza z biblioteki. Biblioteka parametrycznych funkcji formularza jest instalowana z BricsCAD Sheet Metal i możesz tworzyć własne funkcje formularza.
- Utwórz elementy formy żebra z krzywej 2D leżącej na kołnierzu za pomocą polecenia *SmRibCreate*.
- Dostosuj grubość materiału i promień gięcia za pomocą wstępnie zdefiniowanych parametrów.
- Zastosuj operacje modelowania bezpośredniego i wiązania 3D, aby jeszcze bardziej dostosować swój projekt, zawsze zachowując intencje projektowe w zakresie elementów arkusza blachy.

Gdy część arkusza blachy jest gotowa, można ją automatycznie rozłożyć i wysłać wynik do inżyniera produkcji, eksportując odpowiedni rysunek 2D.





Samouczek: Przetwarzanie wsadowe w BricsCAD Sheet Metal

21.5.2 Sheet Metal Commands and Toolbars

Tabela 36: Sheet Metal Commands and Toolbars

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Create a base (initial) flange of a sheet metal part from a closed planar profile.
			Create a sheet metal part by extruding a polyline in the direction of the normal of the curve plane. Thickness is assigned with respect to the external dimensions of the polyline.
			Create sheet metal part with lofted bends and flanges from 2 non-coplanar curves.

Tabela 36: Sheet Metal Commands and Toolbars

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Automatically recognize flanges, bends and lofted bends in a 3D solid.
			Create one or more flanges on a sheet metal part by pulling one or more edges of an existing flange.
			Create a new flange from a closed contour and attach it to an existing sheet metal part.
			Change the reference side for selected flanges with an optional shift of the flange geometry by thickness.
			Rotate a flange.
			Create one or more closed hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more open hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more Teardrop hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Create one or more Round hem features on a sheet metal part by pulling one or more edges of an existing flange.
			Insert form features.
			Create form feature from given set of faces.
			Explode form feature in flanges and bend if possible.

Tabela 36: Sheet Metal Commands and Toolbars

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Create an associative rib form feature on a flange face from a planar 2D curve.
			Replace form features (including recognized ones) in sheet metal parts with form features from a built-in or user library.
			Convert hard edges (sharp edges between flange faces) and junctions into bends.
			Convert bends to lofted bends.
			Bend an existing flange along a line, obeying k-factor for given bend radius.
			Convert hard edges into junctions.
			Change a symmetrical junction feature to one with overlapping faces.
			Close gaps between 2 arbitrarily oriented flanges.
			Create single tab feature to a sheet metal part by switching existing junction into the tab.
			Create array tab feature to a sheet metal part by switching existing junction into the tab.
			Make proper corner and bend reliefs. Corner reliefs are built on corners which have 3 or more adjacent flanges. Bend reliefs are built on the start and end of a flange edge.
			Convert a corner relief to a circular relief.
			Convert corner reliefs to a rectangular relief.

Tabela 36: Sheet Metal Commands and Toolbars

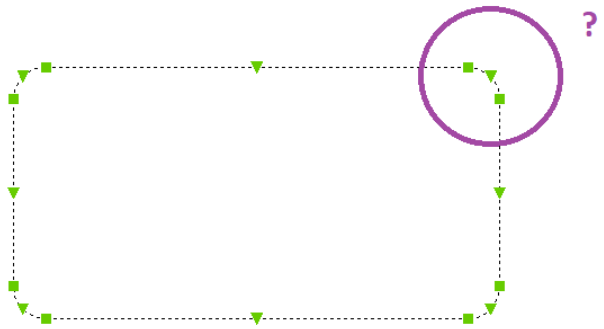
Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
			Convert corner reliefs to a VType relief.
			Convert corner reliefs to a Smooth relief.
			Convert corner reliefs to a Round relief.
			Convert corner reliefs to a Rip relief.
			Split a flange along a line drawn on its face.
			Split a flange by a single vertex in the corner.
			Split a flange by a single vertex in the corner and automatically suggest multiple splits of a similar type.
			Split thickness faces of a sheet metal part by imprinted edges, according to adjacent flanges and bends.
			Remove sheet metal data from the selected faces or entities.
			Remove a bend or a junction by restoring a hard edge between two flanges; remove a flange with all the bends adjacent to it. The adjacent flanges are extended up to a junction configuration with the flange being deleted. Remove miter by restoring geometry being cut by the feature.
			Fix problems specific to Sheet Metal parts, both global and local workflows are supported. Optionally heal lofted bend features: merge adjacent bends and provide tangential connection with flanges.
			Fix problems specific to Sheet Metal parts. Only selected flange or wrong bend are repaired.

Tabela 36: Sheet Metal Commands and Toolbars

Sheet Metal Toolbar icon	Flyout Toolbar icon	Command	Description
		Same form features	Select all identical form features in the sheet metal model.
		Hard edges	Select all hard edges on sheet metal parts.
		Non-orthogonal thickness edges	Select all non-orthogonal thickness edges on the sheet metal part.
		Flat edges	Select flat edges on the sheet metal part.
		side of the sheet metal part	Select side of the sheet metal part.
			Create a consistent set of 3D constraints for the selected sheet metal part.
			Unfold the sheet metal body.
		Associatively	Open an additional window with model's unfolded representation.
			Export a sheet metal solid to the .osm (Open Sheet Metal) file format.
			Export the unfolded representation of a sheet metal body as a 2D profile in .dxf / .dwg file format.
			Perform a batch export of assembly components to unfolding, creating a .dxf per each 3D solid recognized as sheet metal part.
			Toggle the *FEATURECOLORS* system variable.

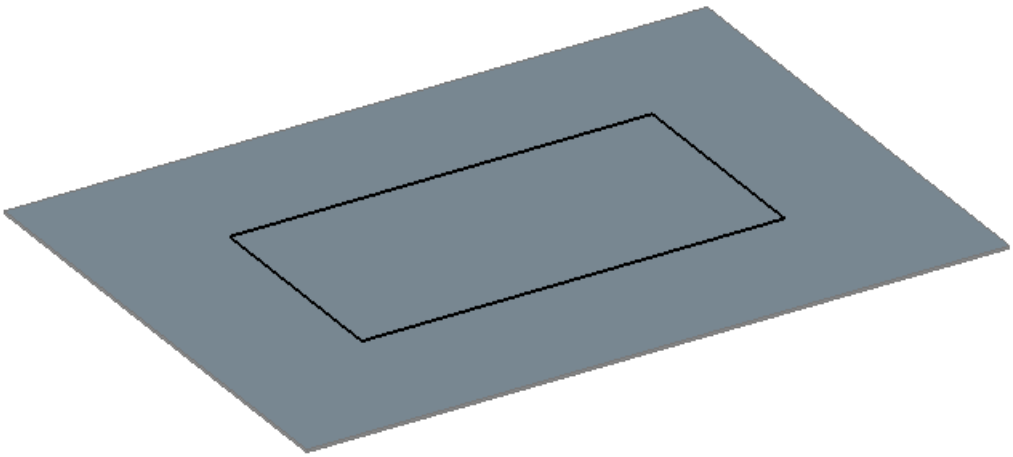
The Sheet Metal tools are available:

- In the **Quad** cursor menu (depending on the feature under the cursor):



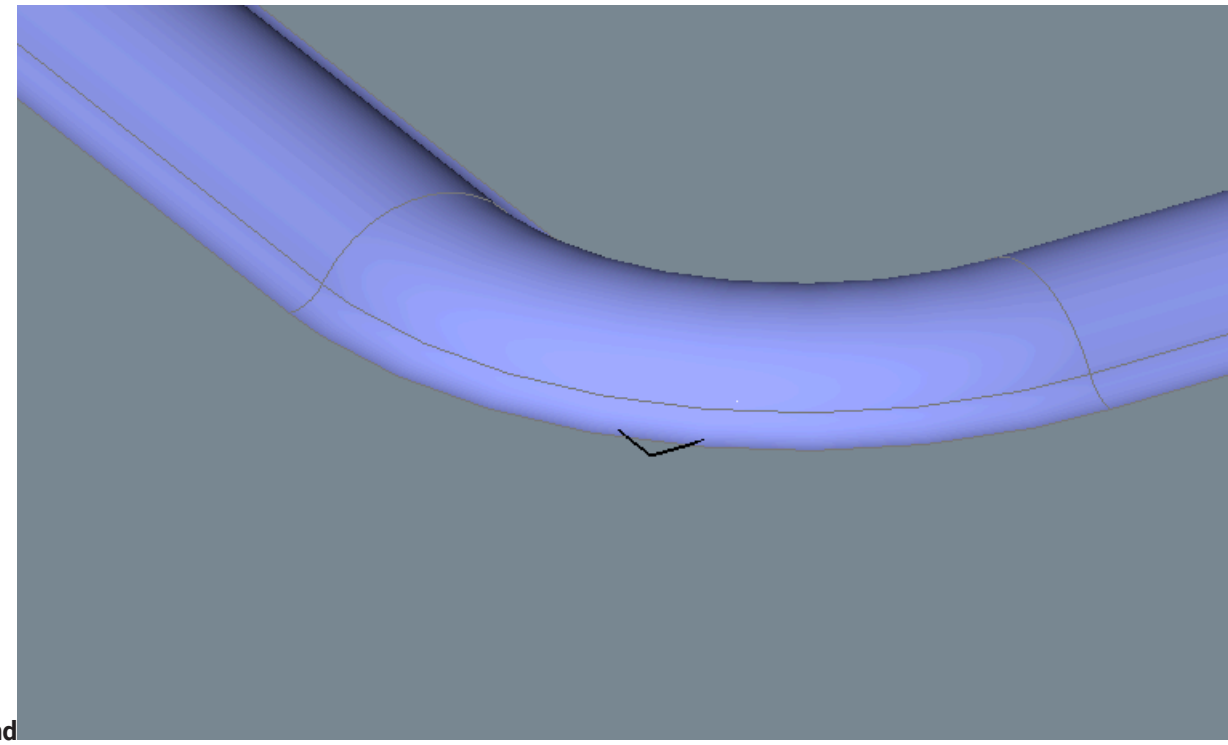
Rysunek 110: Quad cursor menu

- On the **Sheet Metal** toolbars:
  - **Sheet**



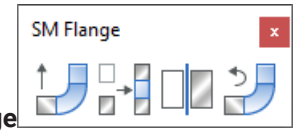
Metal

  - **SM**

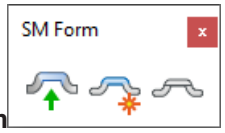


Bend

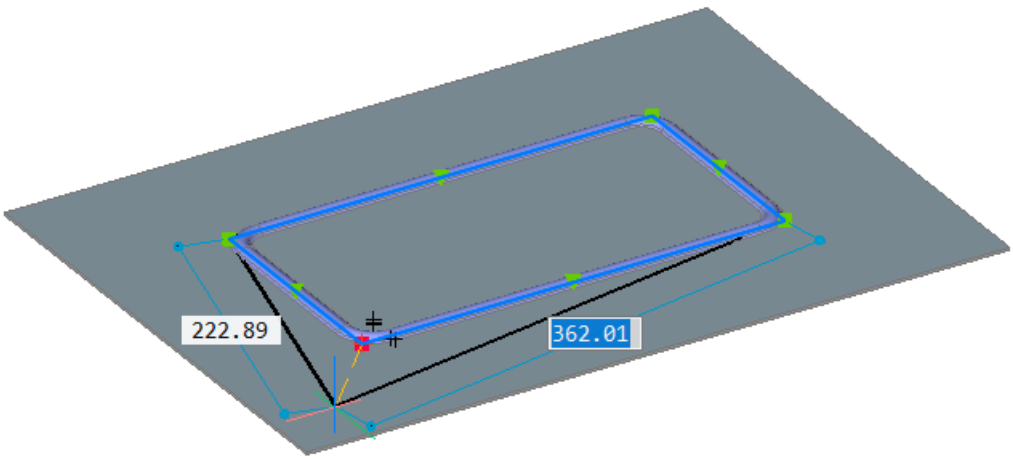
- **SM Flange**



- **SM Form**

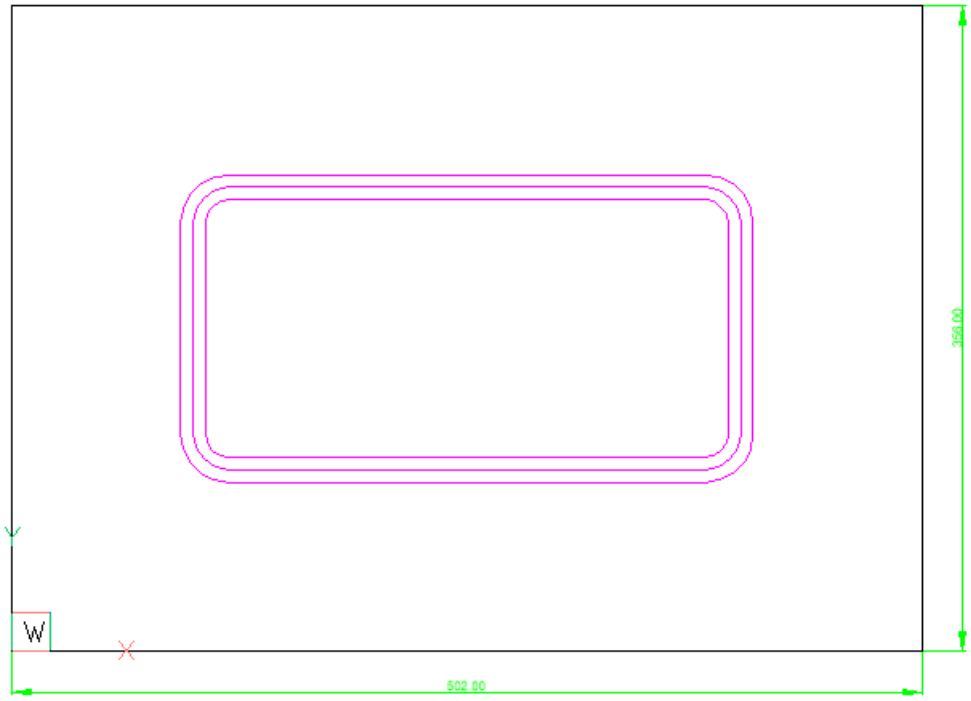


- **SM**



Hem

- **SM**

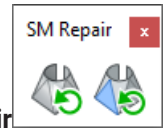


Junction

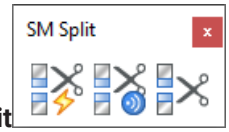
- **SM Relief**



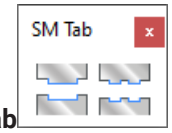
- **SM Repair**

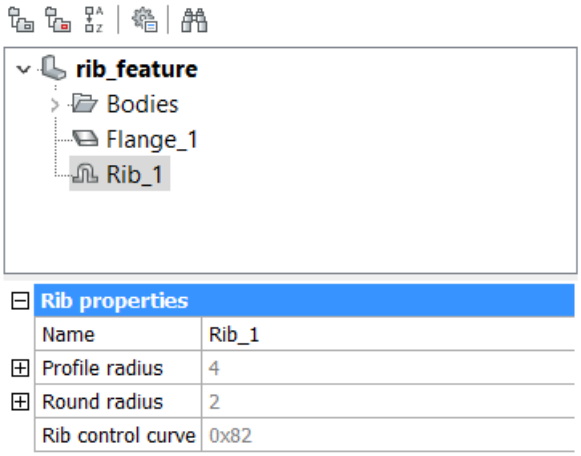


- **SM Split**



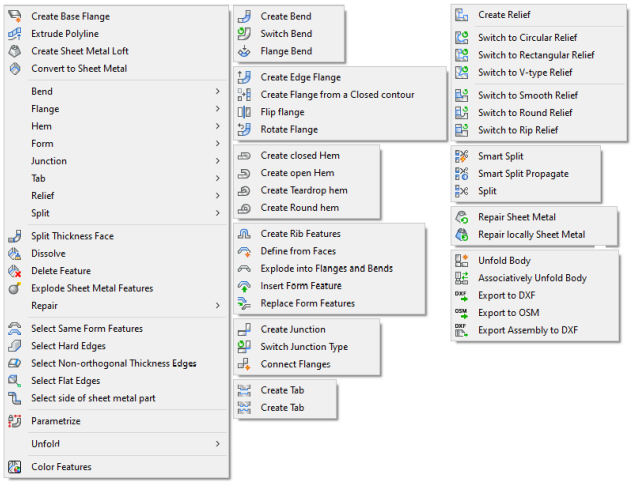
- **SM Tab**





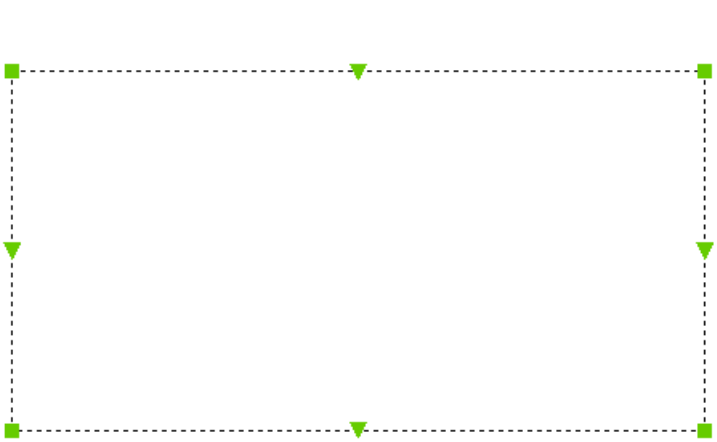
- **SM Unfold**

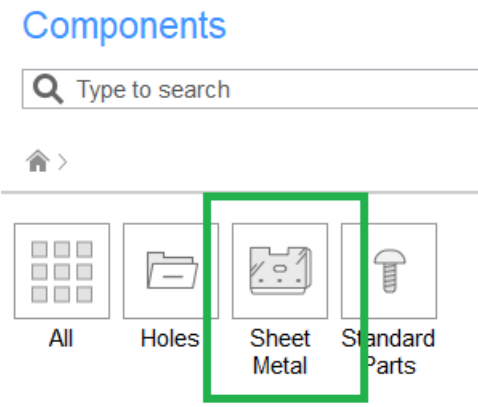
- In the **Sheet Metal** menu (**Mechanical** workspace only):



Rysunek 111: Sheet Metal menu

- In the **Sheet Metal** tab of the **Mechanical** workspace





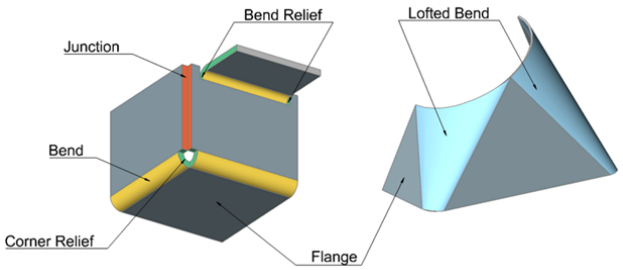
ribbon.

See also Direct modeling operations.

21.5.3 Sheet Metal Features

21.5.3.1 Overview

Features are smart groups of faces in a 3D part. Each feature maintains a specific spatial and parametric relationship between its faces and some adjacent faces. The features allow you to embed design intent into your model. The features are created automatically by a particular geometric operation you apply. Images below show the sheet metal features in BricsCAD:



Rysunek 112: Example 1 of sheet metal features



Rysunek 113: Example 2 of sheet metal features

21.5.3.2 Flange

The main feature of all sheet metal parts is a flange, which consists of 2 parallel planar faces located in front of each other (with some solid volume between them) on the distance, which is significantly smaller than the linear size of the faces. This distance is equal to the material thickness. When you modify the model, this distance relation is always maintained automatically. Other faces, which are adjacent to flange faces and do not belong to bends are called thickness faces. They are always perpendicular to the flange faces.

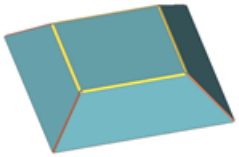
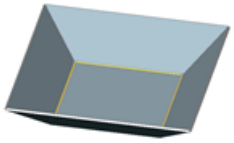
Flange features are created by the following commands: *SMFLANGEBASE*, *SMEXTRUDE*, *SMLOFT*, *SMFLANGEEDGE*, *SMFLANGEBEND* and *SMCONVERT*.

If the *FEATURECOLORS* system variable is *ON*, the display color distinguishes between the reference face of a flange and its opposite side. Parametric modifications of a sheet metal part, such as thickness change, do not move reference faces, if possible. The *SMFLIP* command swaps sides of a selected flange

so that reference faces are on the other geometric side of the flange; optionally the command shifts the flange over the thickness of the sheet metal part.

The images below illustrate the color legend for the flange faces of the same part.

Tabela 37: Flange

	
Reference Faces	Other Faces

21.5.3.3 Bend

Two flanges are connected by a bend. A bend consists of 2 coaxial cylindrical faces, which are always tangent to the adjacent planar faces of the flanges.

Bend features are created by *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND* and *SMCONVERT* commands.

21.5.3.4 Zero Bend

This is a design concept of bend, but geometry has a cylindrical face for the external side of the bend with a radius equal to thickness. Instead of an internal face, there is an edge. So effectively geometry represents a bend of zero radii, which is used for design purposes. *SMUNFOLD* operation supports bends of this kind and generates dimensions as if these bends were regular bends with a radius equal to thickness.

21.5.3.5 Wrong Bend

Wrong bend features are recognized by the *SMCONVERT* command and reflect assumptions on the user's design intent. Like regular bends, wrong bends connect flanges, but their faces can be non-coaxial, or they can have non-tangent connections with adjacent flange faces, which prevent using the geometry as regular bend features. Before unfolding the part, all wrong bends must either be repaired by *SMREPAIR* or removed by *SMDELETE* and then recreated by *SMBENDCREATE*.

21.5.3.6 Lofted Bend

A lofted bend feature consists of 2 ruled surfaces located in front of each other (on the offset equal to the material thickness) with some solid volume between them. A ruled surface can be described as the set of points swept by a moving straight line. Examples are cylinders, cones, helicoids. Lofted bends can be created by the thickening of ruled surfaces. A lofted bend can connect 1 or 2 flanges or be stand-alone, without flanges at all. Some combinations of adjacency between regular bends and lofted bends are also allowed.

Differences between bends and lofted bends:

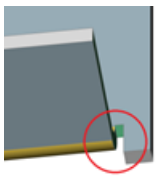
Tabela 38: Differences

	Bends	Lofted Bends
Surface	Cylindrical	Cylindrical, conical or spline
Unfolding	Exactly	The approximate unfolding of lofted bends is controlled by an adjustable number of bend operations, which are called the Number of Subdivisions. The more operations are performed, the smoother the real part will be produced and the better it approximates the initial design. Cylindrical and conical lofted bends have an option for exact analytical unfolding.

Lofted bend features are created by the *SMLOFT* and *SMCONVERT* commands.

21.5.3.7 Bend Relief

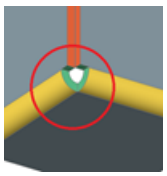
A group of faces representing a technological cut between 2 flanges of different width connected with a bend. The bend relief feature maintains the distance between 2 opposite faces of the cut.



Bend relief features are created by *SMFLANGEEDGE*, *SMBENDCREATE* and *SMRELIEFCREATE* commands.

21.5.3.8 Corner Relief

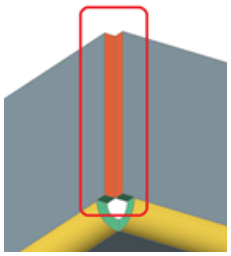
A group of faces representing a cut in the corner where 3 or more flanges meet together. This cut is needed to prevent unwanted material deformation during bending of the sheet metal part. A corner relief feature maintains the form and size of this cut.



Corner relief features are created by *SMFLANGEEDGE*, *SMBENDCREATE* and *SMRELIEFCREATE* commands.

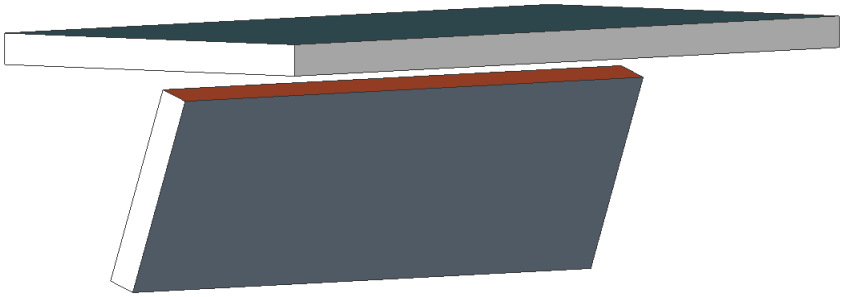
21.5.3.9 Junction and T-junction

A junction feature consists of 2 thickness faces of adjacent flanges, which are not connected by a bend. A junction can be symmetric or overlapping. Junction features maintain the minimal distance between the flanges: the junction gap.



A T-junction feature consists of a thickness face of a single flange, which is located near a face of another flange,. Usually, a T-junction precedes creation of a T-tab.

For clarity, the junction gap in the image below is intentionally rather large.

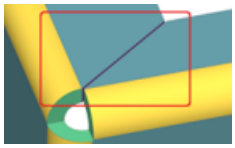


Junction features are created by *SMFLANGEEDGE*, *SMJUNCTIONCREATE* and *SMFLANGECONNECT* commands.

T-junction features are created by *SMJUNCTIONCREATE* and *SMFLANGECONNECT* commands.

21.5.3.10 Miter Feature

A miter feature represents a split between coplanar flanges created by the *SMFLANGEEDGE* and *SMSPLIT* commands. A miter feature maintains the miter gap.



21.5.3.11 Form and Rib Features

Form and rib features represent a rich variety of shapes which are produced by punching and wheeling.

Two representations of the form features are important: one in 3D and a 2D symbol in the unfolded geometry. In case of imported geometry the symbol is obtained by the projection of the 3D faces of the form feature. In case the form feature was inserted/replaced from the Form Feature Library, the unfolding symbol is taken from the library file.

Rib features represent the imprint on a flange made by a wheel following an arbitrary trajectory.

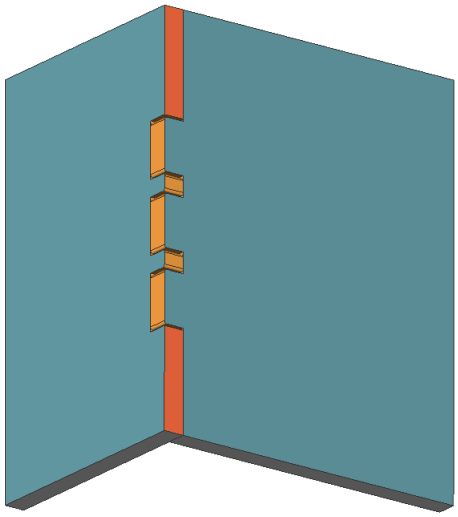
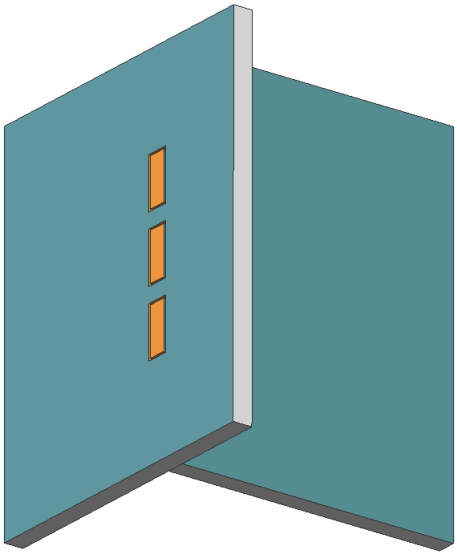
Form and rib features are created by *BMINSERT*, *SMRIBCREATE* and *SMCONVERT* commands.

21.5.3.12 Tab and T-tab

Both features are used to connect 2 flanges, restricting their mutual displacement.

Tab and T-tab features are created by the *SMTABCREATE* command.

Tabela 39: Tab and T-tab

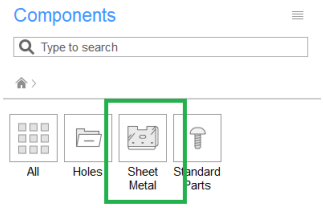
 **Rysunek 114:** Example of tab	 **Rysunek 115:** Example of T-tab
Tab feature	T-tab feature



21.5.3.13 Bevel

Bevel features represent chamfers on thickness faces of Sheet Metal parts. Both sloped and orthogonal faces define a chamfer feature. Bevel features must be shared strictly by a single flange and can not be adjacent to bend features.

Bevel features can be recognized or erased by the *SMCONVERT* command, and are supported by the *SMUNFOLD* command.



21.5.4 Oznaczanie kolorami narzędzi

Aby uzyskać lepsze wrażenia i łatwiejsze rozróżnianie elementów, różne kolory są stosowane do różnych typów elementów w geometrii. Te kolory służą wyłącznie do wizualizacji i nigdy nie będą wyświetlane w wygenerowanych dokumentach, widokach obszaru papieru, wstawkach blokowych itp.

Wyświetlanie kolorów funkcji jest kontrolowane za pomocą zmiennej systemowej *FEATURECOLORS* i szeregu zmiennych systemowych, po jednej dla każdego koloru funkcji. Chociaż domyślne kolory elementów są starannie dobierane, aby łączyć je ze sobą, można je ustawić inaczej w grupie ustawień **Arkusz blachy** w oknie dialogowym *Ustawienia* .

Rysunek 116: Arkusz Blachy

Aby przełączyć wyświetlanie kolorów funkcji:

- Kliknij przycisk narzędzia **Funkcje koloru** () na pasku narzędzi **Arkusz blachy**.
- Kliknij przycisk narzędzia **Funkcje koloru** () na pasku wstążki **Arkusz blachy**.
- Wybierz **opcję Kolor** z menu **Arkusz blachy**.

Samouczek: Kolorowanie elementów w BricsCAD Sheet Metal

21.5.5 Mechanical Browser for Sheet Metal

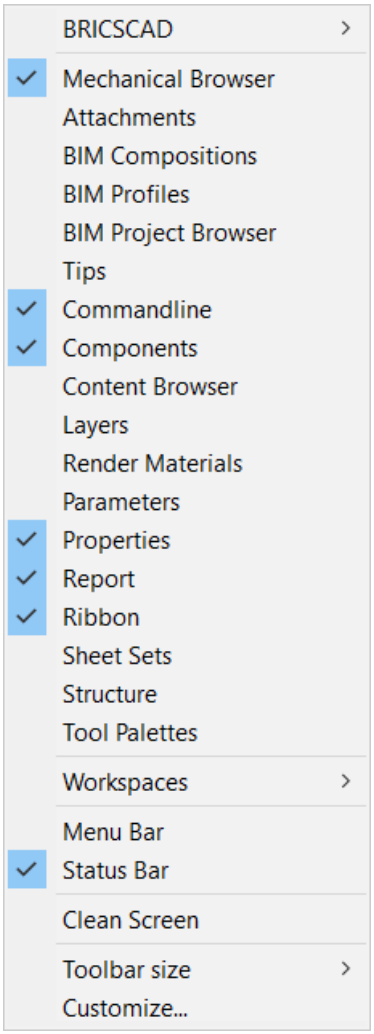
21.5.5.1 Overview

The Mechanical Browser opens automatically when switching to the **Mechanical** workspace.

To open the Mechanical Browser, manually do the following:

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

A context menu displays:

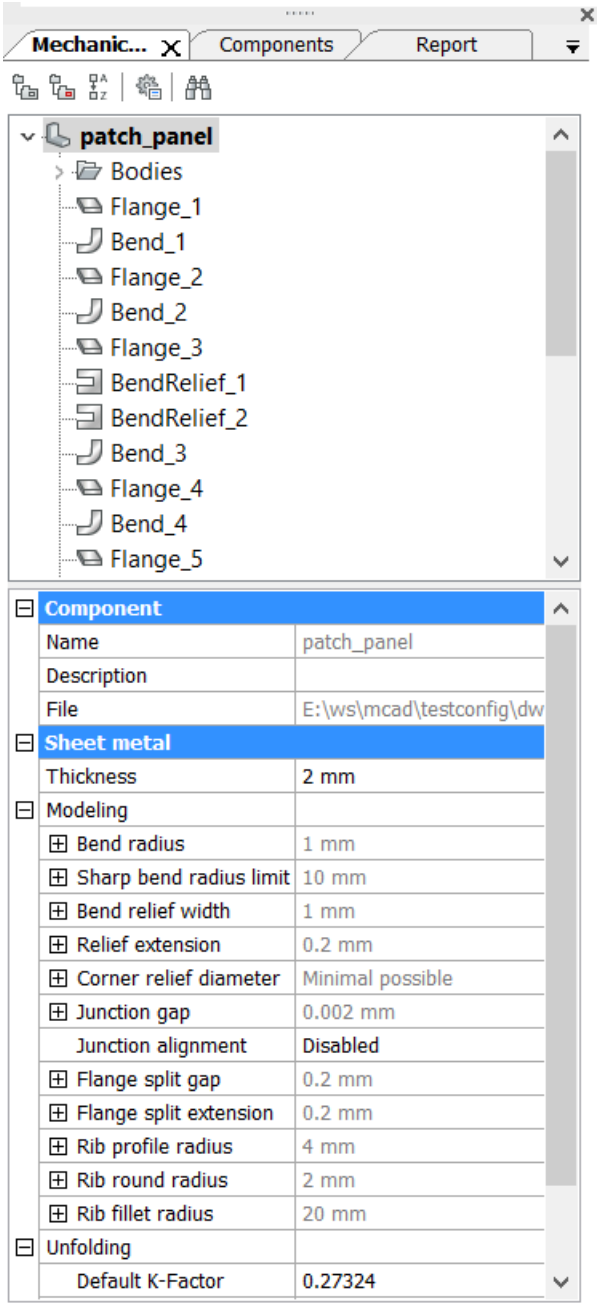


Rysunek 117: Context menu

2 Choose **Mechanical Browser**.

21.5.5.2 Sheet Metal Context

If you work with a sheet metal part (if you created any sheet metal feature), properties of the root node in **Mechanical Browser** contain "Sheet Metal" section (also called Sheet Metal Context associated with the document).



Rysunek 118: Mechanical Browser

Sheet Metal properties are represented as grouped fields.

The **Thickness** field controls the material thickness of sheet metal solids, which has to be equal for all solids of the document.

The Modeling group plays 2 roles:

- It provides default settings for Sheet Metal Commands, such as *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND*, *SMLOFT*, and *SMCONVERT* in a newly created model. The initial parameters are saved in the drawing, which allows creating a set of .dwt template drawings for each type of sheet metal models created in a company.
- New features created by the commands usually derive their properties from Sheet Metal Context. For example, the default value of **Radius Type** property for the bend feature is "Use global value". Thus when you change **Bend radius** in Sheet Metal Context, all the bend features with the derived property will change its radius. So the second application of Sheet Metal Context properties is that they allow globally change properties of sheet metal part.

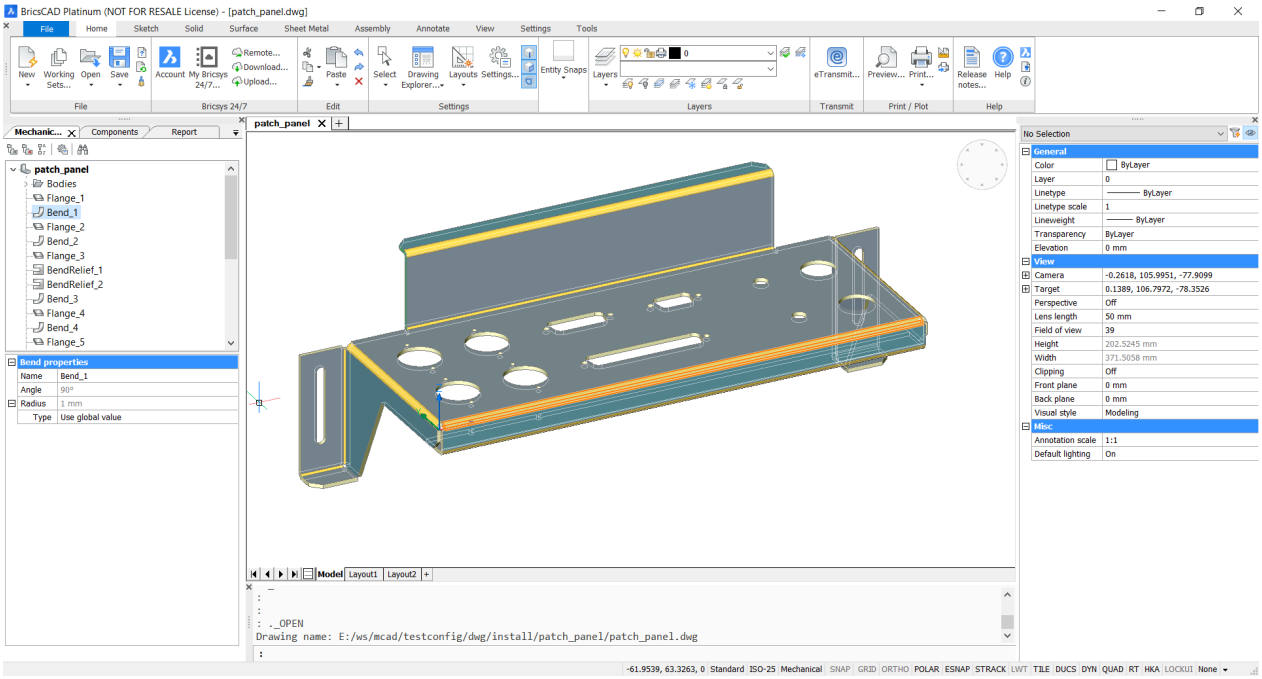
The **Unfolding** group of properties in Sheet Metal Context lists some properties of *SMUNFOLD*, *SMEXPORT2D* and *SMASSEMBLYEXPORT* commands.

Sheet Metal Context is created when first sheet metal part appears in the document and is initialized from settings listed in Initial Parameters. Typical workflows for creating sheet metal part are listed in Sheet Metal Design Methods. Sheet Metal Context is removed when all the features in the document are dissolved by the *SMDISSOLVE* command.

Uwaga: If you just select all sheet metal parts in the document and Erase them, Sheet Metal Context is kept, despite there are no sheet metal solids and no sheet metal features.

21.5.5.3 Working with features

All features of a sheet metal part are listed in the **Mechanical Browser**. When you select a feature in the **Mechanical Browser**, its faces are highlighted in the model area:



Rysunek 119: Mechanical Browser

Uwaga: **Sheet Metal** group of properties has changed to **Bend** properties. Clicking on feature in Mechanical Browser allows to:

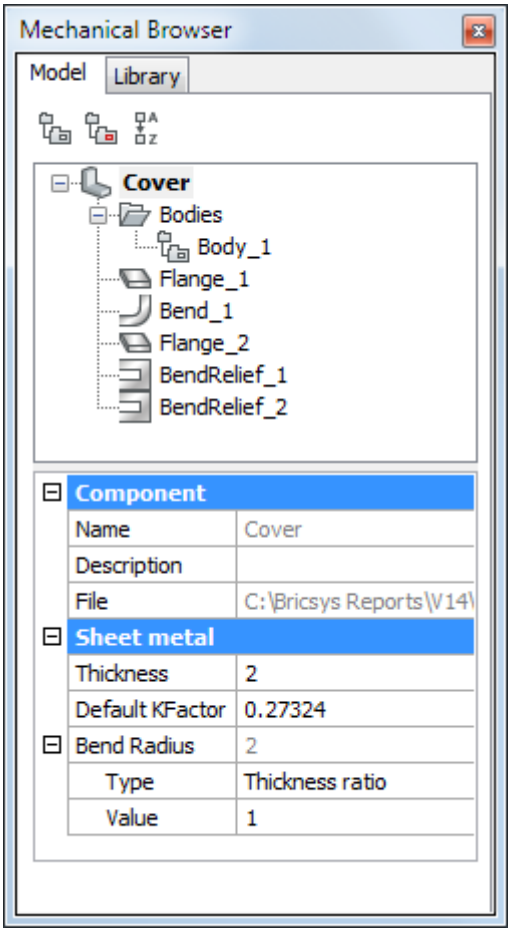
- Locate feature on the part by highlighting its geometry.
- See properties of given feature.

21.5.5.4 Changing the thickness

To change the thickness of a sheet metal part:

1 Select the root node in the **Mechanical Browser**.

2 Type a value in the **Thickness** field.

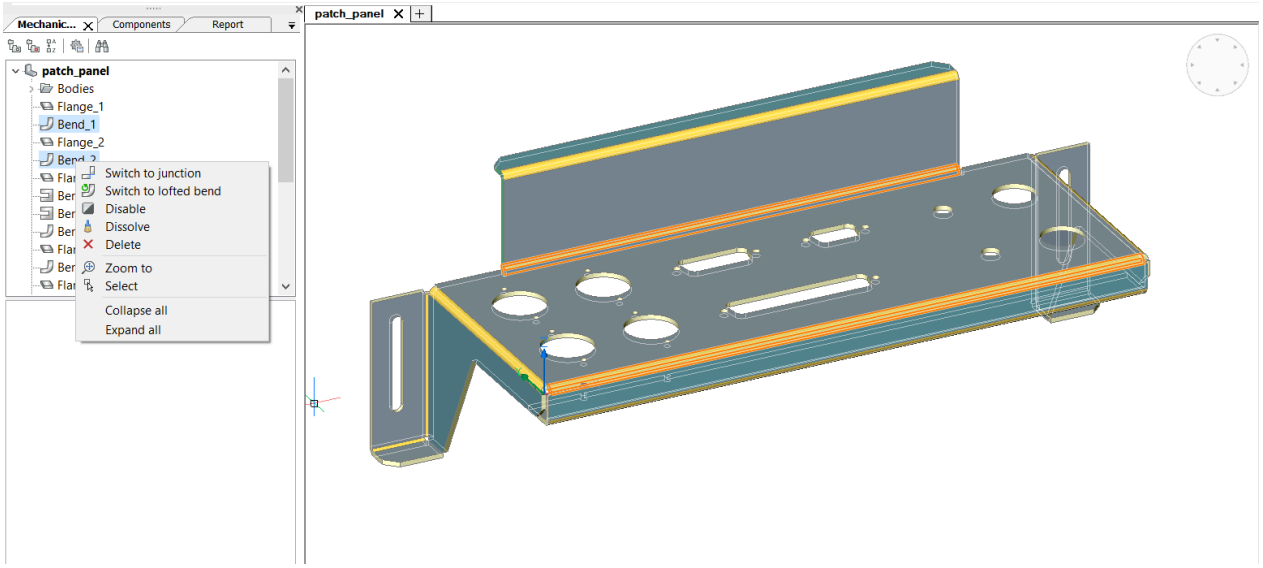


Rysunek 120: Thickness

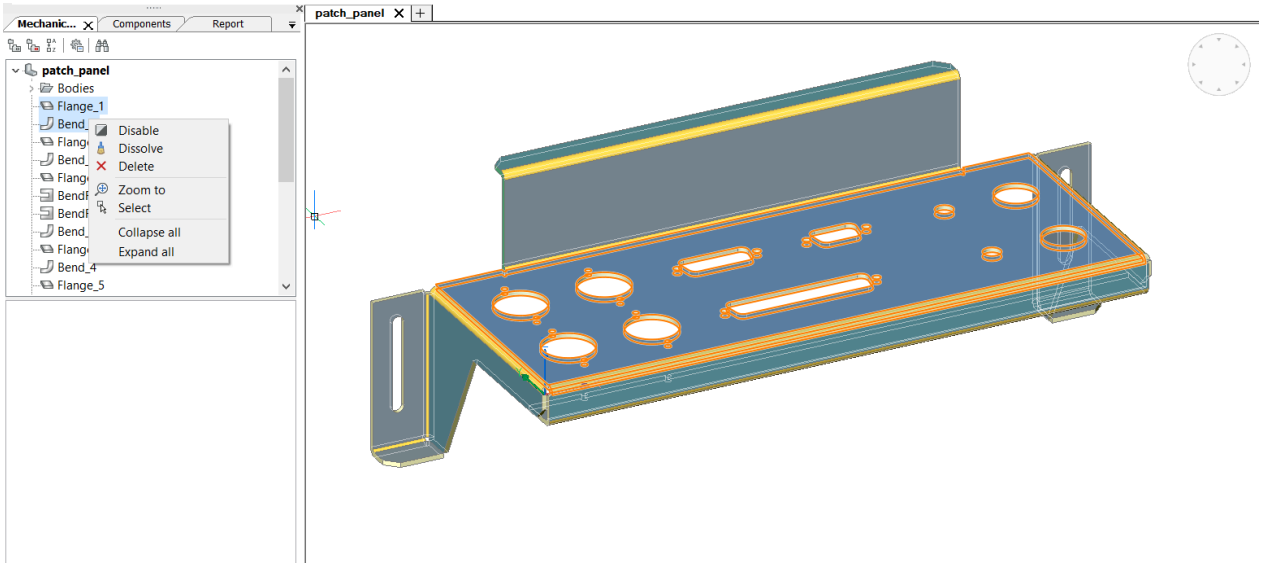
21.5.5.5 Feature operations

Mechanical Browser provides fast access to operations over given sheet metal feature, list depends upon the feature type. List of operations depends upon selected features and contains operations available for the selection.

Selecting 2 bends results in operations for bend feature:



Selecting flange and bend limits list to operations available for both features:



21.5.5.6 Understanding the difference between Disable, Dissolve and Delete

When you right-click a feature in the **Mechanical Browser**, then select **Dissolve** from the context menu the selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

If your intent is temporary forget about given feature, you can choose **Disable** option. The feature will remain in the **Mechanical Browser**, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by the *SMCONVERT* command.

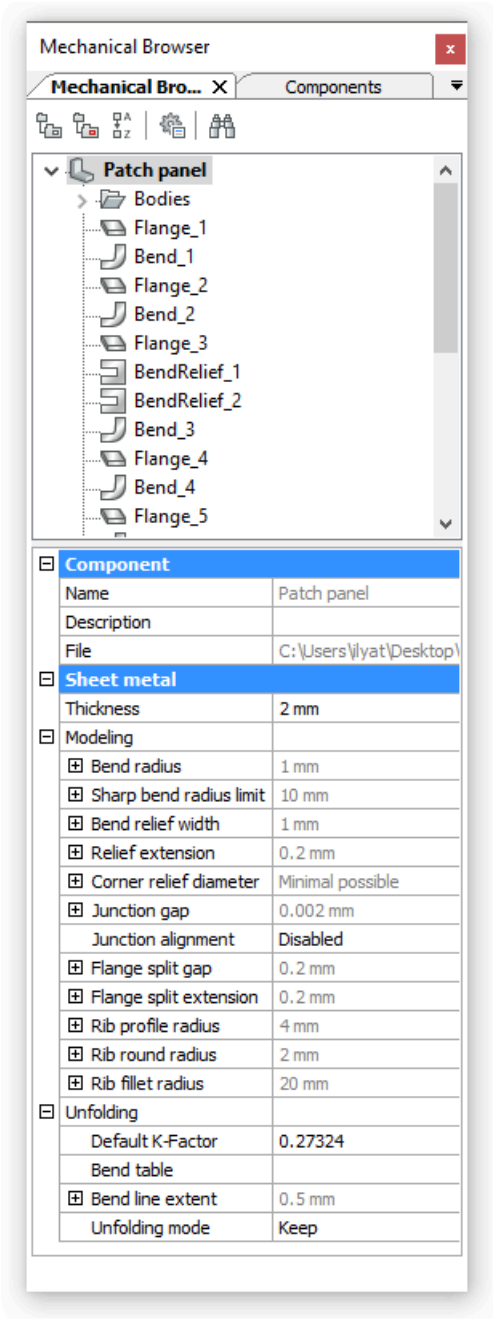
Some of Sheet Metal features support **Delete** operation in the **Mechanical Browser**, which is the analogue of the *SMDELETE* command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

21.5.6 Sheet Metal Global and Feature Properties

21.5.6.1 Overview

Sheet metal features and context menus are located in the **Mechanical Browser**. See Mechanical Browser for Sheet Metal to understand the general principles. This article describes the particular properties and context menus for each node type in the **Mechanical Browser**.

When working with sheet metal parts, clicking the root node in **Mechanical Browser**, displays the properties of entire drawing. Sometimes these properties are referred to as the global Sheet Metal Context associated with the drawing: each Sheet Metal part shares this context by default.



Rysunek 121: Mechanical Browser

Starting from BricsCAD V21, you can assign a local Sheet Metal context for given solid, which allows to define Sheet Metal parts with different design parameters, and in addition, to change the part thickness. Before BricsCAD V21, all the parts in modelspace had to have the same thickness, which forced the user to spread the parts with a different thickness to different blocks or components. Scenarios with multiple imported bodies engaging *SMCONVERT* and *SMASSEMBLYEXPORT* also benefit from this new feature. Another advantage is that the local Sheet Metal context can define its own Bend Table.

21.5.6.2 To switch the context of a solid to local

Do the following:

1 Select the body in **Mechanical Browser**.

2 In Sheet Metal section, switch **Context Type** from **Global context** to **Context for solid**.

Uwaga: The *SMCONVERT* command, in case of multiple input with different thicknesses, switches some contexts to local automatically.

Sheet metal global properties are located in the bottom part of the **Mechanical Browser**.

Tabela 40: Global properties

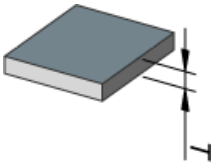
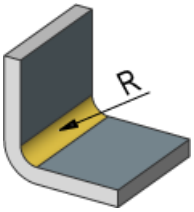
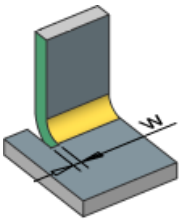
Global Properties	Description
Thickness	Global sheet metal material thickness. 
Bend radius	Default radius of bend features; it can be changed per bend as well. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Sharp bend radius limit	Maximum bend radius to be considered a sharp (regular) bend. Used in the *SMCONVERT* and *SMRELIEFCREATE* commands to distinguish between regular and lofted bends. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value.
Bend relief width	Width of the cut near a single bend. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 

Tabela 40: Global properties

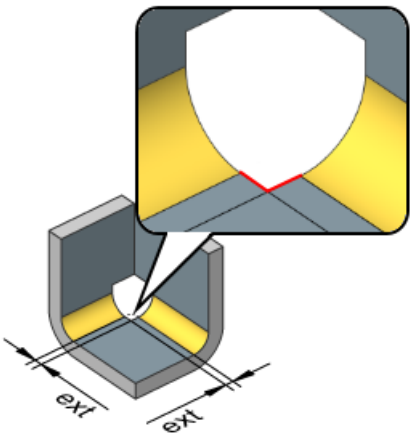
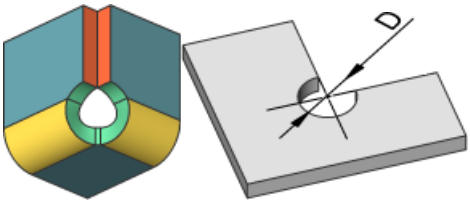
Global Properties	Description
Relief extension	Size of the bend or corner relief extension. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Corner relief diameter	Default diameter of a circular corner relief. **Type**: minimal possible or absolute value. 

Tabela 40: Global properties

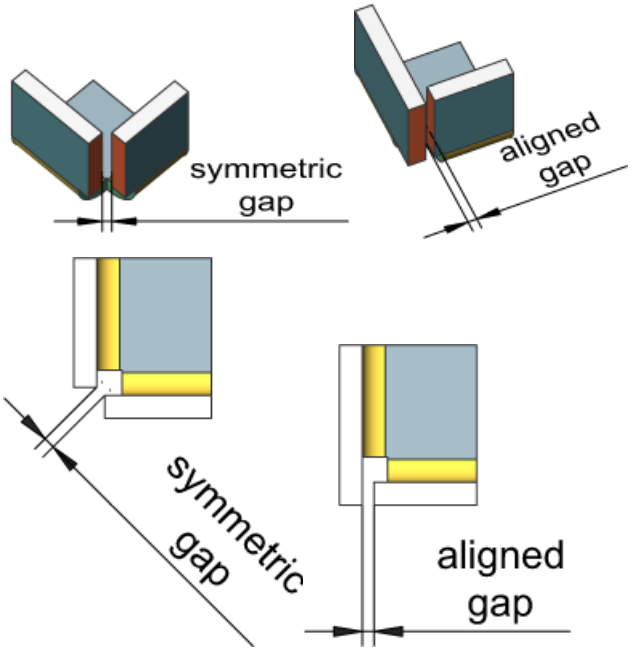
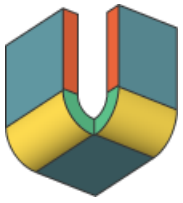
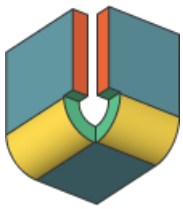
Global Properties	Description
Junction gap	Gap between 2 adjacent flanges. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Junction alignment	Control the alignment of junction faces with the corresponding corner relief faces. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. • **Junction Alignment**: enabled or disabled   **Rysunek 123:** Disabled **Rysunek 122:** Enabled

Tabela 40: Global properties

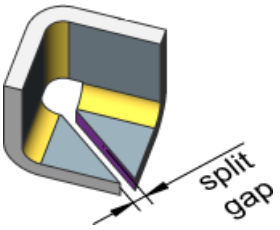
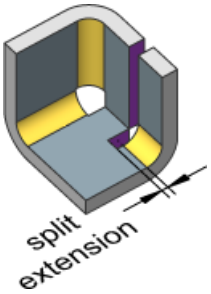
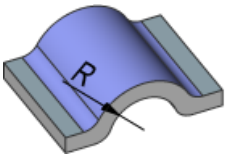
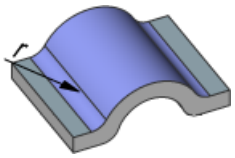
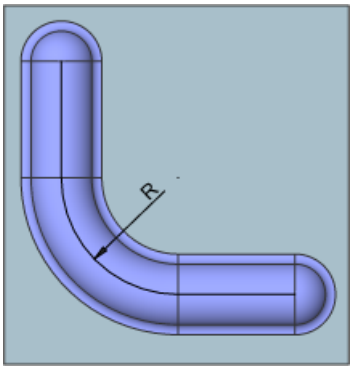
Global Properties	Description
Flange split gap	Gap between the thickness faces of a flange or the adjacent bend when splitting, see *SMSPLIT*. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Flange split extension	Control the depth cut of an adjacent flange in case of a flange split. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Rib profile radius	Radius of a tool, which creates the rib. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 
Rib round radius	Radius of the fillet between the flange and a rib. • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value. 

Tabela 40: Global properties

Global Properties	Description
Rib fillet radius	Radius of the rib control curve filleting. See Working with Form and Rib Features. • **Type**: either profile radius ratio or absolute value. • **Value**: numeric value. 
Unfolding	Parameters for the unfolded representation.
Default K-Factor	K-Factor for the bend radius equal to the thickness.
Bend table	Display the Open dialog for bend tables in CSV file format.
Bend line extent	Control how far a bend line extends beyond the unfolding boundary (zero and negative values are allowed). • **Type**: either thickness ratio or absolute value. • **Value**: either ratio or value.
Unfolding mode	Control the appearance of form features in the unfolded representation. • **Keep**: keep the 3D representation of the form feature. • **Remove**: completely remove form features from the unfolded representation. • **Project**: project all edges of form features as 2D entities. • **Contour**: project only the contour of form features as 2D entities. • **Symbol**: place 2D symbols at the insertion points in unfolded representation.

Feature properties display automatically when selecting a feature in the **Mechanical Browser**.

Right-click a feature to display the feature context menu.

Tabela 41: Feature Context Menus

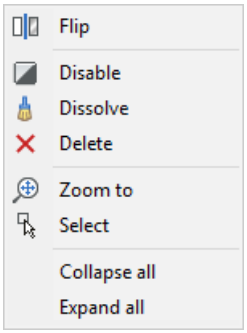
Feature Context Menus	Description
	Common tools: • **Disable**: temporarily disable a feature and skip its constraining. • **Dissolve**: remove a feature but keep its geometry. Design intent associated with the geometry of a dissolved feature will be removed. • **Delete**: See the *SMDELETE* command. Remove features from the Mechanical browser and delete the corresponding geometry in the model. • **Zoom to**: zoom onto the selected feature. • **Collapse all**: collapse the main component including all components and subcomponents. • **Expand all**: expand the main component and all components and subcomponents.
Flange feature	Right-click a flange feature:   **Flip**: swap the sides of a selected flange so that the reference face is on the opposite geometric side of the flange. This operation shifts the flange by its thickness, to preserve the position of the reference face.

Tabela 41: Feature Context Menus

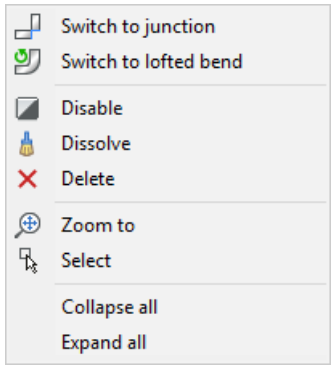
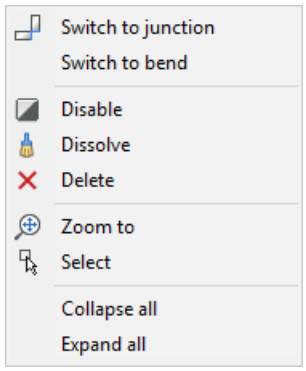
Feature Context Menus	Description
Bend feature	Right-click on a bend feature:  • **Switch to junction**: switch to a junction feature type with a gap between adjacent flanges. • **Switch to lofted bend**: switch to a lofted bend feature.
Lofted Bend feature	Right-click on a lofted bend feature:  • **Switch to junction**: switch to a junction feature type with a gap between adjacent flanges. • **Switch to bend**: switch to a bend feature.

Tabela 41: Feature Context Menus

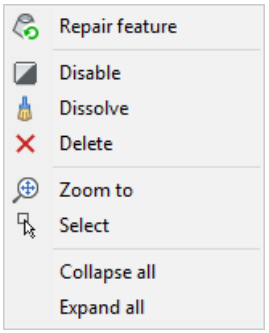
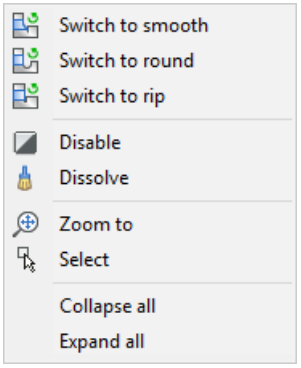
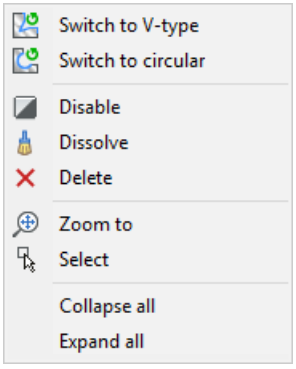
Feature Context Menus	Description
Wrong Bend feature	Right-click on a wrong bend feature:  **Repair feature**: repair a wrong bend feature by creating a bend feature with proper geometry, if possible.
Bend Relief feature	Right click on bend relief feature:  **Switch to smooth/round/rip**: switch a bend relief to the selected feature type.
Corner relief feature	Right-click on corner relief feature:  **Switch to V-type/circular**: switch a corner relief feature to the selected feature type.

Tabela 41: Feature Context Menus

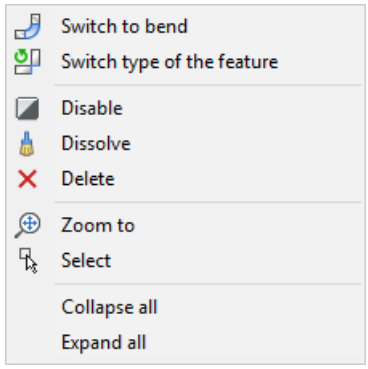
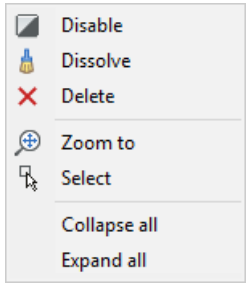
Feature Context Menus	Description
Junction feature	Right-click on junction feature:  • **Switch to bend**: switch to a bend feature. • **Switch type of the feature**: switch the alignment mode between symmetrical and aligned with one of the flanges.
Miter feature	Right-click on miter feature: 

Tabela 41: Feature Context Menus

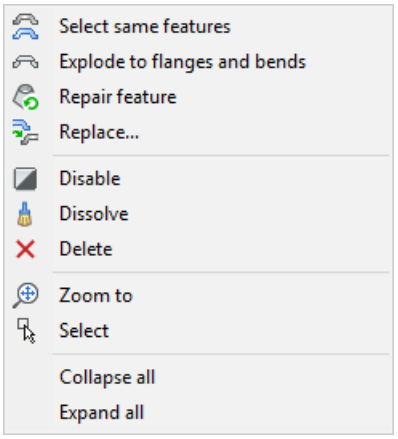
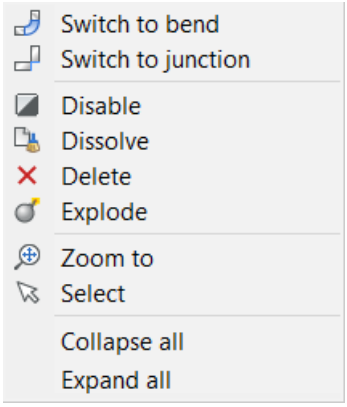
Feature Context Menus	Description
Form feature	Right-click on form feature:  • **Select same feature**: select all form features of the same type in the model. • **Explode to flanges and bends**: explode a form feature into a set of flanges and bends, if possible. • **Repair feature**: recalculate the insertion point of a form feature. • **Replace...**: display the Open dialog box in order to choose a replacement form feature.
Tab feature	Right-click on tab feature:  • **Switch to bend**: convert a tab feature to a bend feature • **Switch to junction**: convert a tab feature to a junction feature

Tabela 41: Feature Context Menus

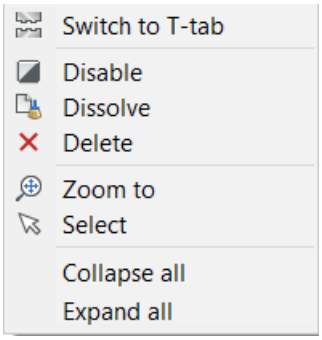
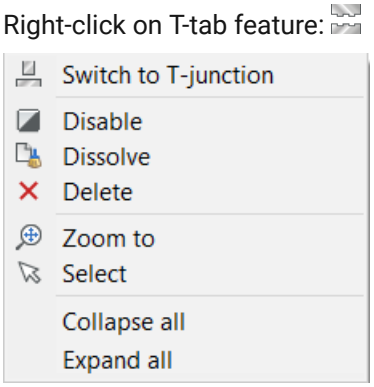
Feature Context Menus	Description
T-junction feature	Right-click on T-junction feature:  **Switch to T-tab**: convert a T-junction feature to a T-tab feature.
T-tab feature	Right-click on T-tab feature:  **Switch to T-Junction**: convert a T-tab feature to a T-junction feature.

Tabela 42: Feature Properties

Feature Properties	Description
	Common property: **Name**: the name of the selected sheet metal feature. Optionally type a name for the feature. The default name of a feature is the feature type plus an increment, for example, Flange_1, Flange_2, ...

Tabela 42: Feature Properties

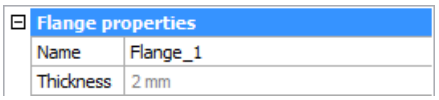
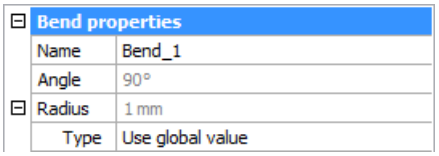
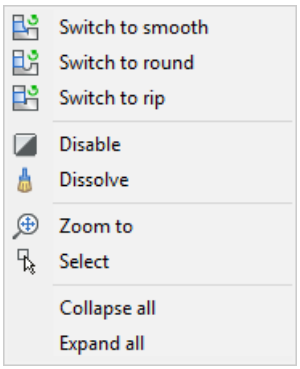
Feature Properties	Description
Flange feature	Click a flange feature:  **Thickness**: current thickness. The value is read-only; it equals the sheet metal component thickness property.
Bend feature	Click a bend feature:  • **Angle**: current angle of the bend feature; can be changed by the *SMFLANGEROTATE* command. • **Radius**: current bend radius. • **Radius type**: - Thickness ratio: ratio of thickness multiplied by a coefficient. - Absolute value: the global value (main sheet metal component properties).
Bend relief	Click a bend relief feature:  • **Relief type**: current relief type; either rectangular, smooth, round or rip. • **Width**: width of the cut near the bend; either global, thickness ratio or absolute.

Tabela 42: Feature Properties

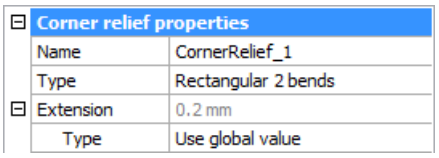
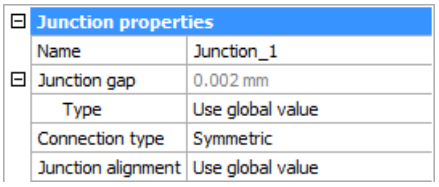
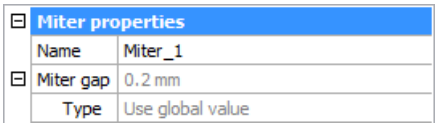
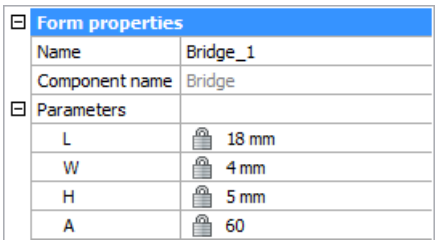
Feature Properties	Description
Corner relief	Click a corner relief feature:  Corner relief properties Name: CornerRelief_1 Type: Rectangular 2 bends Extension: 0.2 mm Type: Use global value • **Relief type**: current relief type; either rectangular 2 bends, rectangular, v-type or circular. • **Width**: width of the cut near corner; either global, thickness ratio or absolute.
Junction feature	Click a junction feature:  Junction properties Name: Junction_1 Junction gap: 0.002 mm Type: Use global value Connection type: Symmetric Junction alignment: Use global value • **Junction gap**: gap between 2 adjacent flanges. • **Junction gap type**: either global, thickness ratio or absolute. • **Connection type**: control the relative position of adjacent flanges; either symmetric or adjacent to one of the flanges. • **Junction alignment**: control the position of the faces of a junction feature; - global - enabled (aligned with the corresponding corner relief feature faces) - disabled.
Miter feature	Click a miter feature:  Miter properties Name: Miter_1 Miter gap: 0.2 mm Type: Use global value • **Miter gap**: gap between the side faces of the miter. • **Type**: either thickness ratio or absolute.

Tabela 42: Feature Properties

Feature Properties	Description
Form feature	Click a form feature:  • **Component name**: name of form feature component definition. • **Parameters**: contains the parameters of a parametric sheet metal form feature.
Bevel feature	Click a bevel feature:  • Bevel type: defines a type of the bevel, depending on the geometry. Possible values: one-side, land, X- and K- bevel. Upon a bevel type, parameters may alter. • Side bevel: parameters of chamfer on one side of this bevel. • Orthogonal bevel: parameters of orthogonal side.

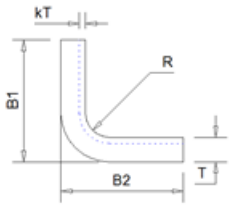
21.5.7 K-factor

21.5.7.1 Overview

When a metal sheet is bent with a press brake, its material is deformed, which is the reason for the difference in overall length of a sheet in folded and unfolded states. BricsCAD automatically considers this

deformation when you unfold a sheet metal part. Two approaches are used to compute it: a simplified one (K-factor) and a more advanced one.

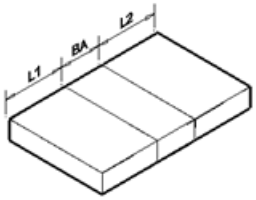
The material deformation properties for bending is based on the assumption that there is internal surface for the sheet metal part which is not deformed when the sheet is bent. From the variety of internal surfaces, one surface is selected which is equidistant from one side of the bend. The surface is local for each bend, and for simple cases it can be propagated on entire part, like the one in the picture. We assume that this surface is not stretched during bending.



T: thickness of the sheet metal part

R: internal radius of the bend

The K-factor (k) is the ratio of the location of the neutral surface to the material thickness. As a result, the neutral surface lies at a distance kT from the internal surface of the bend. The bend radius of the neutral surface equals R + kT. The K-Factor is a simple geometric calculation of the location of the neutral surface. Forming stresses and other unknown (error) factors are not considered. The K-Factor depends on many factors, such as the kind of material, bending type and tools. The K-Factor typically lies between 0.3 and 0.5. The default K-Factor for a bend radius equal to the material thickness (T) is 0.27324.



In the unfolded state of a sheet metal part, the flanges (L1 and L2) are not stretched. The bend part is indicated as BA: bend allowance. The BA equals the length of the unfolded neutral surface:

BA = BendAngle * (R + kT)

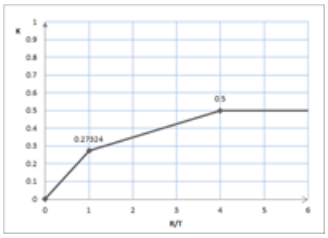
In fact it is hard to measure the K-factor or the bend allowance. The following formula allows to calculate the tangential Bend Deduction (BD):

BD = B1 + B2 - Unfolded Length = B1 + B2 - (L1 + L2 + BA)

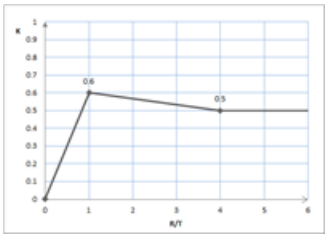
For simple cases the K-Factor value can be overridden in the browser. For maximal precision a bend table containing tangential bend deductions has to be provided. For each sheet metal part, you can either specify a K-factor or use the default one.

21.5.7.2 Changing the K-factor for a Sheet Metal Part

To change the K-Factor for a sheet metal part, select the root node in the Mechanical Browser and type a value in the **K-Factor** field. The value must lie in the range [0, 1], since the neutral surface is located inside the sheet metal part. In BricsCAD the K-Factor is defined for bend radius equal to the material thickness (R/T = 1). To compute the value of **K-Factor** for an arbitrary bend radius, BricsCAD uses a special interpolation technique which is proved to be valid for industrial sheet metal applications. The image below shows the interpolation strategy: for R/T < 1 the linear interpolation is done between 0 and 0.27324, for 1 < R/T < 4 the K-Factor is linearly interpolated between 0.27324 and 0.5 and if R/T is greater than 4, the K-Factor is constant and equals 0.5. Physically it means that if the bend radius is much larger than the thickness (at least 4 times), the material stretch is neglected.



The only K-factor parameter you can control in the browser is for R/T. That means if you set the value to 0.6, the interpolation law leads to the following result:

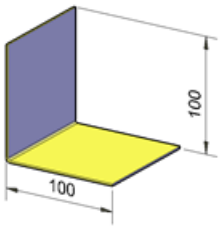


Pic: the varying K-factor in the browser leads to the following effective curve.

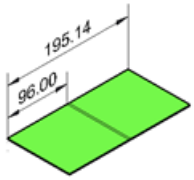
The distinguishing feature of the curve is that varying R in range of (0, T], L1 + L2 + BA gives the same value. This reflects the fact that for radii less than the thickness they relate to are for design convenience rather than for some physical process modeling, since during bending tool radius is usually greater or equal to thickness.

Let us review the example with varying K-factor by control curve setting (the bend radius is kept constant).

T = 2, R = 2, B1 = 100, B2 = 100



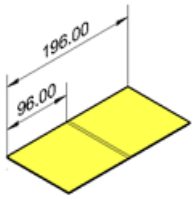
Setting K=0 gives L1=L2=96, Unfolded Length = 195.14, thus BA is 3.14.



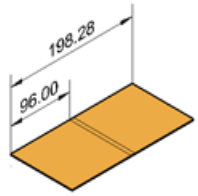
This value fits the formula exactly: a straight angle in radians roughly equals 1.57 (PI/2). The default K-factor 0.27324 gives an Unfolded Length = 196.

Or: L1=L2=96 (flanges are not deformed), the BA = 4 as we can compute from the dimensions (196 - 2*96) or from the Bend Allowance formula:

BA = BendAngle * (R + kT) = 1.57 * (2 + 2*0.27324) = 4



Finally, maximizing K-factor results in an Unfolded Length = 198.28. This is the maximal value one can achieve in this example since the neutral surface is taken from the external side of the bend.



The Bend Deduction (BD) for k = 0.27324:

BD = B1 + B2 - Unfolded Length = 100 + 100 - 196 = 4

21.5.8 Bend Table

21.5.8.1 Overview

Bend tables are a more reliable way to express material deformation properties than K-factor. When you physically bend a flat sheet of a particular material, you can measure its length before and after this process. You can repeat this procedure for different bend angles, bend radius, and sheet thickness, and save the measurements in a bend table. Then these measurements can be considered to compute the correct unfolded length for all parts made of the same material.

In a bend table, you can add the measurements for as many different bend angles as you want. However, in most cases it suffices to measure the length for a 90 degrees angle. BricsCAD automatically computes the length for other bend angles using a reliable interpolation technique.

Bend tables are stored in a *.CSV (comma separated values) file format. You can create a bend table in an ASCII text editor, such as Notepad. However, a more reliable and easier way is to create a table in a spreadsheet, such as Microsoft Excel or LibreOffice, and then export it to .CSV file format.

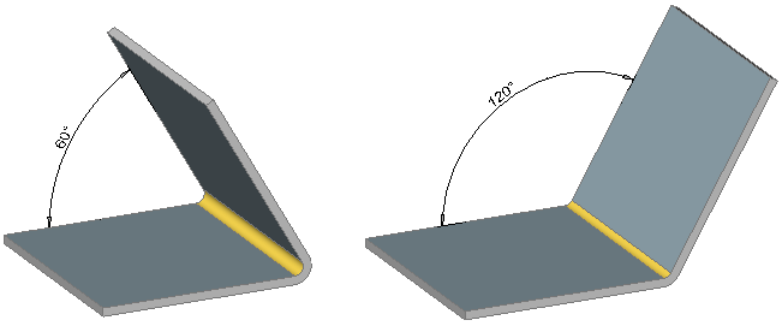
Bend tables in BricsCAD have the following structure:

	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

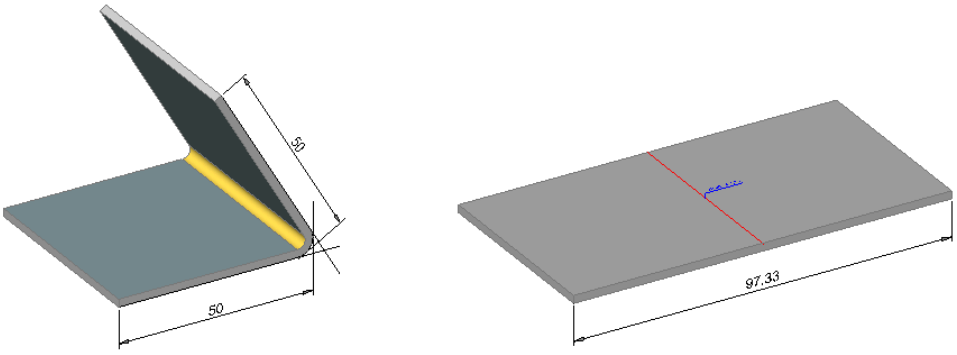
Rysunek 124: Example of a bend table

The following rules apply:

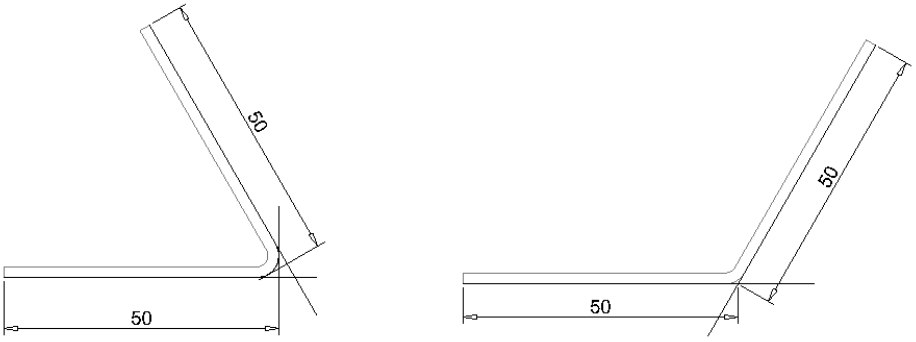
- The content of the first cell reads "BricsCAD”.
- **Version**: currently supported version is 1.
- **AngleType**: currently only internal bend angles are supported:



- **LengthType**: the semantics of the value in the cells of the bend table corresponding to a particular bend angle and bend radius. Now only bend deduction measured from the tangent point is supported (”BendDeductionTangent”). Bend deduction (BD) is the difference between the sum of the lengths of 2 flanges measured to the tangent point on a 3D model and the length of the same fragment in the unfolded state:



In depth: methods to compute tangential bend deduction for acute and obtuse angles.



- Within the defined header, several tables can exist. For each table, the first key is the **Thickness** of the sheet metal part. The second key, **DieWidth**, is optional. The **Thickness**/**DieWidth** pairs must be unique. If the **DieWidth** is not defined, only one unique **Thickness** value key is allowed, but multiple thickness values can be added. When a bend table is used to control the unfolding process, the **Thickness** and **DieWidth**, defined in the sheet metal part, must exist in the bend table. If the table is not found, the default K-factor policy is used.
- Once the bend table is found in the .CSV, it is used to unfold the sheet metal part. Using the sample bend table above, let us check a few cases:

  T = Thickness, R = Internal Bend Radius, A = Bend Angle
  - T = 1, R = 2, A = 45. Then after unfolding it will obtain the BD = 0.466, because both R and A are found in the bend table for the given thickness.
  - T = 1, R = 2.5, A = 45. A = 45 is found, but R = 2.5 is missing: the interpolation of BD between R=2 and R=3 is used.
  - T = 1, R = 2.5, A = 75. Neither R nor A is found in the table: the following cells are used to calculate the value of BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) and (R=3, A=90).

  The algorithm is used to calculate the interpolation. For example, linear interpolation on adjacent BD values render unnatural results, which does not match the real bending results. Also keep in mind that for a given T, R and A not all BD values make sense: a random one might place the neutral surface outside the sheet metal part, or expressed in K-factor (which can be deduced from BD), the K-factor can be out of range [0, 1]. In this case, the BD for the default K-factor is used.
- If you produce your sheet metal parts with different tools, you can describe them in one bend table using the **DieWidth** parameter. A sample bend table is provided for your convenience in the **[BricsCAD Installation folder]\Samples\Mechanical\bend_tables**.

  Uwaga: Writing rights to this folder might be limited. In such case you need to copy the bend tables to a different folder in which you have full access rights.

21.5.8.2 To set the bend table for your sheet metal part

1 Select the root node in the Mechanical Browser.

2 Select the **Bend table** field. 

3 Click the **Browse** button ( ).

4 Select the *.CSV file.

5 Double-click or press the **Open** button.

21.5.8.3 To update the bend table set previously

1 Select the root node in the Mechanical Browser.

2 Select the **Bend table** field.

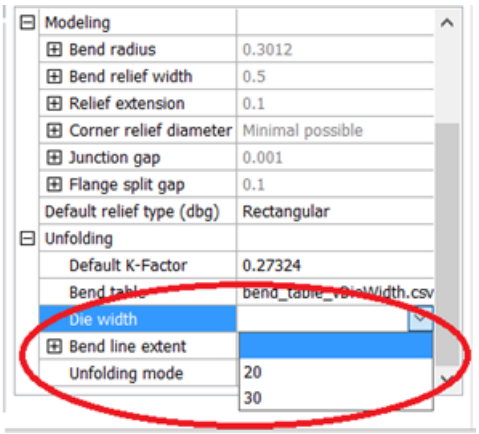
3 Click the **Update** button ( ).

If the bend table is not assigned to the document or has no **DieWidth** secondary key, the **DieWidth** parameter can not be assigned to a single bend or all the bends. In order to make the assignment, perform the following steps:

	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	

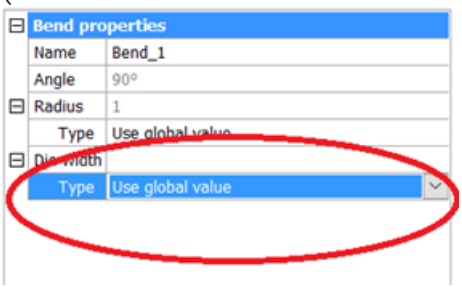
1 Select .CSV file **DieWidth** secondary keys and thickness matching the current part. On given example we assume that part has thickness of 1 mm and bend radius also equals to 1. We will vary bend deduction to different tools: the first subtable is when the tool is not specified, second subtable is for the case when tool **DieWidth** is equal to 20, and the last one is for a tool with **DieWidth** equal to 30.

2 Assign the bend table to the sheet metal part.

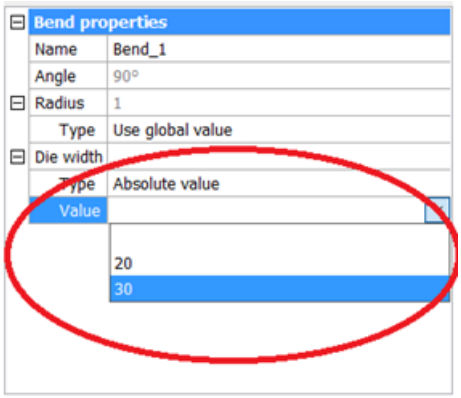


3 Check that in the Mechanical Browser new fields appear:

In this global setting we can select any default **DieWidth** value suitable for the part thickness. The value is assigned by default to the bends in the part, which can be checked by the property of the bend (click on the bend in the Mechanical Browser):



4 However, we can select a specific tool which will be used for the given bend by expanding the drop-down list:



5 With the given selection, an unfolding of Bend_1 uses a third subtable from our bend table for bend deduction and uses the value 1.8. For the rest of the bends, unless **DieWidth** is set for them, the default no-die-width value will be used, which will trigger first the bend table and uses value 2.1 for 90-degrees with a bend radius equal to 1.

Uwaga: After assigning the **DieWidth** to the bends and saving the document, the value is saved to the bends. However, next time the drawing is opened, in order to assign the **DieWidth** value to another bend, you have to reload the bend table first (press the green arrow button), see To update the bend table set previously. After the reload, the values from the bend table will be available.

Uwaga: Starting from BricsCAD V21 you can alter bend tables for different solids in model space by defining a local Sheet Metal Context per each solid.

21.5.9 Ustawienia konstrukcji blachowej

21.5.9.1 Przegląd

Sekcja Arkusz blachy w oknie dialogowym *Ustawienia* kontroluje różne właściwości arkusza blachy w BricsCAD. Właściwości są podzielone na cztery grupy: **Polecenia** , **Kolory funkcji** , **Parametry** początkowe i **Modelowanie** .



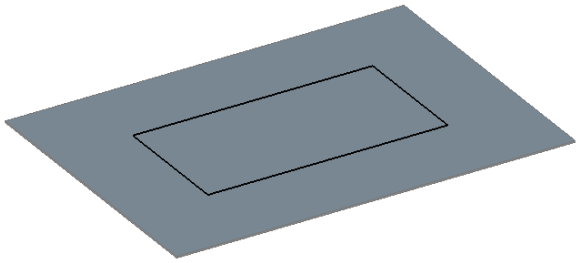
Rysunek 125: Ustawienia: Arkusz blachy

Uwaga: Większość ustawień arkusza blachy jest przechowywana w rysunku. Aby uzyskać najlepsze wyniki, zawsze zaczynaj od jednego z plików szablonów konstrukcji blachowych DWT lub utwórz własny.

21.5.9.2 Polecenia

Sekcja **Polecenia** zawiera ustawienia, które sterują zachowaniem różnych poleceń arkusza blachy.

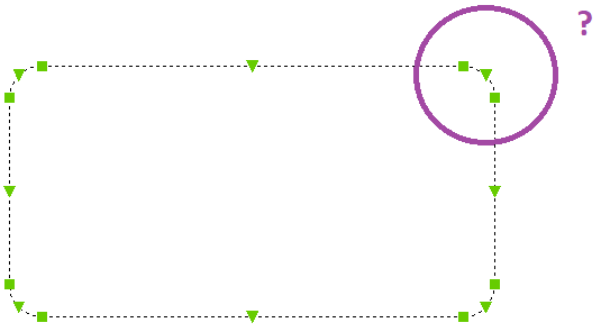
Uwaga: Przed BricsCAD V19 te kontrolki były dostępne jako opcje w poszczególnych poleceniach. Zostały przeniesione do okna dialogowego ustawień, aby uprościć interfejs poleceń.



Rysunek 126: Ustawienia: Polecenia

Warstwy używane do rozwijania arkusza blachy określają wygląd rozwiniętych części.

21.5.9.3 Kolory Cechy



Rysunek 127: Ustawienia: Kolory funkcji

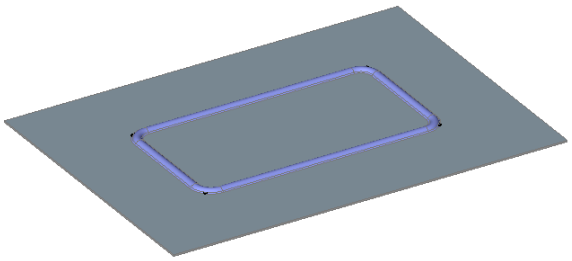
Ustawienia kolorów elementów wizualnie rozróżniają różne typy elementów konstrukcji blachowych według koloru. Możesz dostosować kolory dla każdego typu funkcji.

21.5.9.4 Początkowe parametry

Te parametry są odpowiedzialne za inicjalizację **kontekstu arkusza blachy** skojarzonego z rysunkiem.

Modelowanie

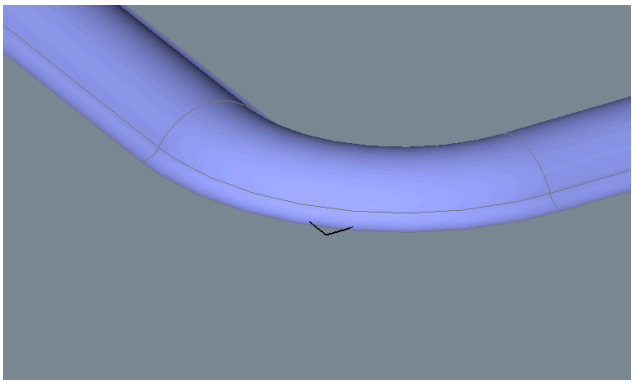
Ustawienia modelowania sterują zachowaniem elementów konstrukcji blachowej. Ustawienia są pogrupowane według typu elementu.



Rysunek 128: Ustawienia: Modelowanie

Rozwijanie

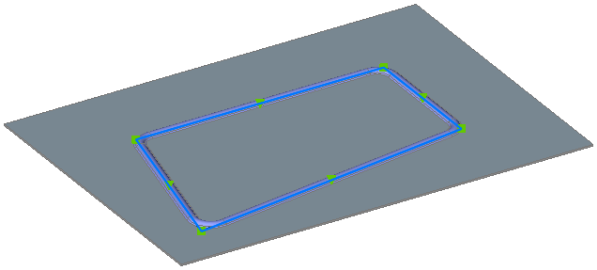
Ustawienia rozwijania kontrolują zachowanie **Wartość współczynnika K** , **Tryb rozwijania funkcji formularza** , oraz **Typ/wartość rozciągłości linii zgięcia** .



Rysunek 129: Ustawienia: Rozkładanie

21.5.9.5 Modelowanie

Ustawienie Modelowanie obejmuje preferencję użytkownika *SmSmartFeatures*, która steruje lokalną, inteligentną aktualizacją geometrii po wykonaniu poleceń arkusza blachy.



Rysunek 130: Ustawienia: Modelowanie

Samouczek: Parametryzacja części arkusza blachy

21.5.10 Creating Flanges

21.5.10.1 Overview

A flange is the main feature of most sheet metal parts. In BricsCAD you can create flanges using many different techniques:

- Create a base flange from a closed 2D profile, which defines its boundary.
- Create one or several edge flanges by pulling edges of existing flanges.
- Create several flanges at once by extruding a 2D polyline (every linear edge of the polyline becomes a flange).

Wskazówka: There are also other techniques you can use to create flanges. These are described in:

- Flanges can be created as elements of lofted sheet metal parts.
- Flanges can be recognized in existing geometry.

In contrast to history-based MCAD, BricsCAD applies the same set of tools to modify any flange - independently.

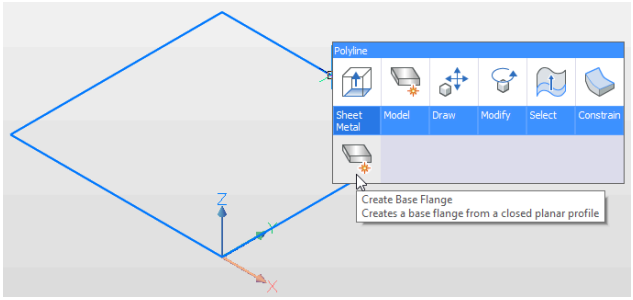
21.5.10.2 Create a Base Flange

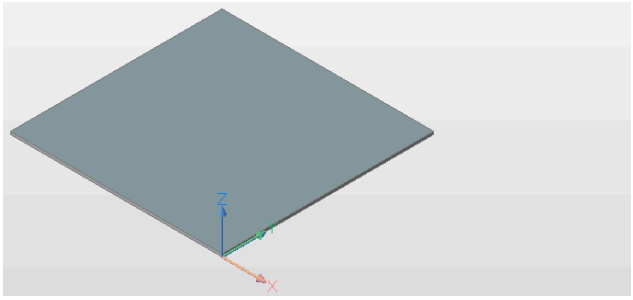
In general, you create a base flange to start your sheet metal part design. To create a base flange, select a closed planar profile and run the *SMFLANGEBASE* command.

Do one of the following:

- Click the **Create Base Flange** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Create Base Flange** in the **Sheet Metal** menu.
- Choose **Create Base Flange** in the **Sheet Metal** group of the **Quad** cursor menu:

Create a Base Flange





To create a base flange the *SMFLANGEBASE* command extrudes a profile.

The height of the base flange is the default **Thickness** property of the sheet metal part. To change the thickness of your sheet metal part, type the appropriate value in the **Thickness** field of the Mechanical Browser.

21.5.10.3 Create Edge Flanges

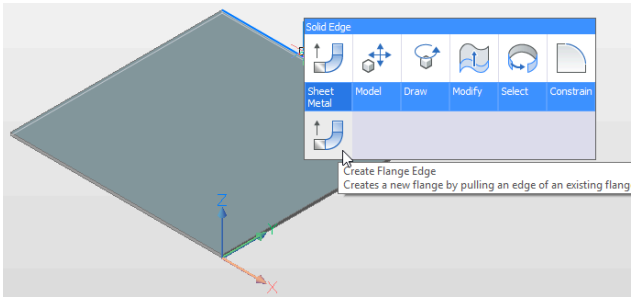
You can add an additional flange to your sheet metal:

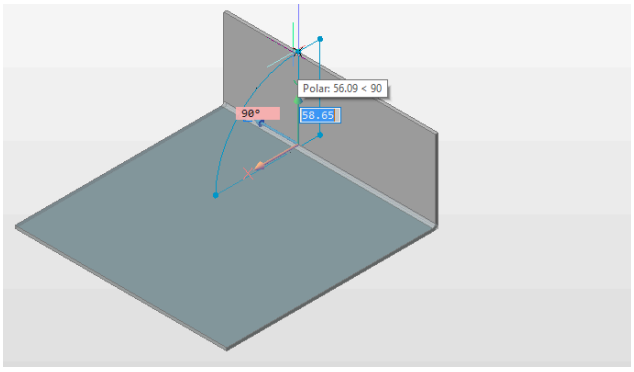
1 Select a linear edge.

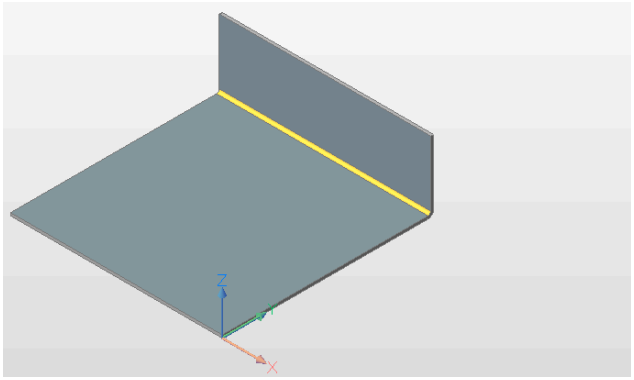
2 Select the *SMFLANGEEDGE* command in the **Sheet Metal** section of the **Quad** cursor menu.

3 Move your mouse pointer to define the desired length of the edge flange and the angle between the 2 flanges. You can also use the dynamic dimensions fields to enter the required values.

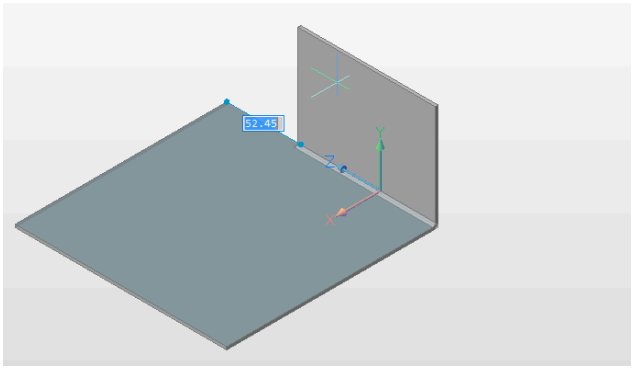
Create Flange Edge

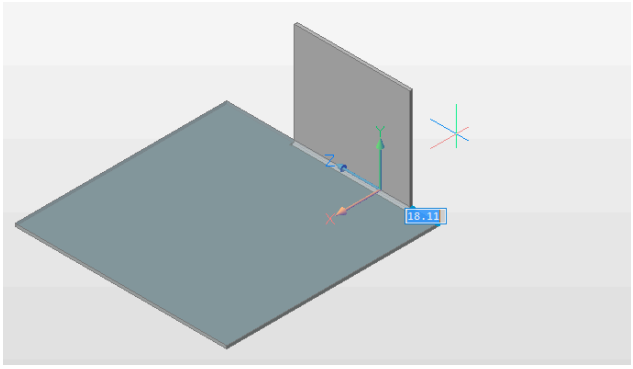


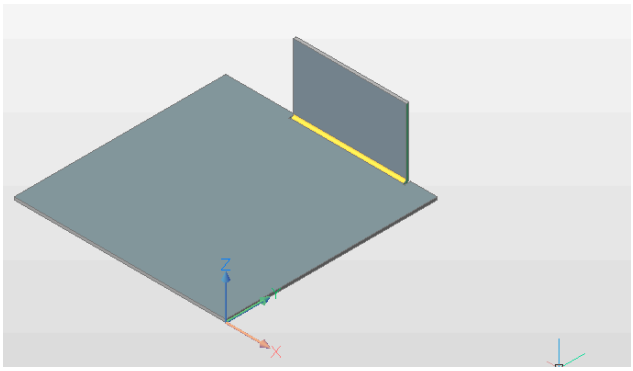




To create an edge flange where the width is different to the width of the edge, select the **Width** option of the *SMFLANGEEDGE* command and define the offset distance from one or both sides of the edge.



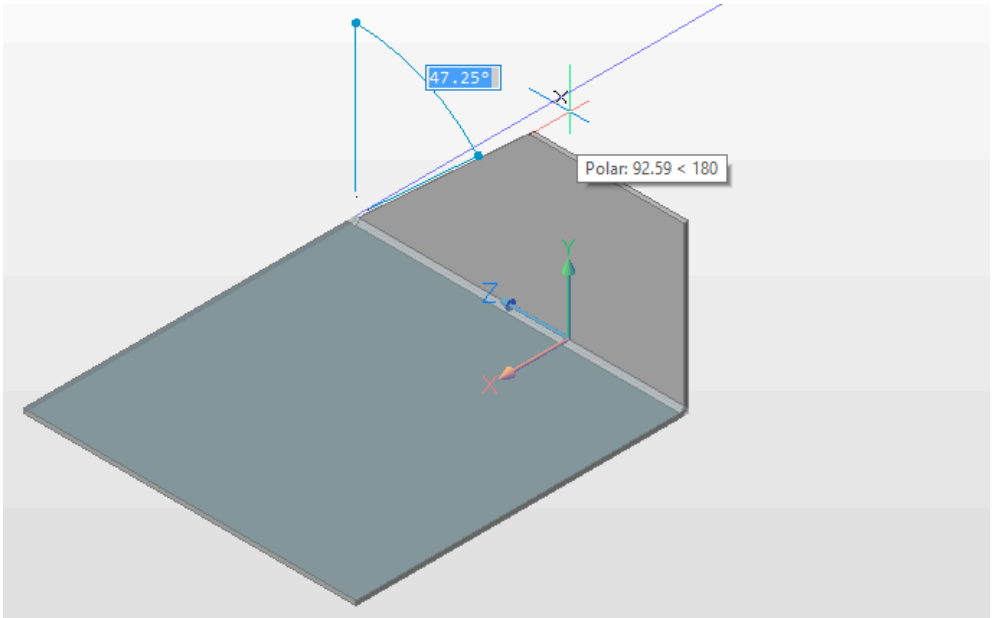


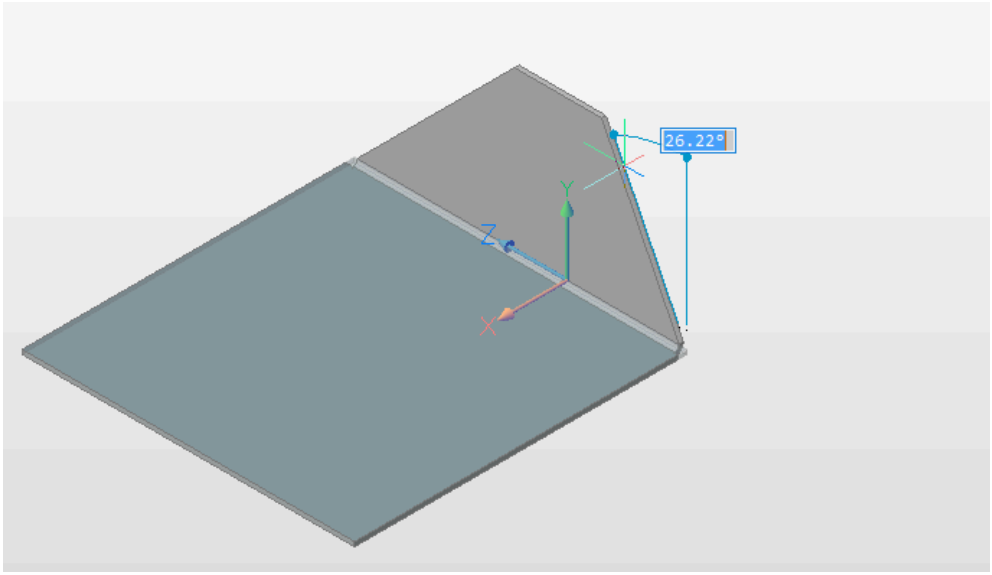


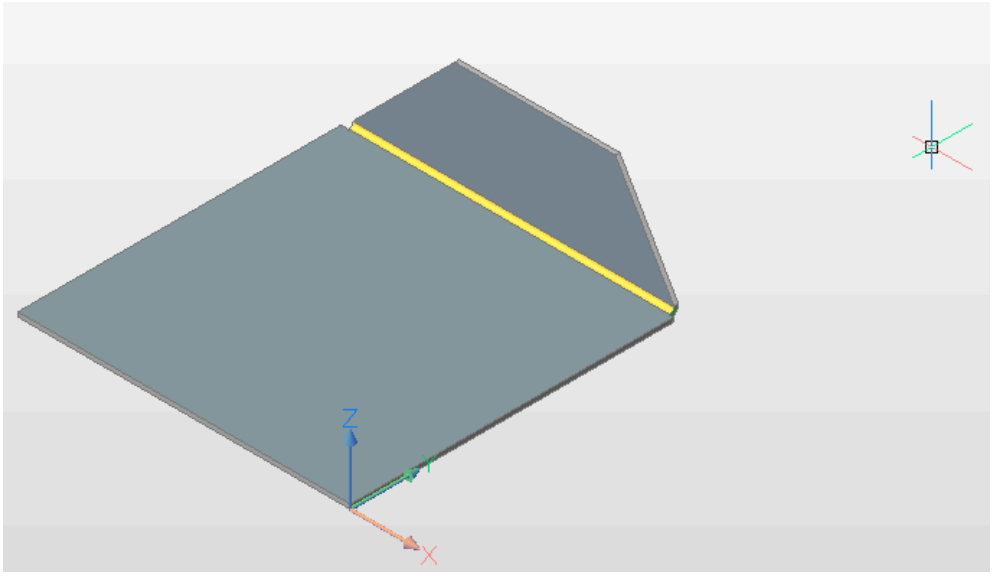
Uwaga: When you create an edge flange of a different width to the edge, bend reliefs are created automatically.

21.5.10.4 Create Edge Flange with Taper Angle(s)

You can create trapezoidal edge flanges with the **Taper angle** option of the *SMFLANGEEDGE* command:

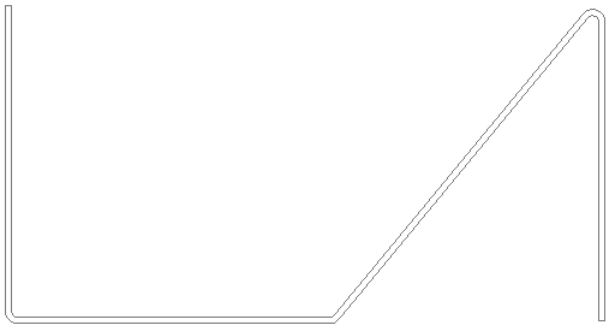






21.5.10.5 Create Multiple Flanges by Extruding a Polyline

There is a simple but important class of sheet metal parts which are seen as a thickened polyline if we view on the part from the "front" side:

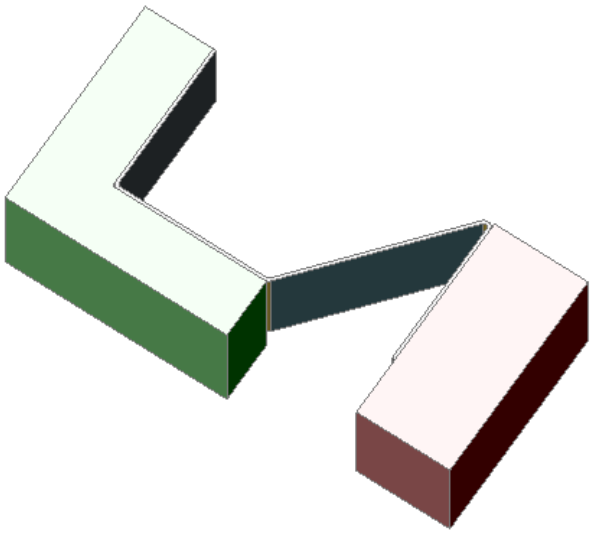


To create this part you can:

- use sequential calls of the *SMFLANGEEDGE* command, or
- draw a polyline and use the *DMEXTRUDE* command, followed by the *DMTHICKEN*, *SMCONVERT*, and *SMBENDCREATE* commands.

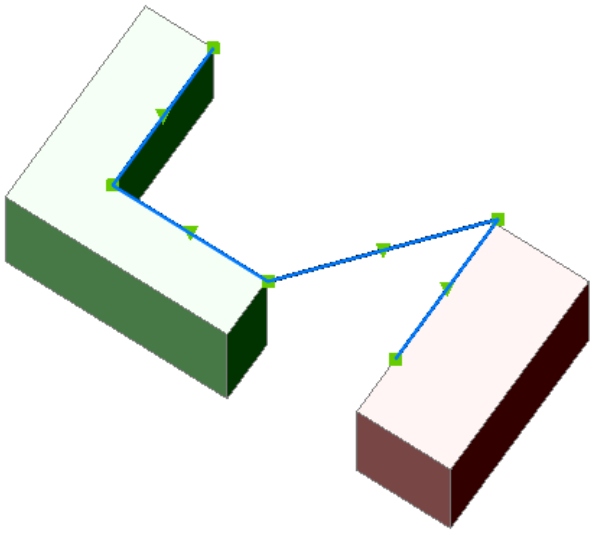
However, we recommend the following procedure:

BricsCAD allows you to draw a part in a complex assembly with other parts so that it does not interfere with other entities in the assembly.



You can control the thickening direction of every line segment individually with *SMEXTRUDE*.

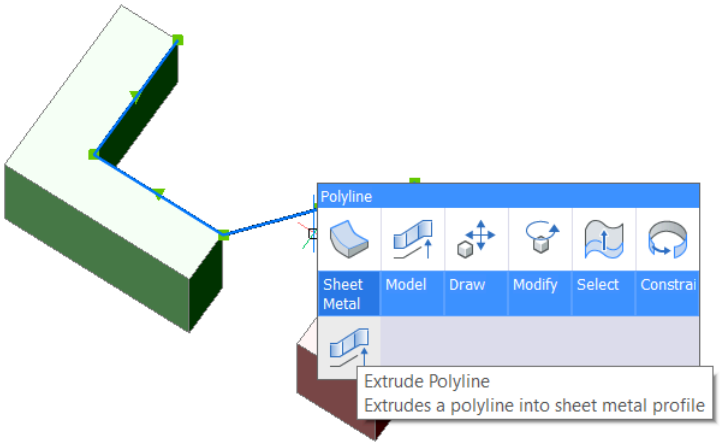
1 Draw a polyline that represents a sketch of the future sheet metal



part:

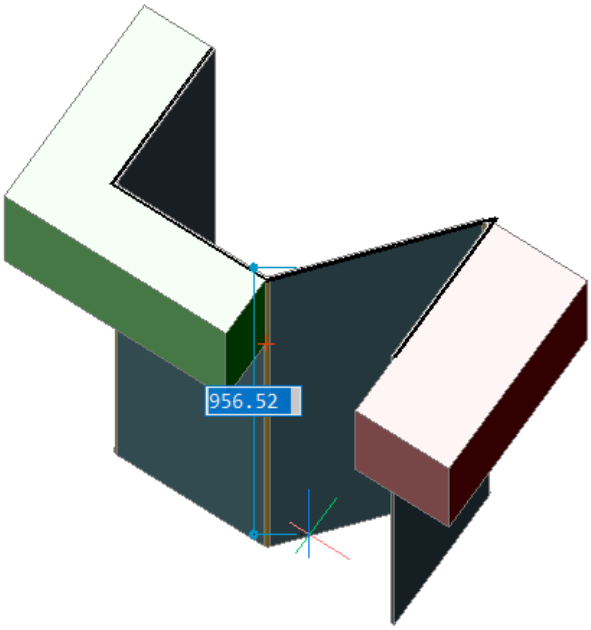
Do one of the following:

- Click the **Extrude Polyline** tool button ( ) on the **Sheet Metal** toolbar.
- Choose **Extrude Polyline** in the **Sheet Metal** menu.
- Choose **Extrude Polyline** in the **Sheet Metal** group of the **Quad** cursor



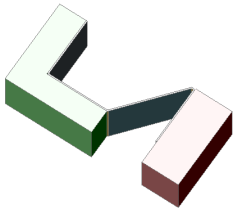
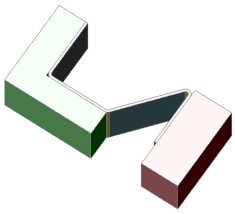
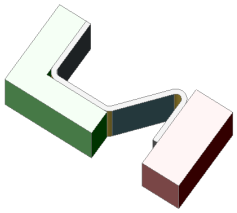
menu:

2 Hover your cursor to select **extrusion height** or enter the value and press **Enter** to



accept:

The body is created and the extensions are respected. The sheet metal part does not exceed the dimensions of the original polyline. The behavior is the same, even with different thicknesses:

Thickness 1 mm	Thickness 2 mm	Thickness 4 mm
		

Uwaga: The policy of preserving external dimensions also works for sheet metal parts created by other workflows: sequential flange building with *SMFLANGEEDGE*, part bends with *SMFLANGEBEND* and sheet metal design recognition with *SMCONVERT*. If you want to change the extrusion direction for a flange, apply the *SMFLIP* command to it.

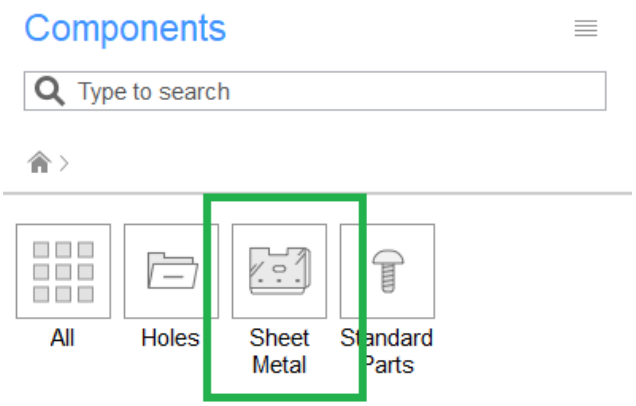
21.5.10.6 Extrude a Contour to Create a Flange

The *SMFLANGECONTOUR* command allows you to extrude a given contour to a new flange and attach it to the model. It can be thought of as a sequence of the *SMFLANGEBASE*, *SMFLANGECONNECT*, and *SMBENDCREATE* commands. In reality, it intelligently chooses the thickness faces on selected flanges and connects them with the thickness faces on a new flange.

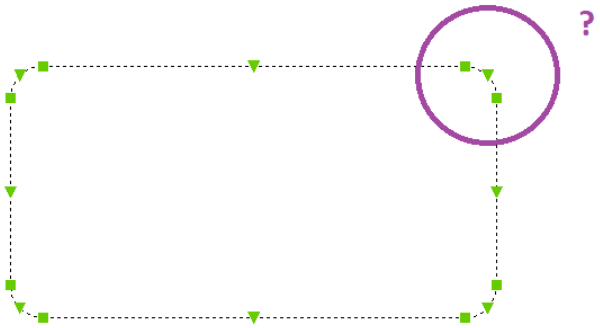
Do one of the following:

- Click the **Create Flange from closed Contour** tool button ( ) on the **Sheet Metal** toolbar.

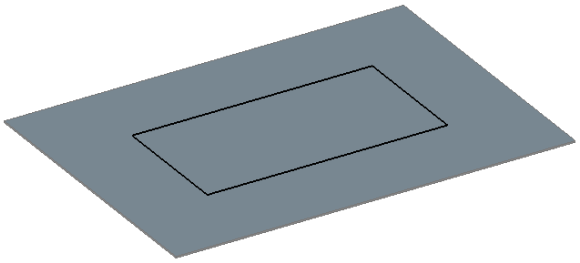
- Choose **Create Flange from Contour** in the **Sheet Metal** menu.
- Choose **Create Flange from Contour** in the **Sheet Metal** group of the **Quad** cursor menu:



Once all the required flanges are selected, click on the contour:

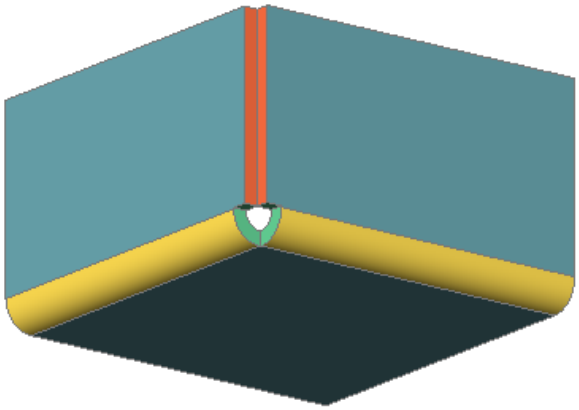


The command creates a new flange and merges it with the selected flanges on the part:



21.5.10.7 Bends, Corner Reliefs and Junctions

When you pull an edge flange with the *SMFLANGEEDGE* command, a bend feature between the edge flange and the base flange is automatically created. A corner relief feature is automatically created in a corner where 3 flanges meet together. This operation also creates a junction between 2 flanges, which are not connected with a bend.

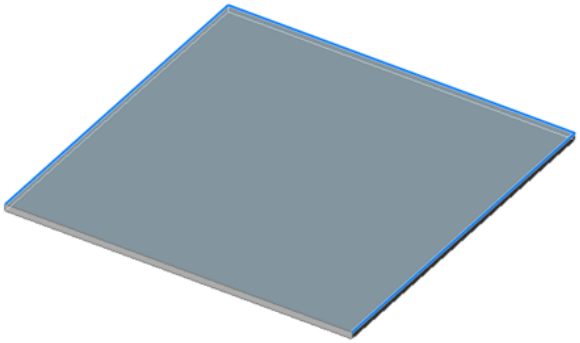


21.5.10.8 Create Multiple Flanges

You can create several flanges simultaneously.

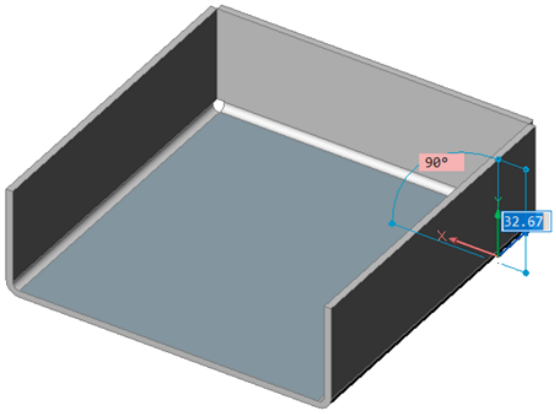
Do one of the following:

- Select multiple edges, then choose **Create Flange Edge** ( ) in the **Sheet Metal** command group of the **Quad** cursor menu.
- Launch the *SMFLANGEEDGE* command and select multiple edges.

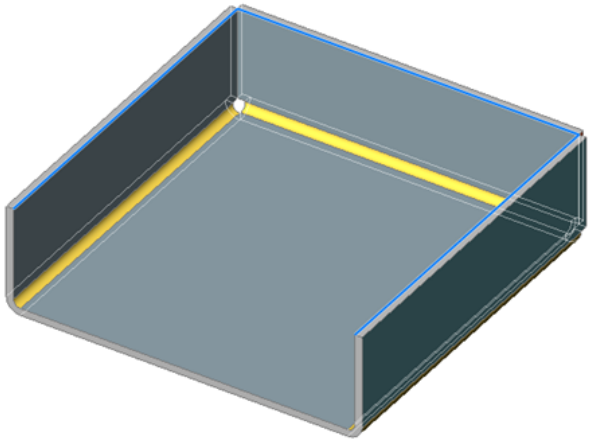


- Create the edges dynamically.

  Define the angle and the flange height using the dynamic entry fields. Press the **TAB** key to switch between the **Angle** and the **Distance** field.

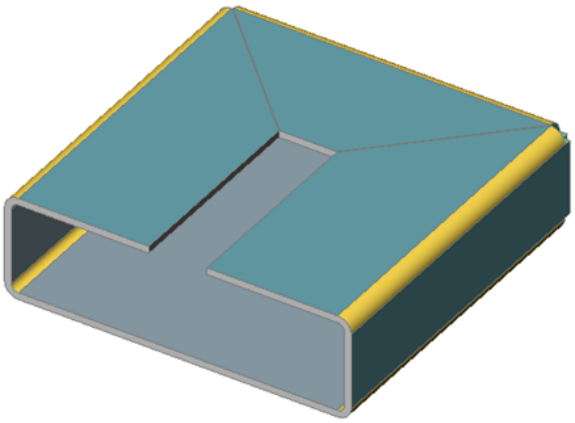


- Select the inside edges of the newly created flanges:

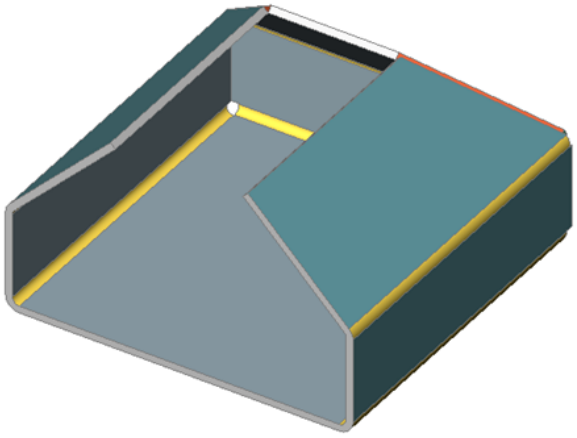


- Create the flanges.

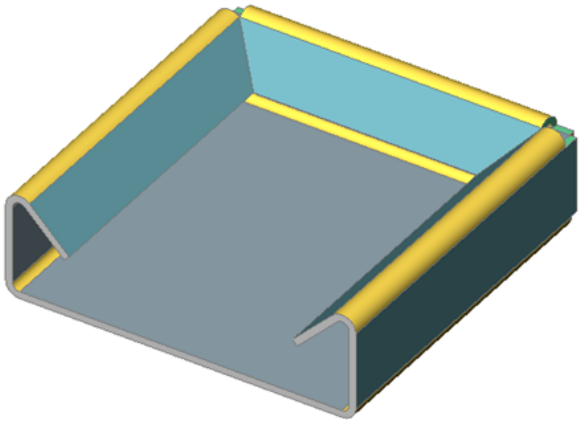
  The command automatically calculates the intersection of the flanges and creates a junction or miter features. At 90 degrees miters are created:



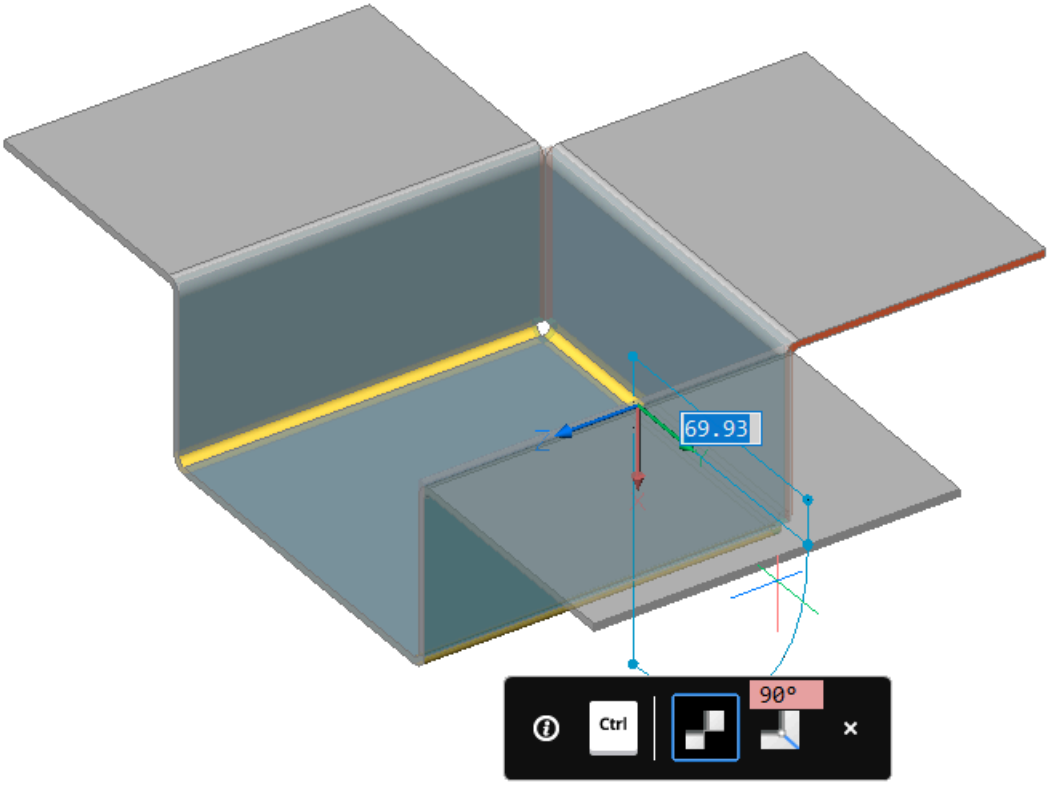
  For other angles junctions are created:

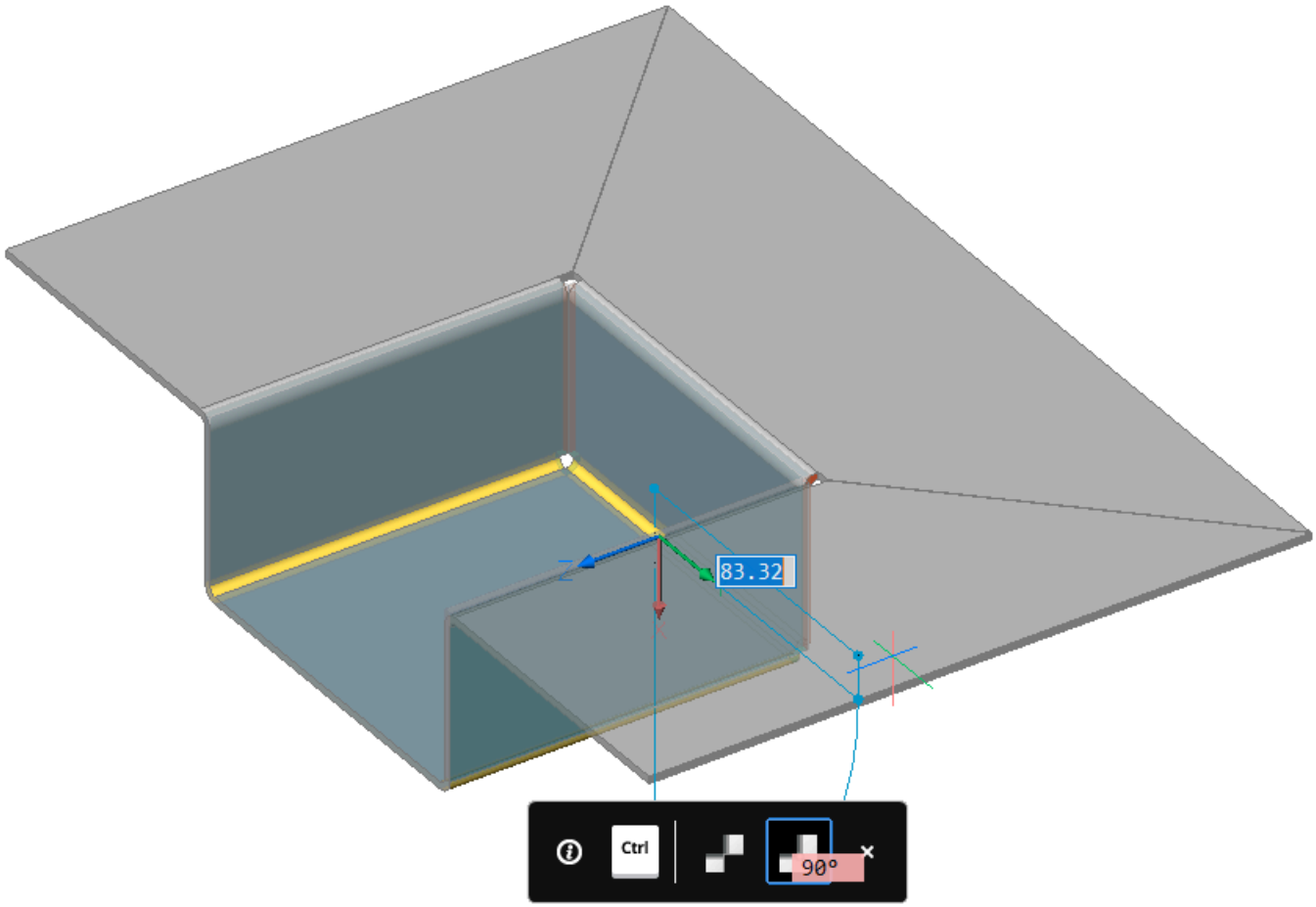


In all cases material conflicts are avoided:

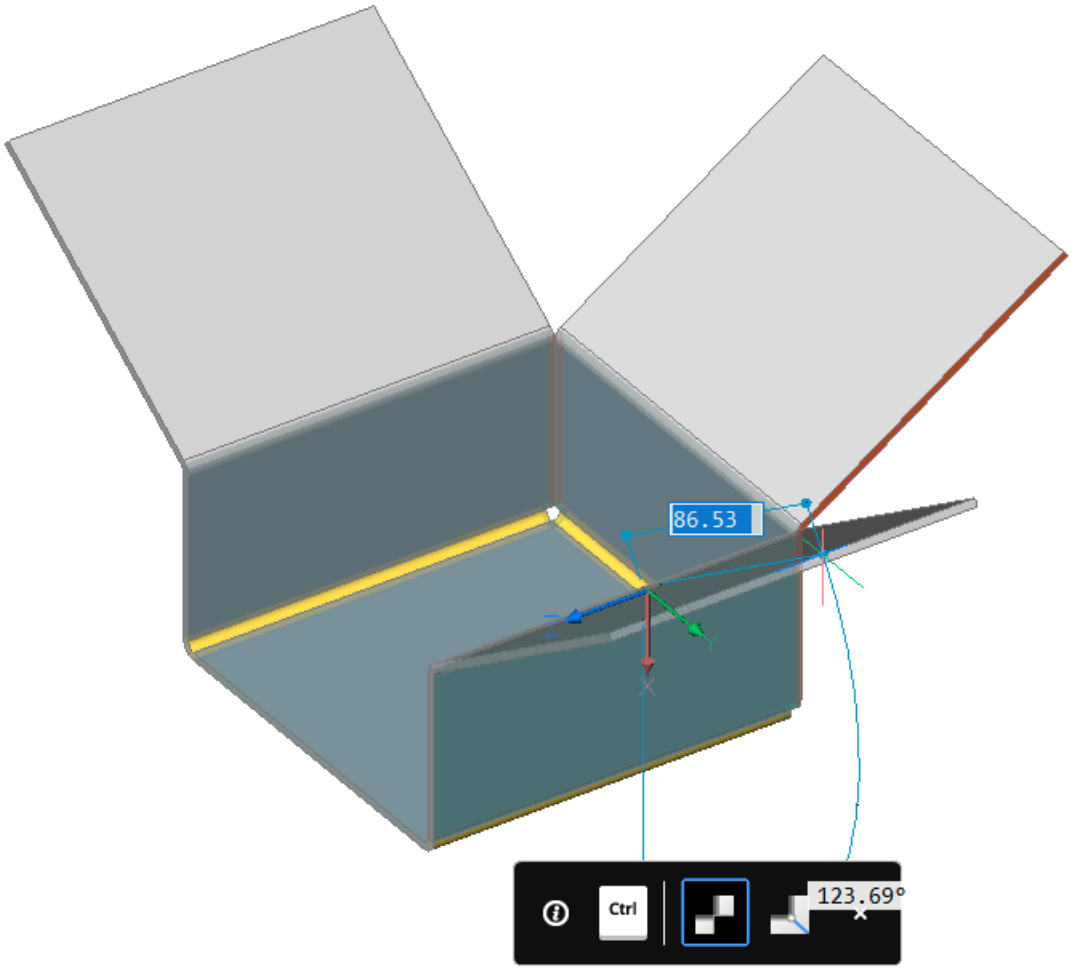


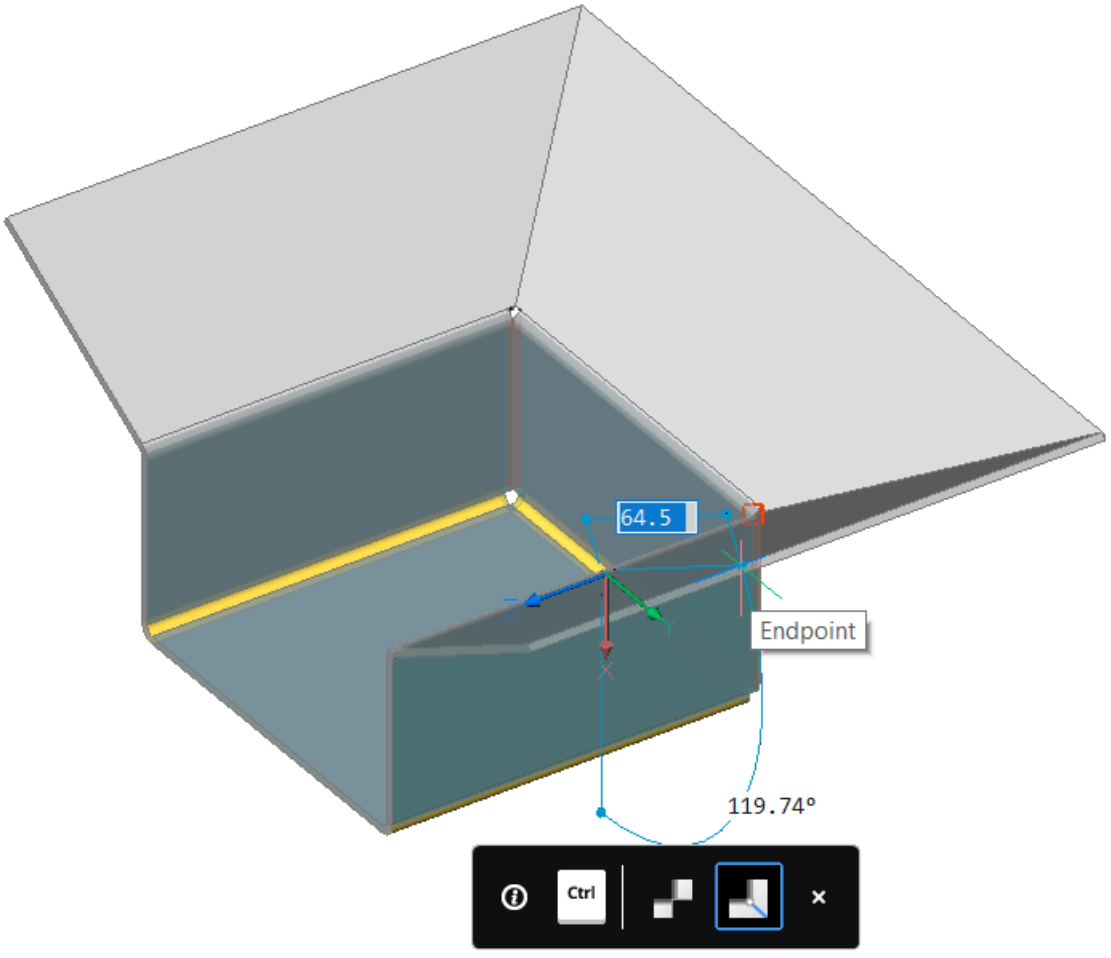
If the flanges are created externally, use the **Toggle** option (as well as **widget control**) to control the connection type for miters:


69.93
Ctrl
90°


83.32
Ctrl
90°

and junctions:


86.53
Ctrl
123.69°


64.5
Endpoint
119.74°
Ctrl

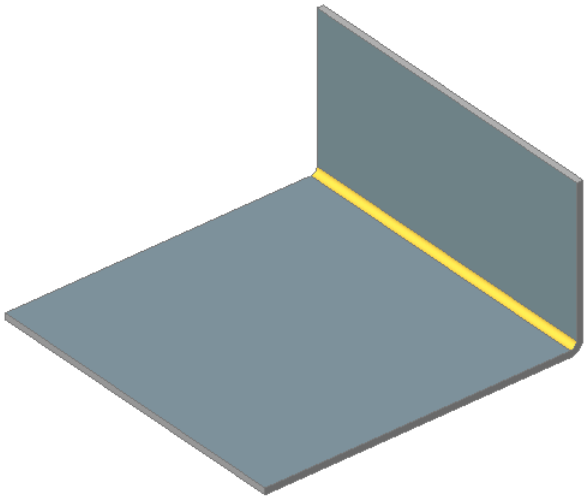
21.5.11 Modifying Flanges

BricsCAD provides several tools to modify flanges in sheet metal parts. These tools can be applied consistently to any part - independent of its design history. In this article we use Solid Modeling tools and specific sheet metal tools. Almost all solid modeling tools apply to sheet metal parts, while preserving the design intelligence provided by Sheet Metal Features.

21.5.11.1 Changing flange size and shape

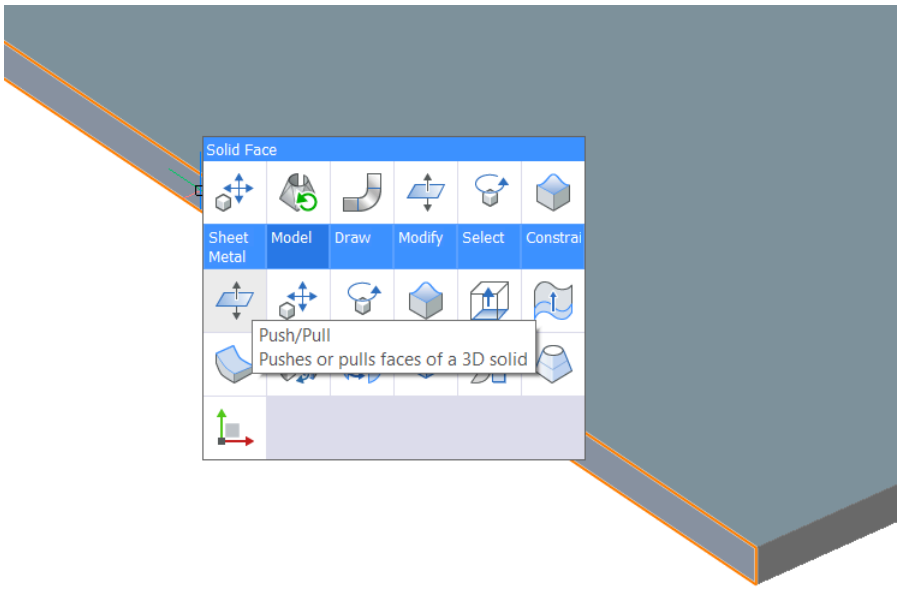
On a given simple model we will apply several Solid Modeling operations to modify the shape of its flanges.

Uwaga: We work on thickness faces of the model only

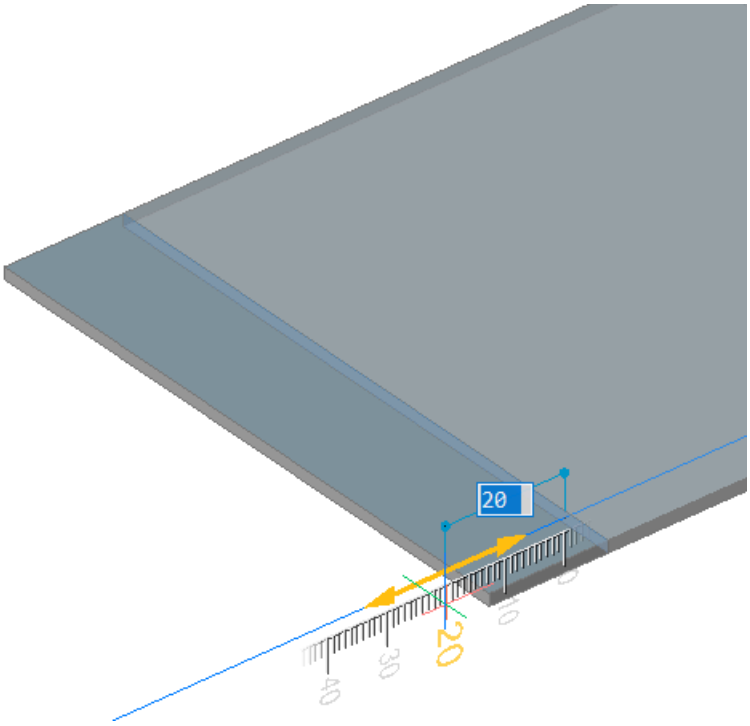


21.5.11.2 Stretching flanges

1 Hover over the thickness face, then select **PushPull** in the **Model** command group in the **Quad**



menu.

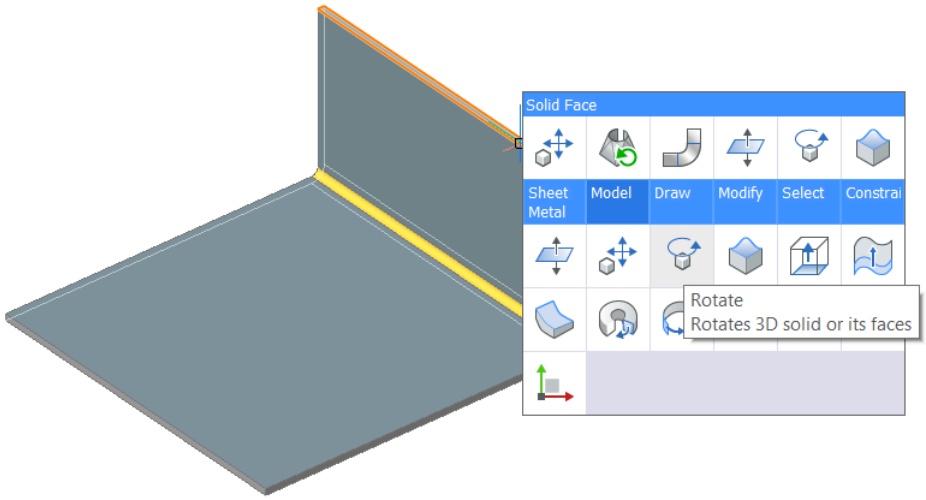


2 Drag the face, then right-click.

The changes will be applied to the part, the flange is stretched over the specified distance.

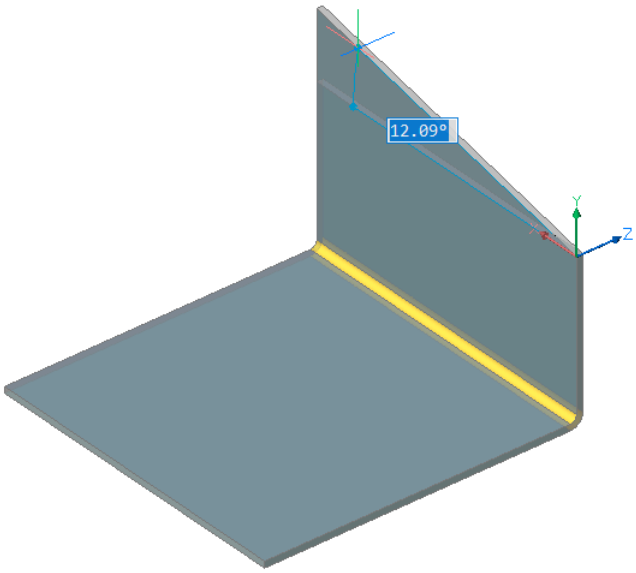
21.5.11.3 Rotating a thickness face

1 Hover over the face and choose **Rotate** in the **Model** command group in the **Quad**



menu.

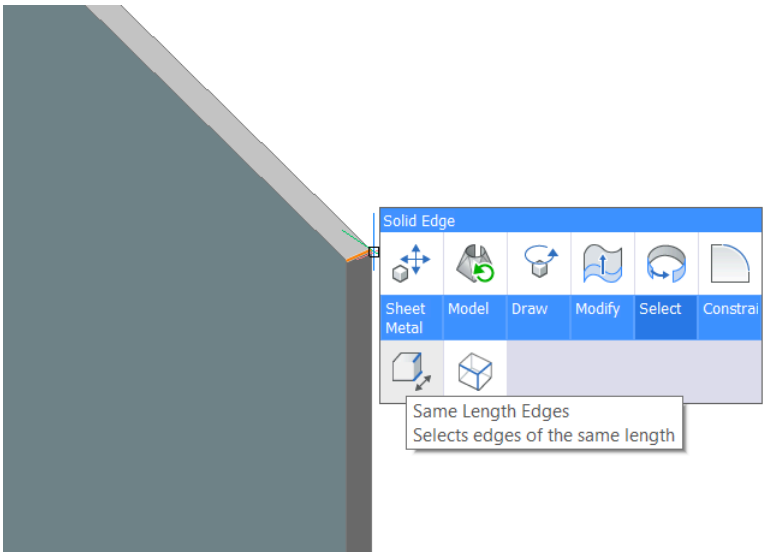
2 Specify the rotation axis, base point and angle (for more details, see DMROTATE in the Command



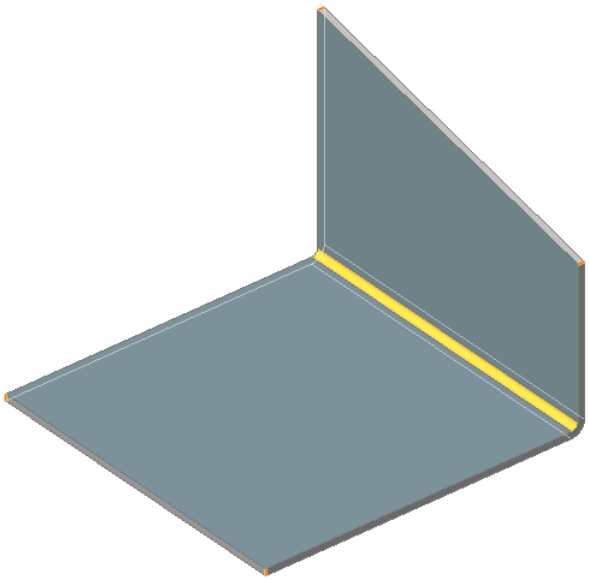
Reference):

3 Fillet the sharp edges.

Hover over a thickness edge and choose **Same Length Edges** ( ) in the **Select** command group in the **Quad** menu.



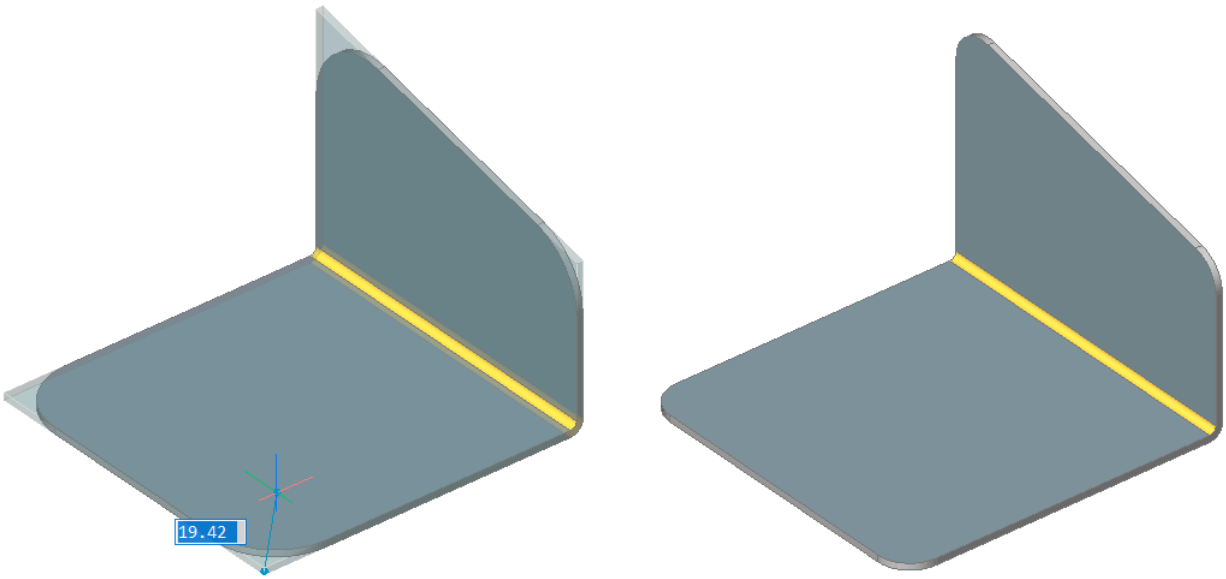
4 Press **Enter**. Edges with the same length are



selected.

5 Choose **Fillet** in the **Model** command group in the **Quad** menu.

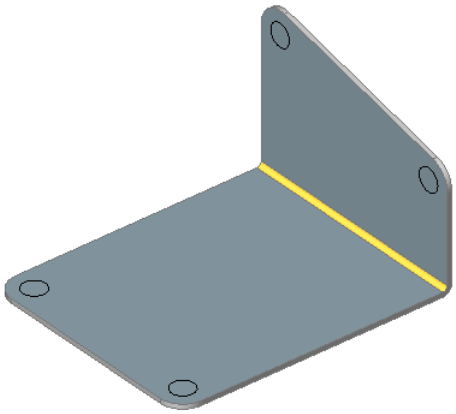
6 Specify the desired fillet



radius:

21.5.11.4 Making a hole in a flange

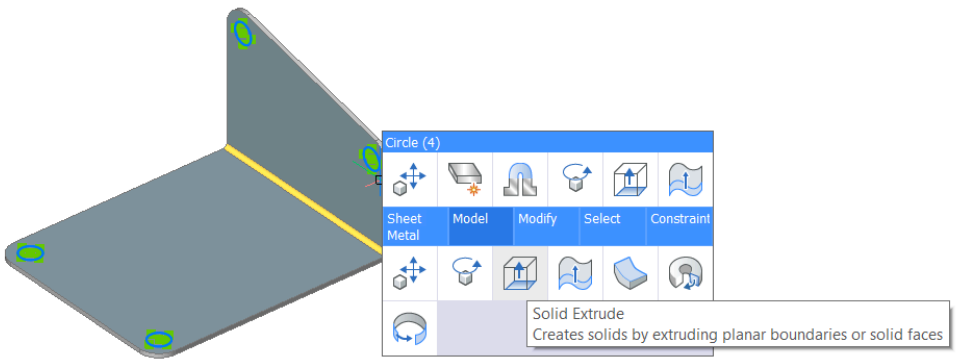
1 Create 4 circles as indicated in the image below. Use the Snap to Center entity snap to define the



center points.

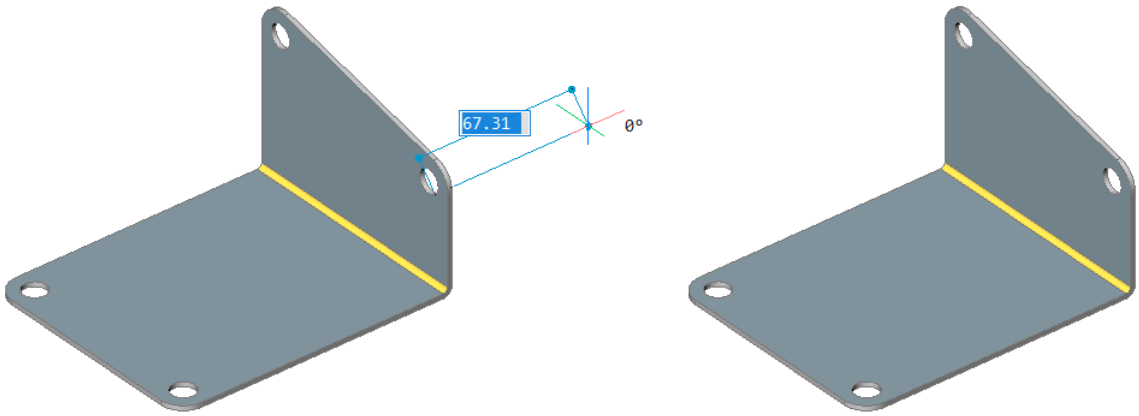
2 Select the circles.

3 Choose **Solid extrude** in the **Model** command group in the **Quad**



menu.

4 Move the cursor to create the

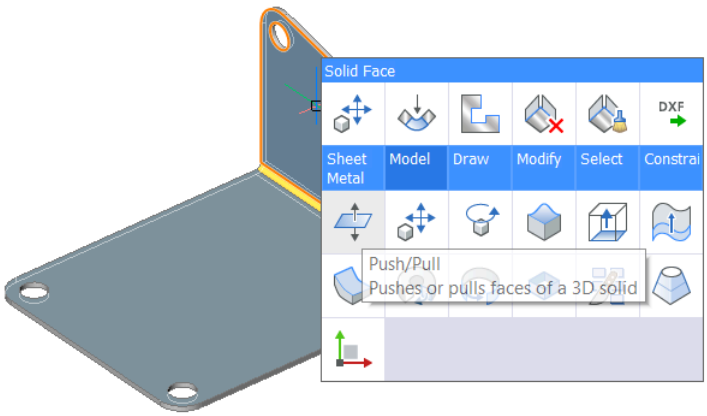


holes.

21.5.11.5 Moving a flange

1 Do one of the following:

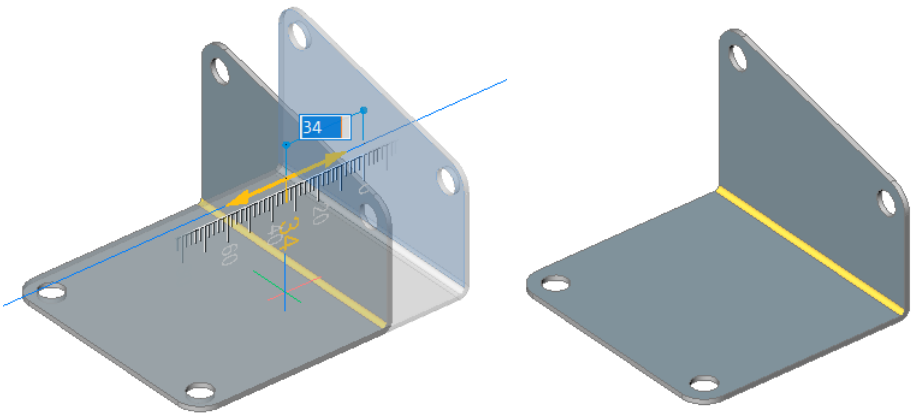
- Click the **Push/Pull** tool button ( ) on the **Direct Modeling** toolbar.
- Choose **Direct Modeling > Push/Pull** in the **Model** menu.
- Choose **Push/Pull** ( ) in the **Edit** panel of the **Sheet Metal** tab in the ribbon.
- Run the *DMPUSHPULL* command for given face.
- Hover over the face you want to move and choose **Push/Pull** in the **Model** command group in



the **Quad** menu.

2 Select new position of the flange.

Uwaga: Due to presence of **Sheet Metal Features**, part thickness is preserved.



21.5.11.6 Rotating a flange

Although it is possible to rotate a flange using the *DMROTATE* command (), it is recommended to use *SMFLANGEROTATE* () instead, because this command automatically selects the rotation axis to respect the design intent of the sheet metal part.

1 Launch the *SMFLANGEROTATE* () command.

You are prompted: Select a flange face to rotate:

Flange faces under the cursor highlight.

2 Click to select a flange face.

The flange rotates dynamically.

A dynamic dimension displays the current angle with respect to the base flange.

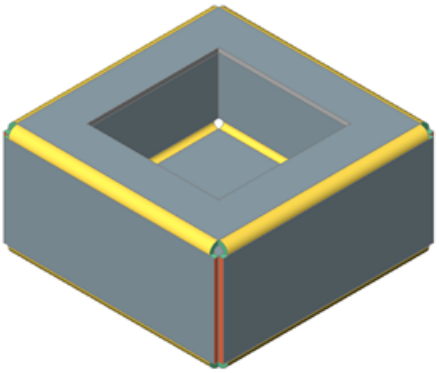
3 Do one of the following:

- Select a point.
- Type a value in the dynamic dimension field.
- Press the **TAB** key, then specify the absolute rotation angle in the dynamic dimension field.

Uwaga: When the *DMROTATE* command is used to rotate a flange, which is connected to other flanges by junction features, these features are automatically dissolved before the rotation.

21.5.11.7 Splitting a flange

In some cases flanges are better split to minimize the material consumption, for example, when a flange has a large hole, as shown in the image below:



1 Launch the *SMSPLIT* () command.

You are prompted: Select lines or edges to split the flange or [SMart split/draw a New line/selection options (?)] <draw a New line>:

2 Press Enter to accept the default option.

You are prompted: Start point of the line:

3 Hover over the face you want to split (make sure Dynamic UCS (DUCS) is ON).

The face highlights.

4 Specify the start point of the line.

You are prompted: End point of the line:

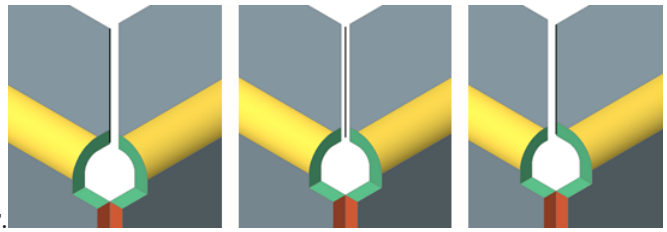
5 Specify the end point of the line:

You are prompted: Make split Center/Left/Right/<Accept model>:

6 To adjust the position of the cut with respect to the line, do one of the following:

- Hit the **CTRL** key to toggle the **Center**/**Left**/**Right** options.

  If **TIPS** is ON, the currently selected option is indicated in the Tips Widget.
- Select **Center** , **Left** or **Right** in the Prompt Menu.
- Type an option in the Command line, then press **Enter**.



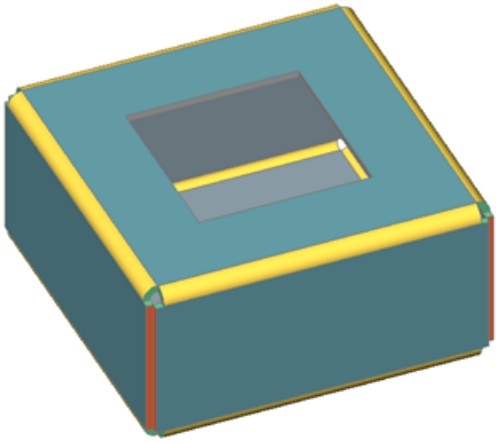
The flange is cut by the line: left, center or right.

Uwaga: A split operation creates a miter feature. You can restore the initial flange by removing the miter feature from the feature list in **Mechanical Browser**.

21.5.11.8 Smart tools to split a flange

The scenario below illustrates the other capabilities of the *SMSPLIT* command:

- Smart split by selecting a single vertex in the corner.
- Split off the bend and convert the part adjacent to the corner into a



junction.

1 Do one of the following:

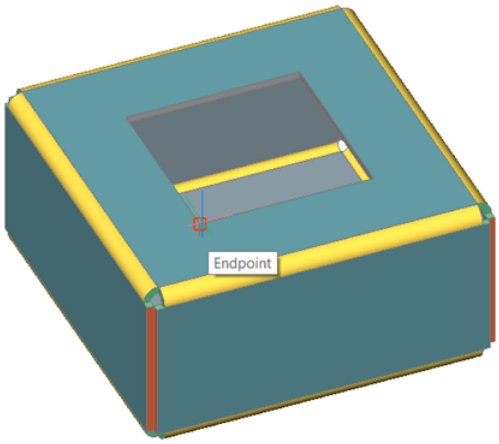
- Click the **Use Smart Split Flange** tool ( ) on the **Sheet Metal** ribbon tab.
- Choose **Use Smart Split Flange** in the **Sheet Metal** menu.
- Type *SMSPLIT* in the Command line.

  You are prompted: Select lines or edges to split the flange or [SMart split/draw a New line/ selection options (?)] <draw a New line>:

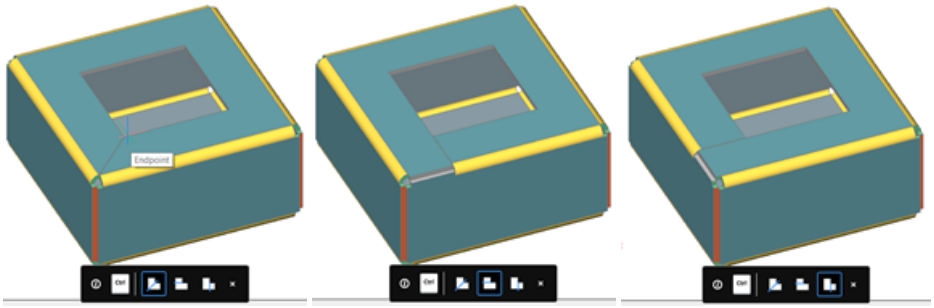
2 Choose the Smart Split option.

You are prompted: Select point on flange or lofted bend border for smart split or [Propagate] <Propagate>:

3 Select a corner vertex or snap to a point on an edge off the flange you want to split:



You are prompted: Make split [Center/Left/Right/Accept] <Accept>:

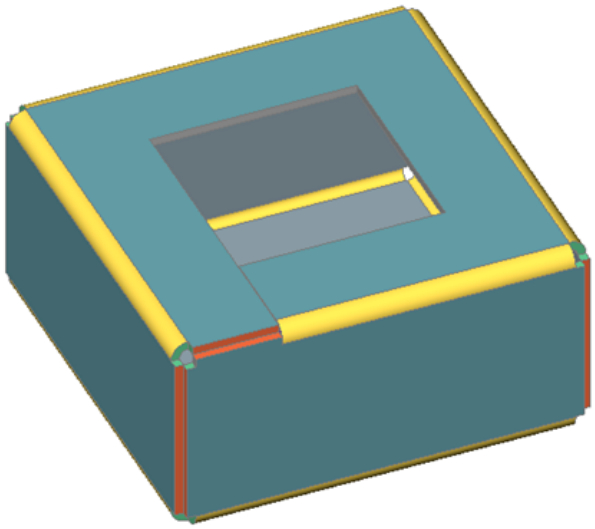


4 Do one of the following:

- Press Enter to accept the split.
- Repeatedly press the **CTRL** key.

  If **TIPS** is ON, the currently selected option is indicated in the Tips Widget.

  The flange is cut by a line through the specified point. The bend is partially converted into the junction:



In case of the **Center** option if **Smart Features** are turned on, corner reliefs adjacent to the split will be converted into bend reliefs.

Uwaga: The conversion of part of the bend into a junction is unrelated to the **Smart Split** mode.

21.5.11.9 Bending a flange

The *SMFLANGEBEND* command bends one or more flanges using a line. You can either draw the line first, then start the command, or use the **New Line** option of the command.

To bend multiple flanges simultaneously

1 Launch the *SMFLANGEBEND* command.

You are prompted: Select a flange face [selection options (?)]:

2 Select the flange you want to bend.

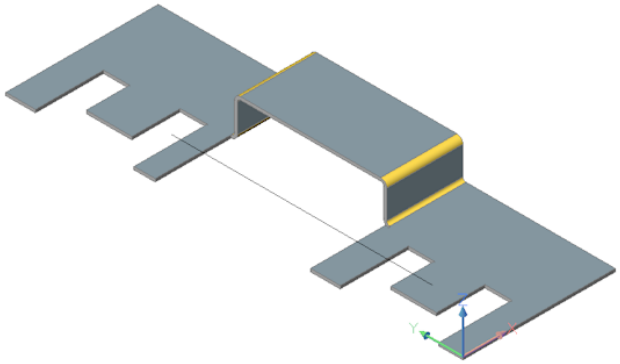
You are prompted: Select line or edge to bend the solid or draw a [New line] <New line>:

3 Do one of the following:

- Select a line that overlaps the flange(s) to be bent.
- Choose the **New Line** option.

  You are prompted to draw a line on top of the flange(s) to be bent.

1 You are prompted: Select flange position [Angle/Radius/Switch side/Toggle bend extents/ACcept] <Accept>:



Pick a point at the side of the line you want to move.

The selected part is bent dynamically.

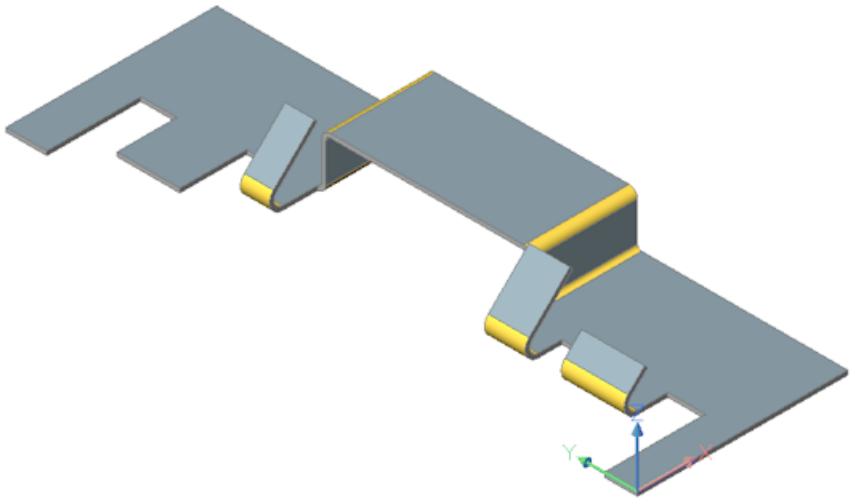
2 (option) Type a value to define the bend angle, using the default bend radius.

3 (option) Select the **Radius** option.

- You are prompted: Enter bend radius [Back] <Back>:

  Type a value to override the default bend radius and press **Enter**.
- You are prompted: Enter bend angle or set [Radius/Switch side]:

  Type a value to define the bend angle.



21.5.11.1 Connecting flanges

0

The *SMFLANGECONNECT* command closes gaps between 2 arbitrarily oriented flanges.

1 Launch the *SMFLANGECONNECT* command.

You are prompted: Select planar thickness faces of two flanges:

Flange faces under the cursor highlight.

2 Select the thickness face of the first flange.

You are prompted:

Entities/subentities in set: 1

Select planar thickness faces of two flanges:

3 Select the thickness face of the second flange.

The selected thickness faces are connected.

Uwaga: After connecting 2 flanges, you can convert the junction feature to a bend.

21.5.12 Working with Bends and Junctions

Two adjacent flanges of sheet metal parts are connected by a bend or a junction. As different from many other mechanical CAD systems, in BricsCAD, bends and junctions are separate features. A bend can be switched to a junction and vice versa, which allows you to fix or optimize the unfolded representation of a sheet metal part easily.

21.5.12.1 Converting Hard Edges to Bends or Junctions

If adjacent flanges of a sheet metal part are not connected by a bend or junction, the part cannot be unfolded or exported to a CAM system. Therefore such hard edges must be converted to bends or junctions.

Converting hard edges to bends

1 Do one of the following:

- Hover over a hard edge and click the **Create Bend** () tool in the **Quad** menu
- Click the **Create Bend** () tool button in the **Sheet Metal** ribbon tab
- Choose **Create Bend** in the **Sheet Metal** menu
- Type *SMBENDCREATE* or the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:

2 Select a flange face or a hard edge.

You are prompted:

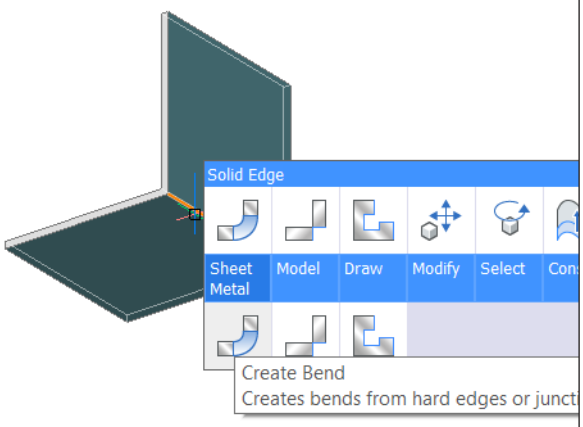
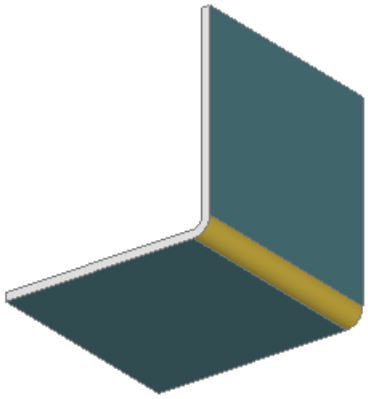
Entities in set: 1

Select flange faces or hard edges:

3 Do one of the following:

- Select more faces or hard edges.

- Right-click or press **Enter** to create the bends or junctions on the selected items.

	
Selected a hard edge	A bend is created

Converting Hard Edges to Junctions

1 Do one of the following:

- Hover over a hard edge and click the **Create Junction** ( ) tool in the **Quad** menu.
- Click the **Create Junction** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create Junction** in the **Sheet Metal** menu.
- Type *SMJUNCTIONCREATE* in the Command line.

  You are prompted: Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:

2 Select a flange face or a hard edge.

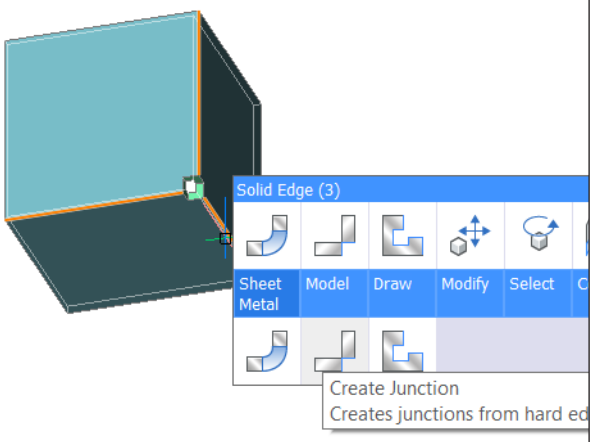
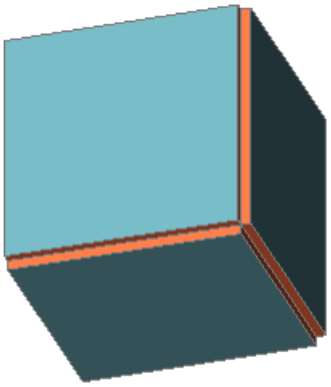
You are prompted:

Entities in set: 1

Select flange faces or hard edges:

3 Do one of the following:

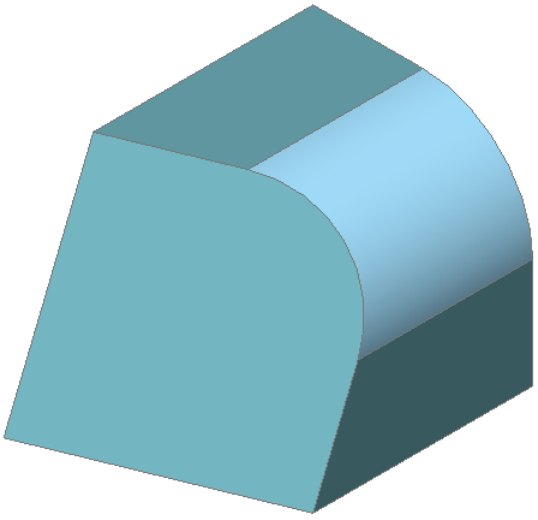
- Select more faces or hard edges.
- Right-click or press **Enter** to create the bends or junctions on the selected items.

	
Selected hard edges. Notice that a corner relief must be created first.	Junctions created. Notice that unnecessary material cuts have been removed due to **Smart Features**.

21.5.12.2 Creating Curved Junctions

Let us have a look at a more complex scenario where a junction has to be cut over a non-linear edge. To make the part unfoldable, we must:

- Cut the bend relief for one of the linear hard edges.
- Cut the junctions for the remaining linear and curved hard edges.



- Create a bend on the remaining hard edge.

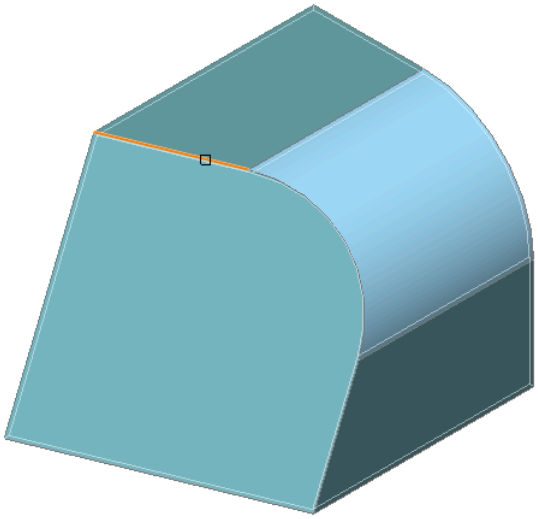
1 Create bend reliefs.

Do one of the following:

- Click the **Create Relief** tool button () on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.
- Type *SMRELIEFCREATE* in the Command line.

You are prompted:

Select a hard edge, bend face, flange face or 3D solid [Entire model]:



2 Select a hard edge.

You are prompted:

Select another hard edge or bend face for the corner relief:

3 Press **Enter** or right-click.

You are prompted:

Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

4 Press **Enter** or right-click.

2 bend reliefs are created for the edge:



We created 2 bend reliefs because a hard edge side with a "free" vertex needs a relief in order to make orthogonal thickness face.

We created a bend relief on the side adjacent to the lofted bend because this feature is different from a regular bend: creating a corner relief there would be too big.

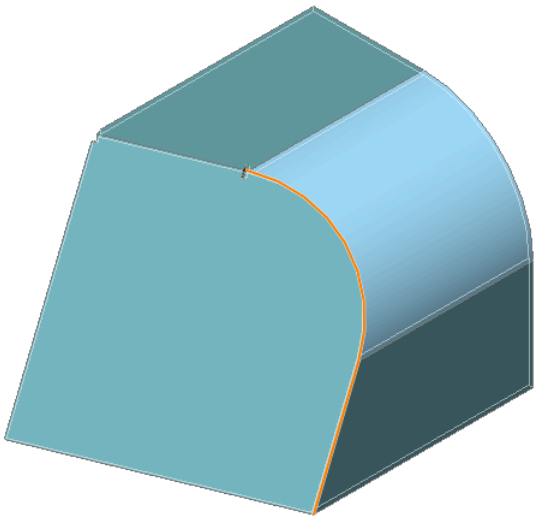
5 Create 2 junctions.

Do one of the following:

- Click the **Create Junction** ( ) tool buttons in the **Sheet Metal** ribbon tab;
- Choose **Create Junction** in the **Sheet Metal** menu;
- Type *SMJUNCTIONCREATE* in the command bar.

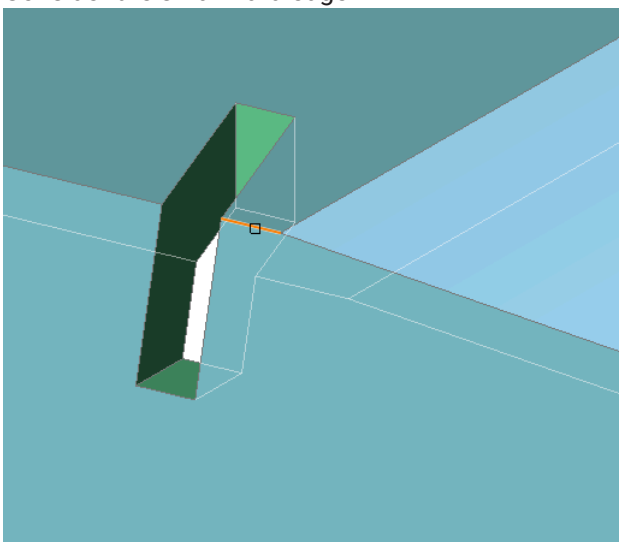
You are prompted:

Select hard edges or bends or flanges or 3D solids [Entire model] <Entire model>:



6 Select the following 3 hard edges:

Consider the small hard edge:



The cut is created. Parts which corresponded to linear edge get regular junction features.

7 Create a bend on the hard edge. Because it is the only hard edge left on the model, apply the command to the entire model.

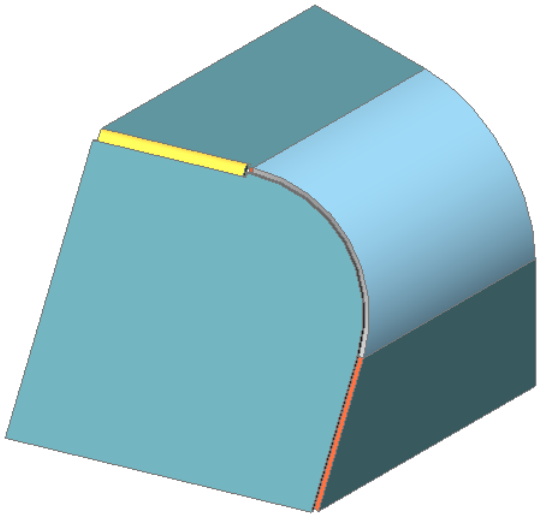
- Click the **Create Bend** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create Bend** in the **Sheet Metal** menu.
- Type *SMBENDCREATE* or the Command line.

  You are prompted:

  Select hard edges or junctions or flanges or 3D solids or [Entire model] <Entire model>:

8 Press **Enter**.

The bend is created.



9 Unfold the part.

- Click the **Unfold Body** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Unfold Body** in the **Sheet Metal** menu.
- Type *SMUNFOLD* or the Command line.

  You are prompted:

  Select a flange or lofted bend face to start unfolding [Associative]:

10 Select some flange face.

You are prompted:

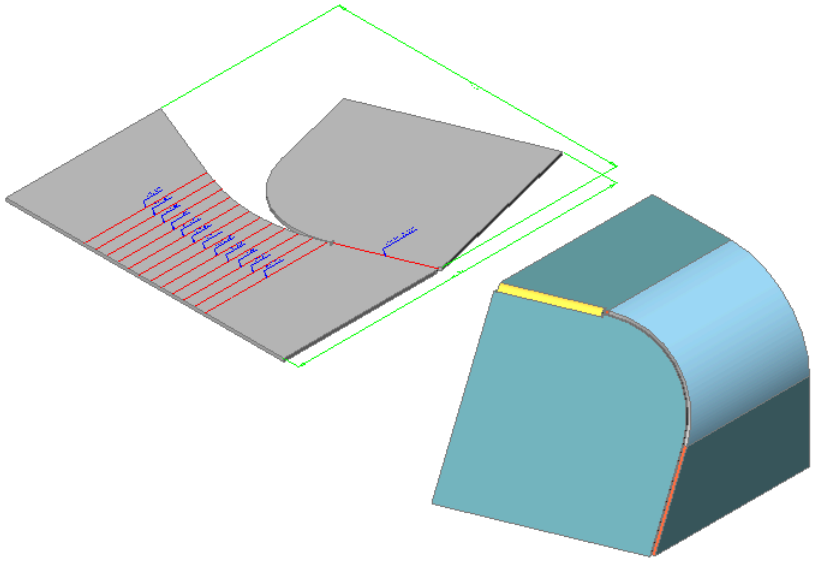
Select position of the unfolded body:

11 Place the unfolded part.

You are prompted:

Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>:

12 Press **Enter** to keep the



result.

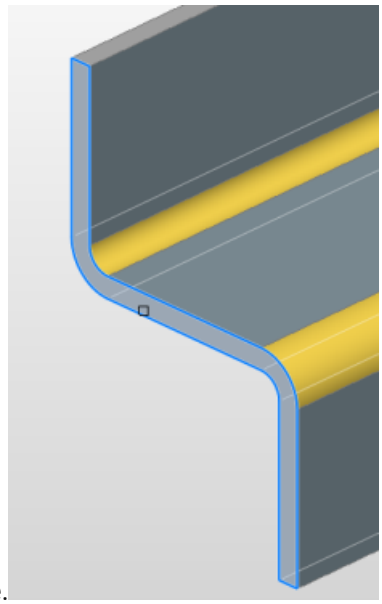
21.5.12.3 To Split a Thickness Face

To extend a flange which shares the same thickness face with other flanges, the thickness face must be split.

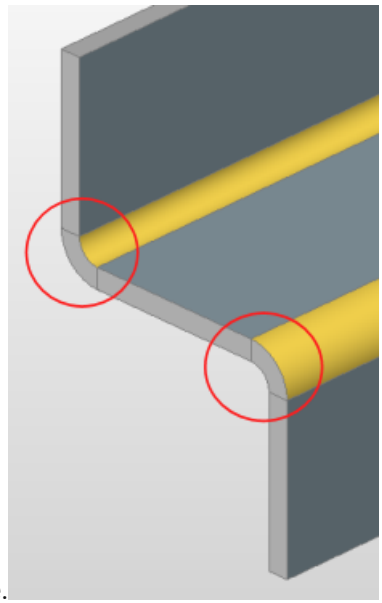
1 Launch the *SMIMPRINT* command ( ).

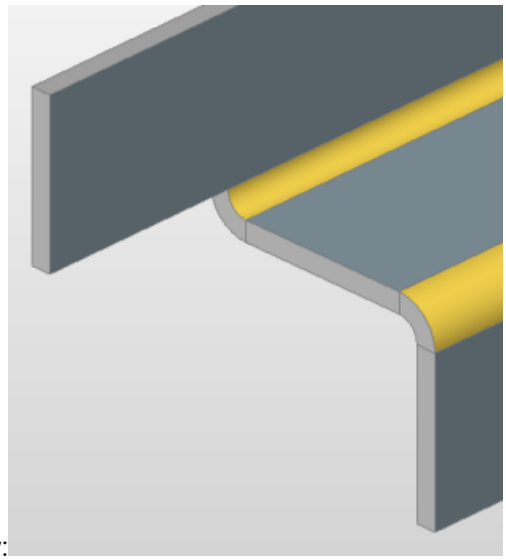
Prompts you: Select a thickness face:

2 Select the shared thickness face.



3 Right-click or press **Enter** to split the face.





4 The flanges can now be extended separately:

5 Execute the *DMSIMPLIFY* command to remove the unneeded splits, thus unifying the coplanar thickness faces.

21.5.12.4 Changing the Bend Radius

To change the overall bend radius:

1 Select the root node in the **Mechanical Browser**.

2 Find the **Bend radius** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the radius value in the **Value** field.

  The **Bend Radius** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Value** field.

The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

To change the bend radius for a particular bend:

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The Bend Properties display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Global value**.

  The current global bend radius is applied to the selected bend.

- Set the **Type** to **Absolute value**, then type the radius value in the **Value** field.

  The **Bend Radius** field changes accordingly.

- Set the **Type** to **Thickness Ratio** and type a value in the value field.

  The **Bend Radius** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

21.5.12.5 Changing the Junction Gap

To change the overall junction gap:

1 Select the root node in the **Mechanical Browser**.

2 Locate the **junction gap** group in the bottom part of the **Mechanical Browser**.

3 Do one of the following:

- Set **Type** to **Absolute Value**, then type the gap value in the **Value** field.

  The **junction gap** field changes accordingly.

- Set **Type** to **Thickness ratio**, then type a value in the **Value** field.

  The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

To change the junction gap for a particular bend:

1 Select the bend node in the **Mechanical Browser**. The selected bend highlights in the model. The Bend Properties display in the bottom part of the **Mechanical Browser**.

2 Do one of the following:

- Set the **Type** to **Global value**.

  The current global junction gap is applied to the selected bend.

- Set the **Type** to **Absolute value**, then type the gap value in the **Value** field.

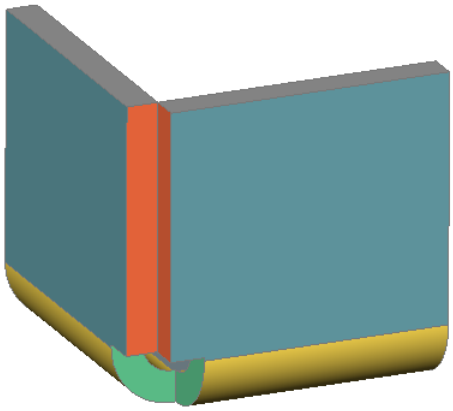
  The **junction gap** field changes accordingly.

- Set the **Type** to **Thickness Ratio** and type a value in the **Value** field.

  The **junction gap** field is calculated as the product of the **Thickness** and the **Thickness ratio**.

21.5.12.6 Changing the Junction Type

By default, BricsCAD creates symmetric junctions, as shown in the image below:

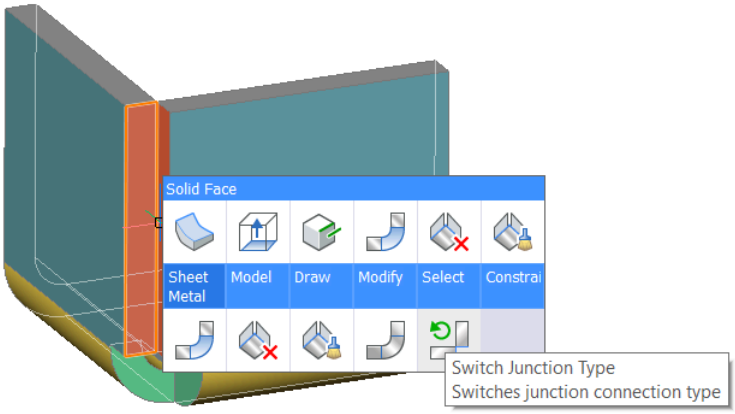


The *SMJUNCTIONSWITCH* command allows changing a symmetrical junction feature to one with overlapping faces.

To change the junction type using the Quad menu:

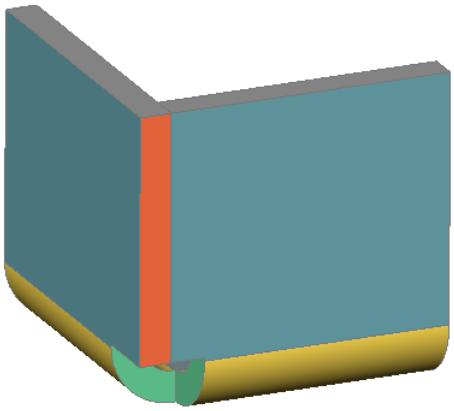
1 Hover the cursor over one the thickness faces of this junction.

2 Select **Switch Junction Type** in the **Sheet Metal** command group in the **Quad**



menu.

The selected thickness face is aligned with the flange face.



To change the junction type in the Mechanical Browser:

Method 1:

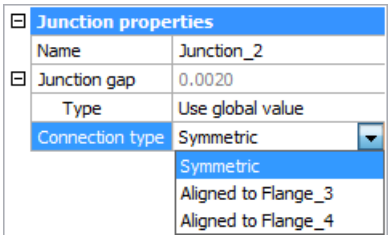
1 Select the corresponding junction in the tree.

The junction properties display in the bottom part of the **Mechanical Browser**.

2 Select the **Connection type** property.

3 Choose the desired junction type.

The options are: **Symmetric**, **Aligned to Flange_X** or **Aligned to Flange_Y**.



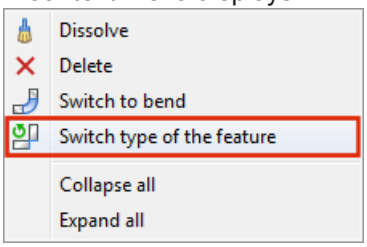
Rysunek 131: Junction properties

The junction is updated accordingly.

Method 2:

1 Select the corresponding junction in the tree.

A context menu displays:



2 Right-click and choose **Switch type of feature** in the context menu.

Uwaga: Both symmetric and aligned junctions maintain the same gap value between connected flanges.

21.5.13 Working with Hems

A hem feature is produced when the edge of a sheet metal flange is folded back on itself. *SMHEMCREATE* simplifies the creation of sheet metal hems. In BricsCAD hems have their own set of properties depending on the type, "flange" and "bend" type of the geometry. Hem features are recognized by the *SMCONVERT* command and have a special symbol when unfolded with *SMUNFOLD*.

21.5.13.1 Create a Hem

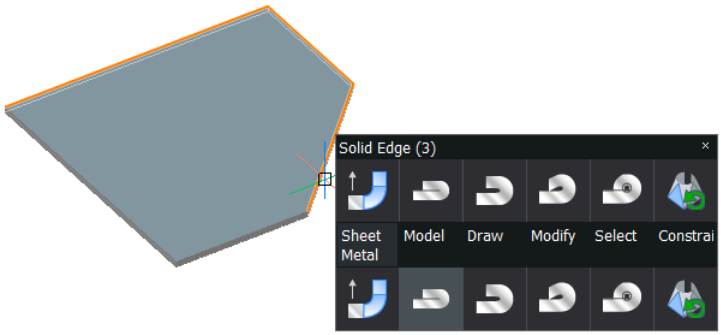
1 Do one of the following:

- Select several linear edges on the flange.
- Hover the cursor over a linear edge of the flange.

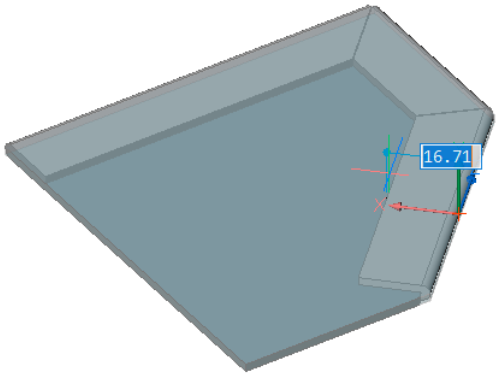
2 Do one of the following:

- Select **Create a Closed Hem** ( ) tool in the **Quad** menu.

- Click the **Create a Closed Hem** ( ) tool buttons in the **Sheet Metal** ribbon tab.
- Choose **Create a Closed Hem** in the **Sheet Metal** menu.
- Type *SMHEMCREATE* in the Command

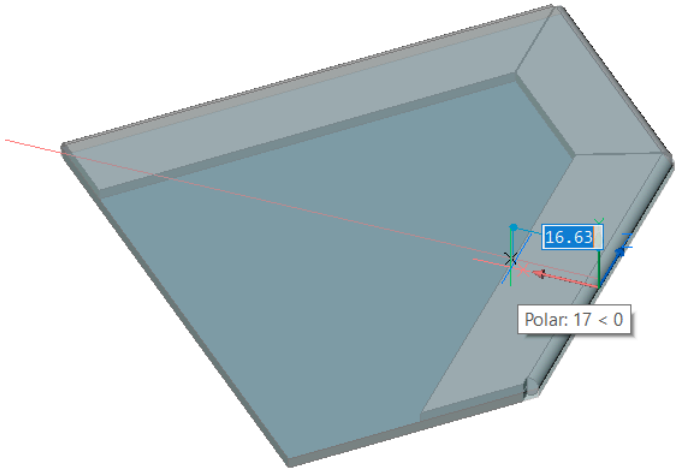


line.



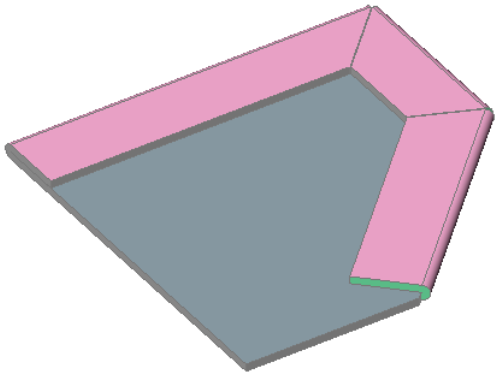
3 Hover the cursor to adjust the hem length:

4 Use the **Ctrl** widget or **Toggle tapering** option to flip the adjustment of a hem to the



edge:





5 When finished, press **Enter**:

Hems and auxiliary features (reliefs and miters) are created.

21.5.13.2 Modify Hem Parameters

To modify Hem parameters select the feature and use the **Mechanical Browser**. Different parameters are available, depending on the type of the hem.

Tabela 43: Hem parameters

Type of hem	Available parameters
Closed	Radius Length

Tabela 43: Hem parameters

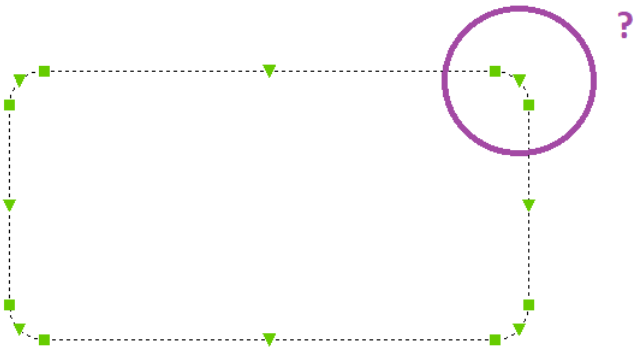
Type of hem	Available parameters
Open	Radius Length
Teardrop	Radius - change the angle of the hem Length - change the angle of the hem
Round	Radius - change the "planar" part of the hem

In the case of any parametric change, the hem type is preserved and the geometry is moved inside, to maintain external dimensions.

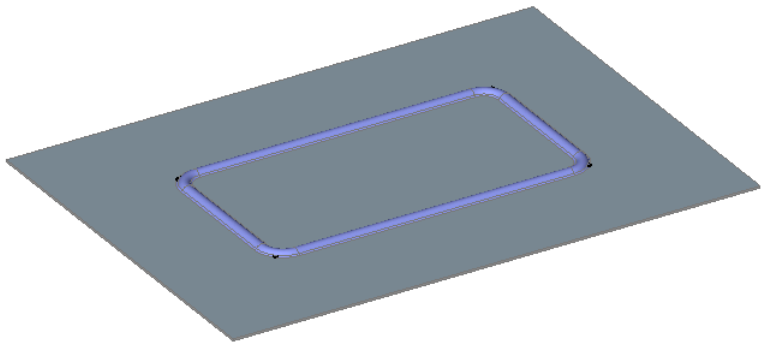
21.5.13.3 Erase a Hem

When a hem feature is erased, BricsCAD intelligently restores the geometry's dimensions and reliefs.

The flange part below has external dimensions equal to 100.

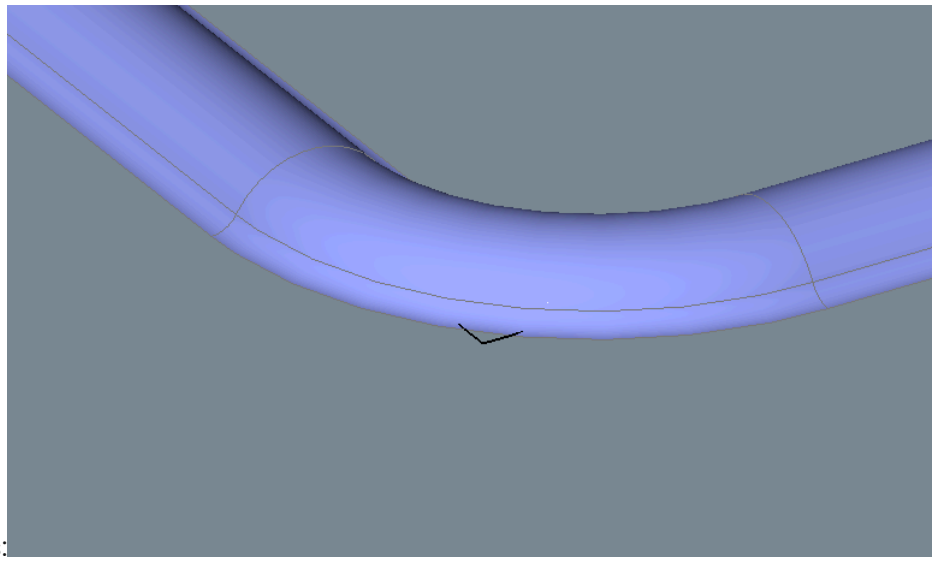


1 Use the Width option of the *SMHEMCREATE* command to create a partial hem on a single

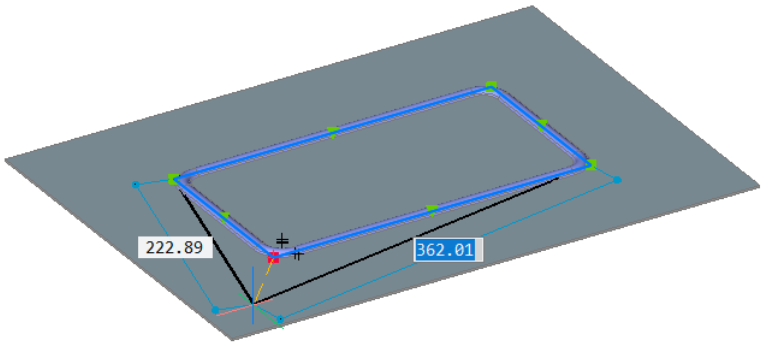


edge:

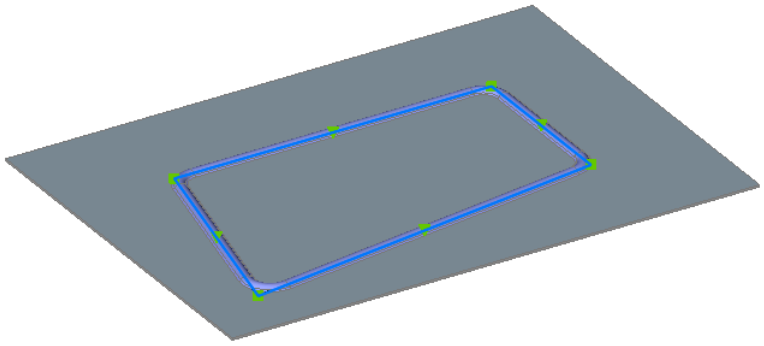
2 Create hems on 2 other

edges: 

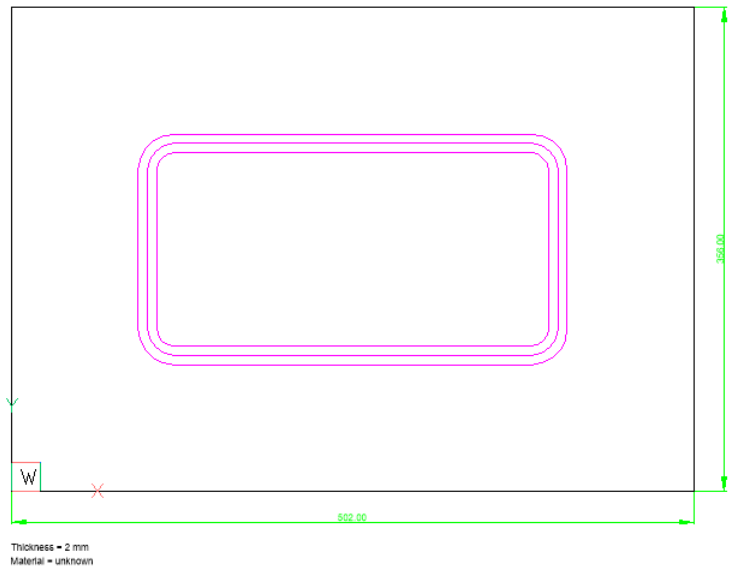
These operations lead to the appearance of complex relief geometry:



Uwaga: The external dimensions of the part are preserved.

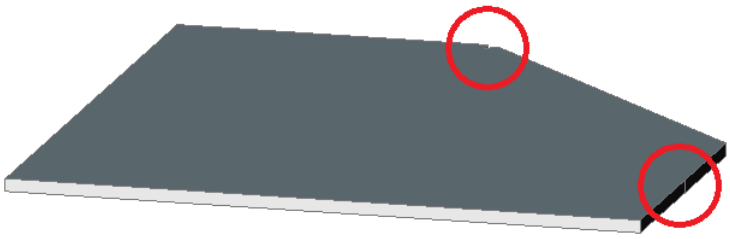


3 Select the hem features and delete

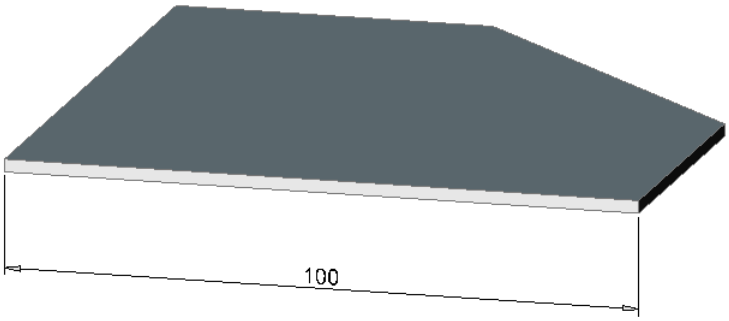


them.

Some imperfections are left on the corner relief feature and the imprinted edge:



4 Delete the corner relief feature and use *DMSIMPLIFY* to get the original part with original



dimensions.

21.5.13.4 Related commands

The *MECHANICALBROWSEROPEN* command displays the Mechanical Browser.

The *SMDELETE* command restores a hard edge between 2 flanges to remove a bend or a junction - removes a flange and all adjacent bends. The adjacent flanges are extended up to a junction configuration and the flange is deleted. Restores geometry removed by the miter.

The *SMHEMCREATE* command creates one or more hems to a sheet metal part by pulling one or more edges of an existing flange.

21.5.14 Working with Tabs

Tab features are used to create easy and robust connections between sheet metal parts.

Tab features are created from junctions.

21.5.14.1 Creating a Single Tab

A single tab is created by default by the *SMTABCREATE* command. This feature only has a single slot.

1 Do one of the following:

- Hover over a junction face and click the **Create single Tab** ( ) tool in the **Quad** menu.
- Click the **Create single Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create single Tab** in the **Sheet Metal** menu.
- Type *SMTABCREATE* in the Command line.

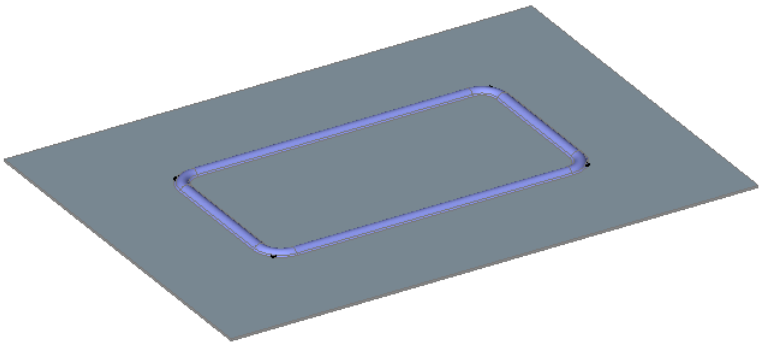
  You are prompted: Select junction to create single tab or [Array]:

2 Select a junction face.

  You are prompted: Enter the length of the tab or [Central point]:

3 Define the length of the tab.

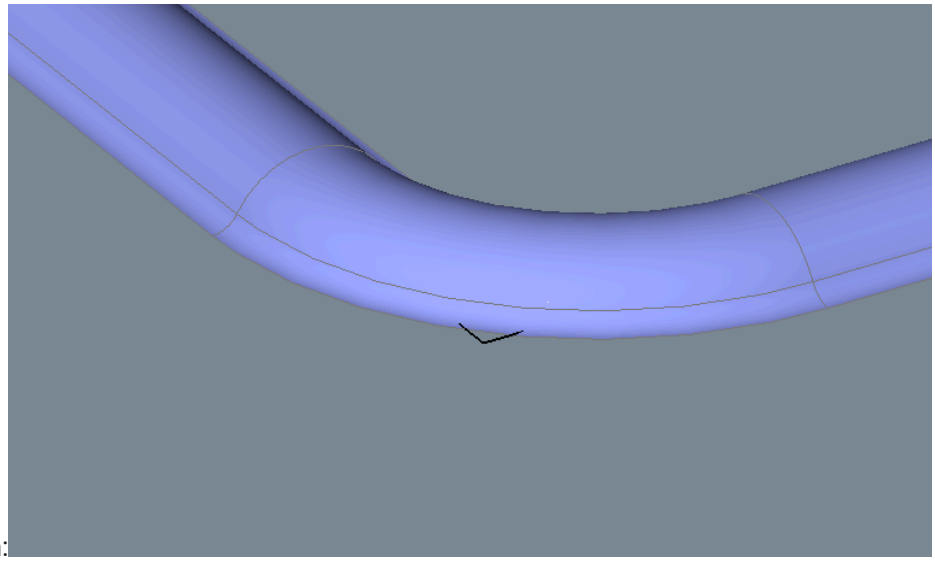
  A tab is placed symmetrically in relation to a central point. Type values in the dynamic length entry fields to define the length of a tab.



4 Select the **Central point** option to position the tab.

  You are prompted: Select the central point or [Length]:

2 Tab-controlled dynamic dimensions are used to create a tab relative to a junction:



5 Select the **Length** option to specify the length, or press **Enter** to finish the command.

21.5.14.2 Creating an Array Tab

An array tab feature has several slots, in contrast to Single tabs.

1 To create an array tab, do one of the following:

- Hover over a junction face and click the **Create array Tab** ( ) tool in the **Quad** menu.
- Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create array Tab** from the **Sheet Metal** menu.
- Type *SMTABCREATE* in the Command line.

  You are prompted: Select junction to create single tab or [Array]:

2 Select the **Array** option.

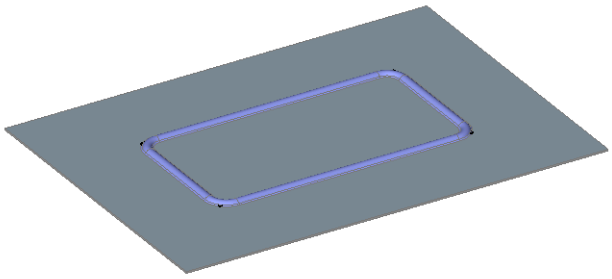
You are prompted: Select junction or curved hard edge to create array tab or [Single tab/selection options (?)]:

3 Select a junction face.

You are prompted: Accept tab configuration or [Flip/slot Number/Distance/Length/Round radius/ CHamfer distance/Accept] <Accept>:

4 You can use options to modify the appearance of an array tab (see the example below).

- **Slot Number = 3** - notice that the picture is not symmetric. If you click the **Flip** option, 3 slots will appear on the other side of the junction.
- **Length = 8** - the length of the slot.
- **Distance = 40** - the dimension between slot

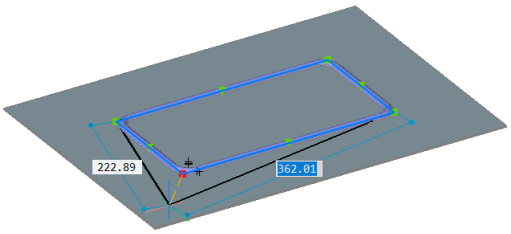
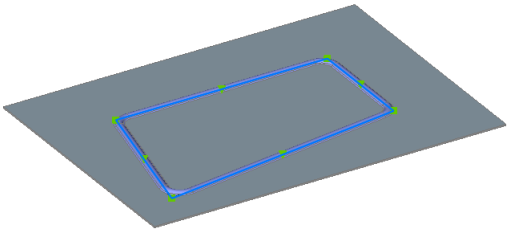


centers.

5 Change the **Round radius** and **Chamfer distance** to define the fillets and chamfers for the edges.

Wskazówka: If you set fillets or chamfers by mistake, use zero values to turn them off.

6 Once you have set all the options, choose **Accept**, or press **Enter** to create the tab and finish the command.

Fillets	Chamfers
	

21.5.14.3 Creating a Curved Array Tab

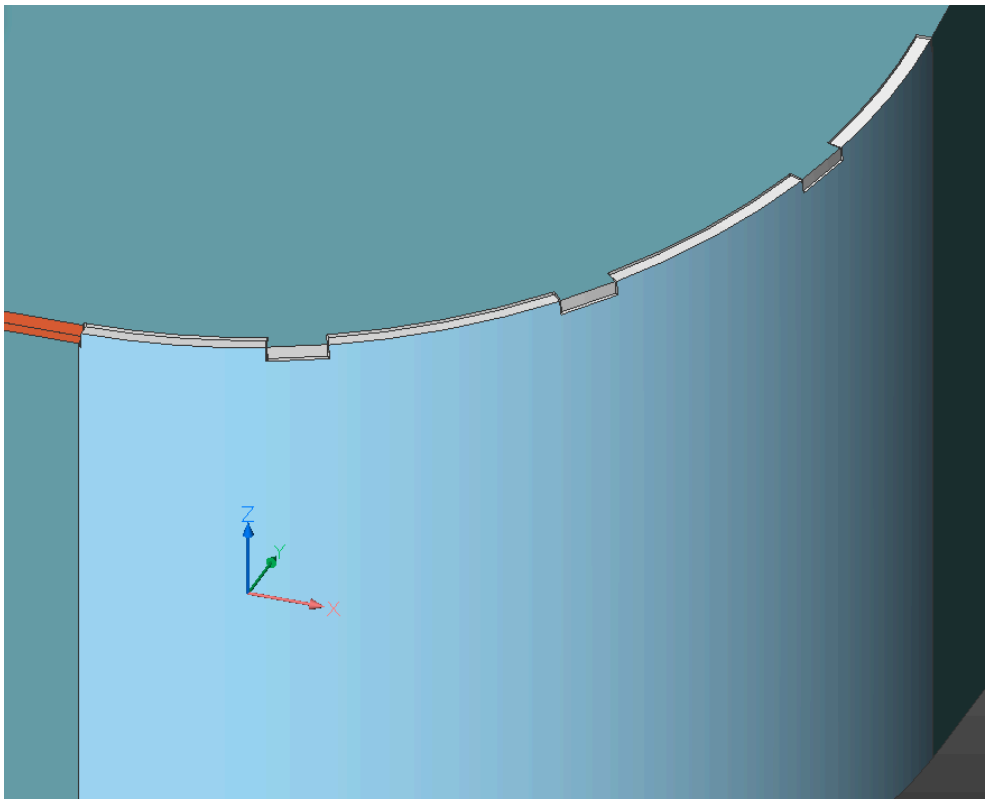
This case is similar to creation of straight Array Tab, but the tabs are placed along a curved path.

1 To create a curved array tab, do one of the following:

- Hover over a curved hard edge and click the **Create array Tab** ( ) tool in the **Quad** menu.
- Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
- Choose **Create array Tab** from the **Sheet Metal** menu.
- Type *SMTABCREATE* in the Command line.

  You are prompted: Select junction or curved hard edge to create array tab or [Single tab/ selection options (?)]:

2 Select a curved hard edge.

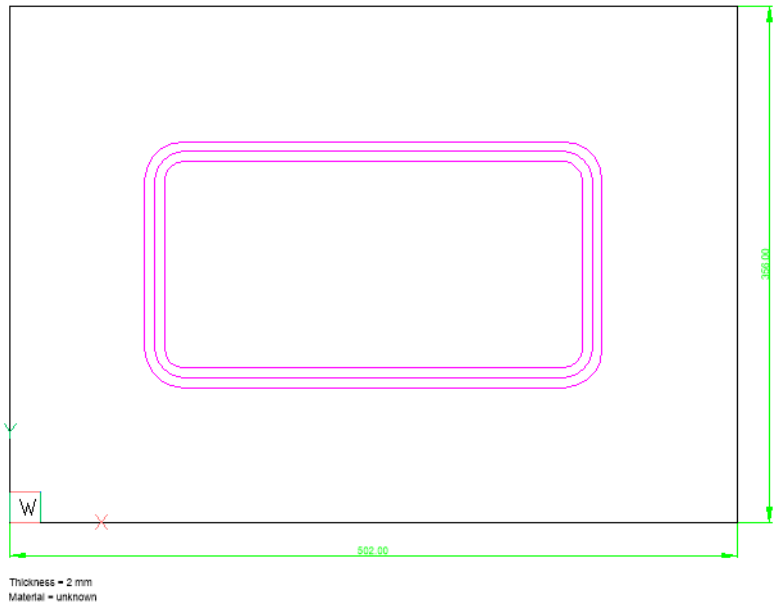


21.5.14.4 Modifying Tabs

Single and array tabs have different editable properties, which are available in the **Mechanical Browser**.

Click the tab feature name to edit.

A single tab has **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance** parameters. **Tab clearance** defines the gap between the tab and the slot faces.



An array tab has **Slot number**, **Distance**, **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance** parameters.

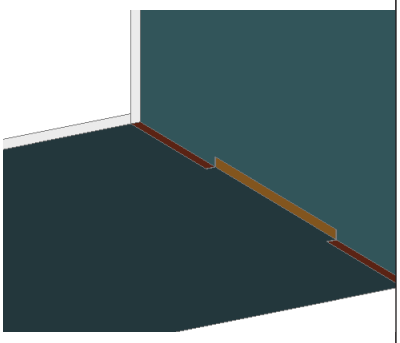
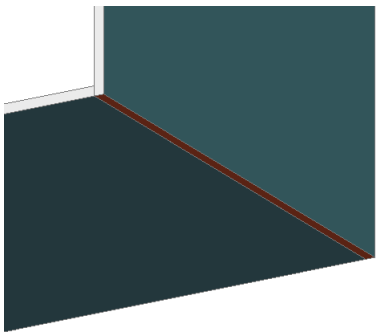
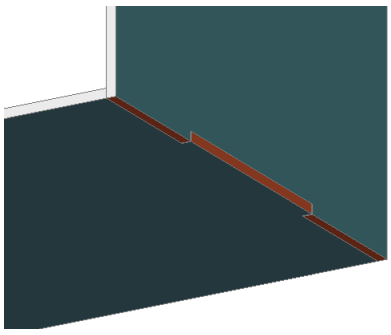
Uwaga: Curved Array Tabs can not be modified.

21.5.14.5 Deleting Tabs

Use the *SMDELETE* command to delete tabs (not applicable for curved). A junction feature is created in place of the tabs.

21.5.14.6 Exploding Tabs

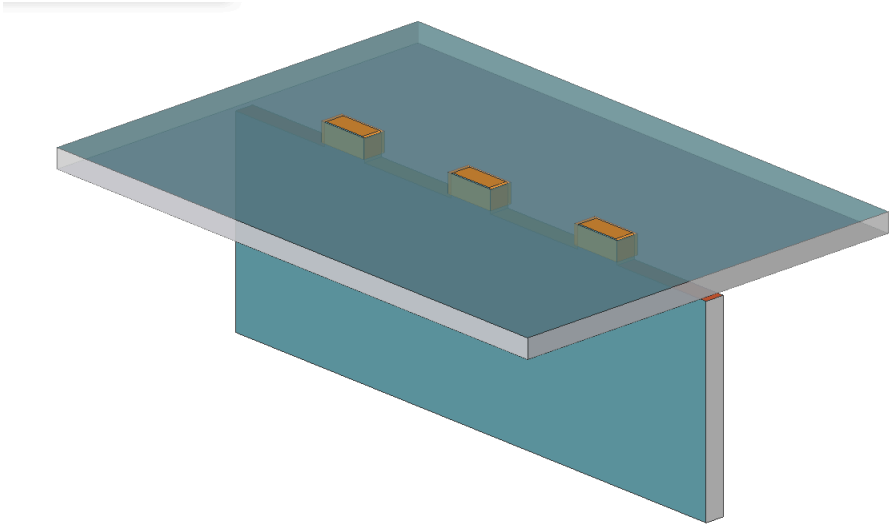
Use the *SMEXPLODE* command to explode a tabbed junction to a set of tab features. The tab geometry remains the same and its faces are distributed among several junction features.

Original tab feature	Result of deletion	Result of explode
		

21.5.15 Working with T-tabs and T-junctions

21.5.15.1 Overview

You can use T-tab features to create a connection with robust positioning between 2 sheet metal flanges. Unlike normal tabs, T-tabs form a T-like configuration, when one flange is attached to the other.



T-tabs can be created from T-junctions by the *SmTabCreate* command. T-junction features are often used as an intermediate stage for making T-tabs.

21.5.15.2 Creating a T-junction

A T-junction can be created on existing separate flanges in the following way:

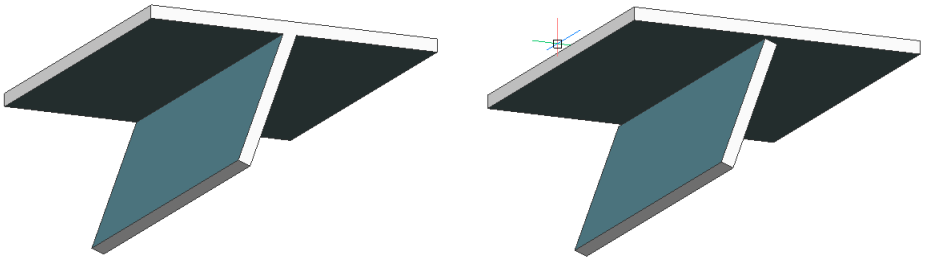
1 Select one thickness face on the flange.

2 Do one of the following:

- Click the **Connect Flanges** () tool button in the **Sheet Metal** | **Junction** ribbon tab.
- Choose **Connect Flanges** in the **Sheet Metal** | **Junction** menu.
- Type *SMFLANGECONNECT* in the Command line.

3 Press **Enter**.

Alternatively, you can create a T-junction feature in place of a T-like hard edge between 2 flanges (image at the left):



1 Hover over a hard edge between the flanges and click the **Create Junction** ( ) tool in the **Quad** menu, or do one of the following:
   - Click the **Create Junction** ( ) tool button in the **Sheet Metal** ribbon tab.
   - Choose **Create Junction** from the **Sheet Metal** | **Junction** menu.
   - Type *SMJUNCTIONCREATE* in the Command line.
2 Select a hard edge.

21.5.15.3 Creating a Single T-tab

The *SMTABCREATE* command creates a single T-tab by default. This feature has only a single slot.

To create a single T-tab:

1 Hover over a T-junction face and click the **Create single Tab** ( ) tool in the **Quad** menu, or do one of the following:
   - Click the **Create single Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
   - Choose **Create single Tab** from the **Sheet Metal** | **Tab** menu
   - Type *SMTABCREATE* in the Command line.
2 Select a T-junction face.

21.5.15.4 Creating an Array T-tab

Different from Single T-tabs, array T-tab features have multiple slots.

To create an array T-tab:

1 Hover over a T-junction face and click the **Create array Tab** ( ) tool in the **Quad** menu, or do one of the following:
   - Click the **Create array Tab** ( ) tool button in the **Sheet Metal** ribbon tab.
   - Choose **Create array Tab** from the **Sheet Metal** | **Tab** menu.
   - Type *SMTABCREATE* in the Command line.

     You are prompted: Select junction to create single tab or [Array]: select **Array** option.
2 Select a T-junction face.

21.5.15.5 Modifying T-junction and T-tab Parameters

A T-junction has only the **Junction gap** parameter.

Array T-tabs and Single T-tabs have the following properties: **Distance**, **Length**, **Clearance**, **Fillet radius**, and **Chamfer distance**. Array T-Tabs also have the **Slot number** parameter.

You can set the value of all parameters in the **Mechanical Browser** panel.

21.5.15.6 Deleting a T-junction or T-tab

When a T-tab feature is deleted, BricsCAD restores a T-junction feature with the corresponding geometry. Erasing a T-Junction joins the flanges and creates a T-like hard edge between them.

21.5.15.7 Related commands

The *MECHANICALBROWSEROPEN* command displays the Mechanical Browser.

The *SMDELETE* command erases a feature.

21.5.16 Working with Bevels

21.5.16.1 Overview

Bevel features represent a chamfer operation made on Sheet Metal parts. On the level of BricsCAD V21 each bevel should be exactly on a single flange. There are no specific operations to create a bevel, so the *DMCHAMFER* command must be used on an edge of a flange, and then use *DMROTATE* to adjust the bevel angle. The *SMCONVERT* command recognizes the bevels and is processed by *SMUNFOLD*. Bevel features are also supported by the *SMDELETE* and *SMREPAIR* commands.

This procedure applies to 2 scenarios:

- Working with importing geometry, including batch export by *SMASSEMBLYEXPORT*.
- Modifying parts.

We distinguish between the following bevel types:

Tabela 44: Bevel types

	One-side bevel

Tabela 44: Bevel types

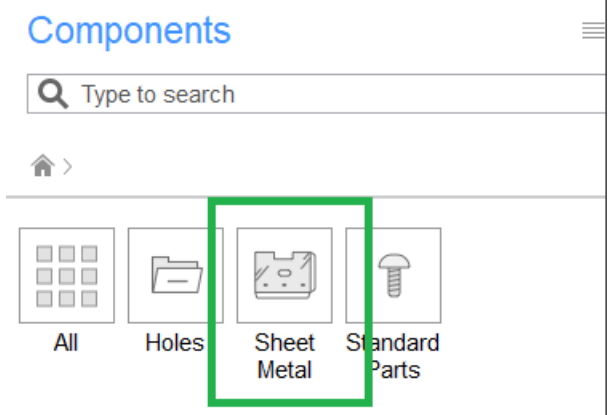
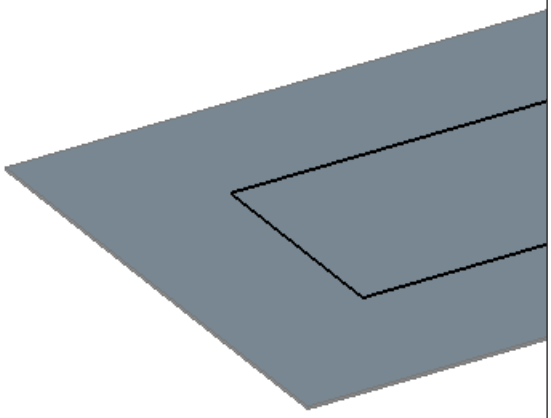
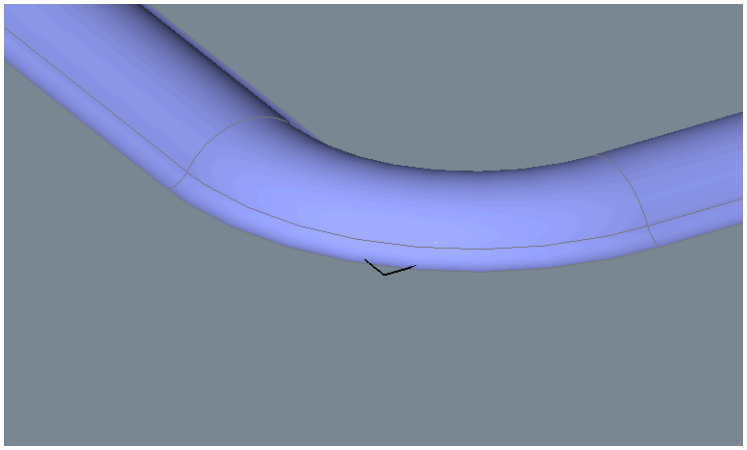
	Land bevel
	K-bevel

Tabela 44: Bevel types

	X-bevel

In the **Mechanical Browser**, the properties of each chamfer of a bevel feature are available separately. These properties are:



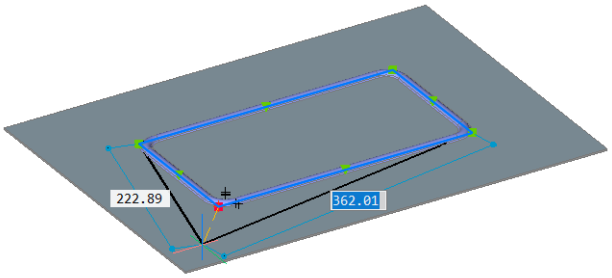
Height (H): the depth of the chamfer operation, much like the *DMCHAMFER* command does.

Angle (A): the slope of the chamfered face.

Uwaga: In the Mechanical Browser properties, Top or Bottom are not used to describe bevel features.

21.5.16.2 Recognizing bevels

Let us start with a sheet metal part with bevel geometry (chamfered flange edges).

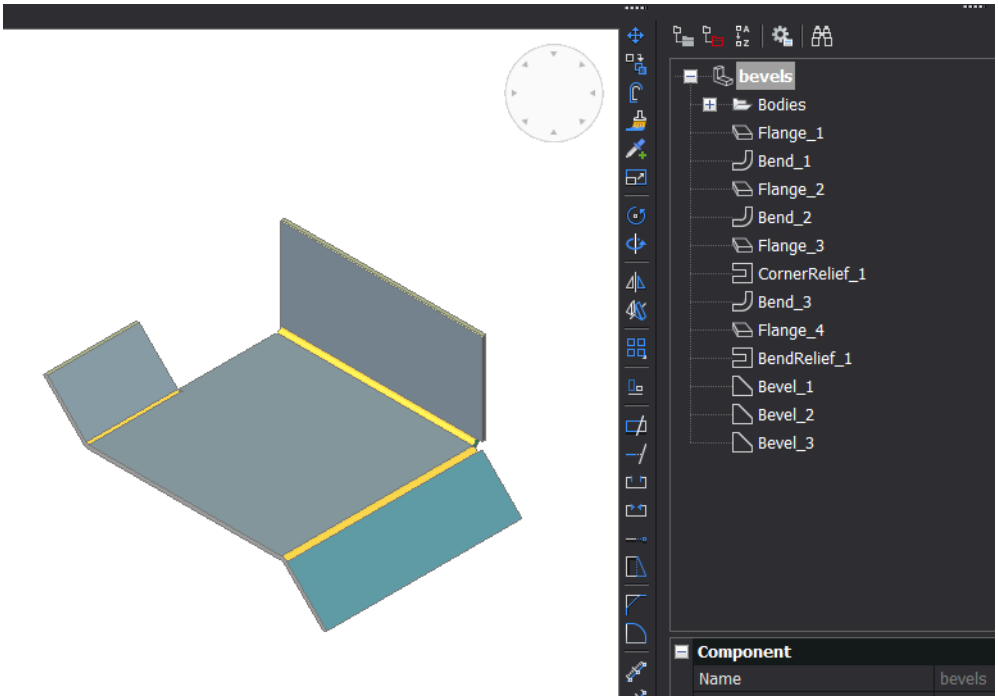


1 Click the **Convert to Sheet Metal** ( ) tool button on the **Sheet Metal / Create** Ribbon panel.

You are prompted: Select 3D solids or seed faces or [Entire model/selection options (?)] <Entire model>:

2 Press **Enter** to convert the 3D solid to a sheet metal part.

The bevel features are recognized and appear in **Mechanical Browser** and show on the part:



21.5.16.3 Delete and Repair

The following round trip does not change the overall dimensions of the part: Create the bevel geometry and feature using *DMCHAMFER*/*SMCONVERT*, then delete the bevel using *SMDELETE* and restart the procedure.

When using *SMDELETE* to delete a bevel the volume is **always added** to the part because the dimensions of the part can not be reduced.

Support in the *SMREPAIR* command is also important, since this command is a part of the Import and Rework workflow.

Uwaga: If you apply the *SMREPAIR* command to a part with chamfers but without bevel features, chamfers will be lost and dimensions will change, depending on the selected side of the part.

21.5.16.4 Unfold

Bevel features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

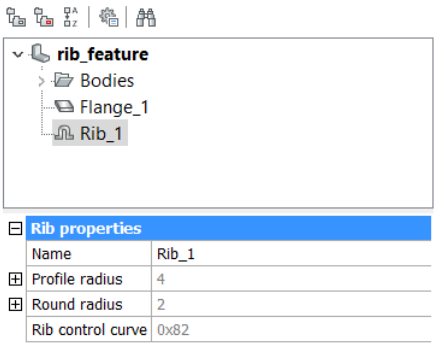
Keep: Does nothing with bevel geometry, resulting in a 3D solid with chamfers. These chamfered faces will likely produce artifacts in the 2D .DXF.

Remove: Same result as applying *SMDELETE* on all the bevel features: all the chops are filled.

Annotate: Each bevel feature is removed and the edge is annotated. This mode is recommended.

This is the result for the example part:

- Each bevel edge has been annotated with a pink text on the **Bevel Annotations** layer.
- UP and DOWN are with respect to the selected side for unfolding. When choosing the opposite side, all the UP annotations become DOWN, and vice versa.
- For **One-side** and **Land bevels** a single chamfer operation is needed only: a single UP or Down annotation is created.
- For **K-** and **X- bevels** 2 chamfer operations are needed: UP and DOWN annotations are created.



21.5.17 Tworzenie i Dzielenie Wyciągniętych Gięć

BricsCAD pozwala tworzyć lub rozpoznawać wyciągane części arkusza blachy z krzywoliniową geometrią, które można wytwarzać za pomocą wielokrotnego gięcia.

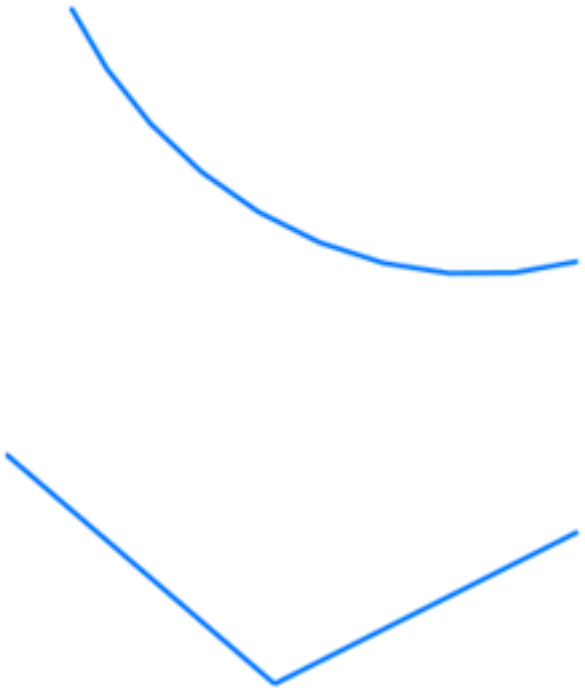
21.5.17.1 Ogólna procedura tworzenia części arkusza blachy z wyciągniętym gięciem

1 Uruchom *SMLOFT* Komenda.

2 Wykonaj jedną z następujących czynności:

- Kliknij przycisk narzędzia **Utwórz wywinięte gięcie** () na pasku narzędzi **Arkusz Blachy**.
- Kliknij przycisk narzędzia **Utwórz wywinięte gięcie** () na pasku wstążki **Arkusz Blachy**.
- Wybierać **Utwórz wyciągnięcie złożone arkusza blachy** w menu **Metalowa blacha** .

  Pasek poleceń wyświetli: Wybierz dwa przekroje:



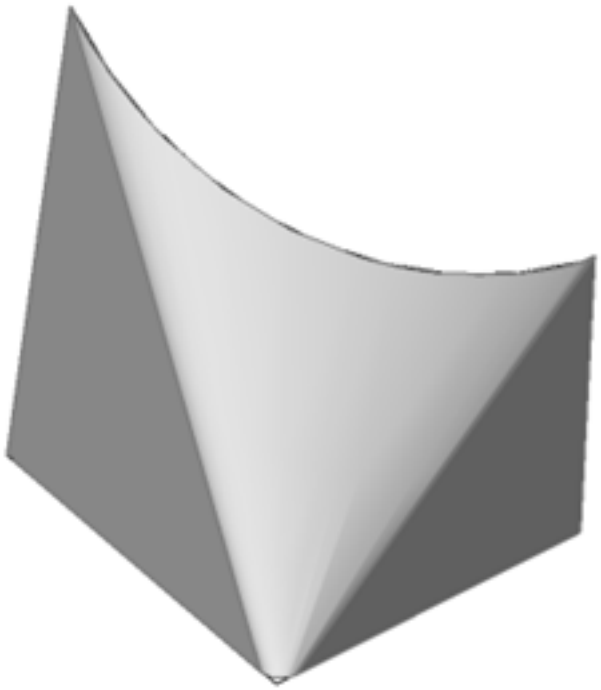
3 Wybierz dwa elementy profilu.

Pasek poleceń wyświetli: Ustaw zaokrąglenia Promień/Grubość/Pojedyncze gięcie/<Kołnierze z zagięciami>:

4 naciskać **Wejść** aby zaakceptować ustawienia domyślne.

Tworzony jest podgląd części arkusza blachy.

Pasek poleceń wyświetli: Ustaw zaokrąglenia Promień/Grubość/Pojedyncze zagięcie/Kołnierze z zagięciami lub pogrub Obie strony/Wewnątrz/<Na zewnątrz>:

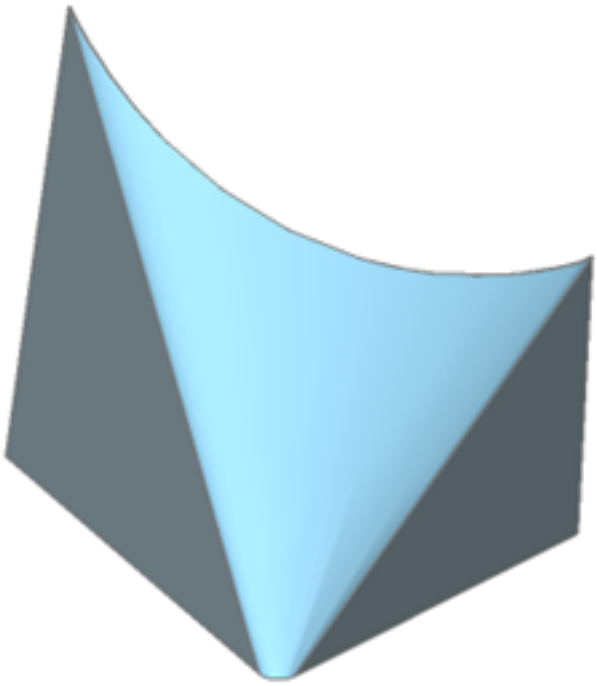




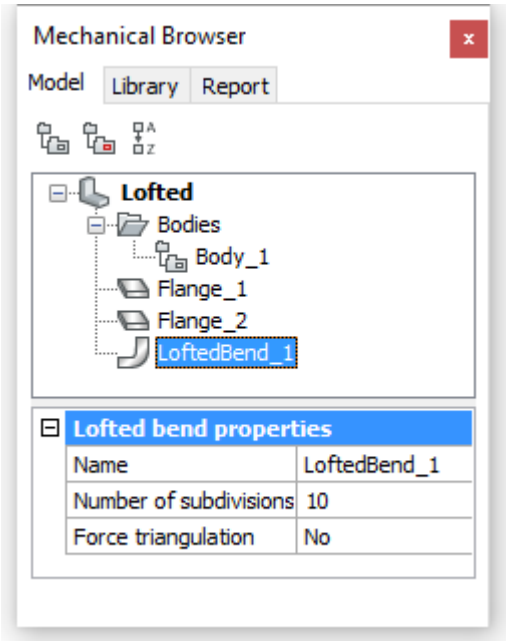
5 wciśnij **KLAWISZ KONTROLNY** klucz do wyboru położenia grubości części arkusza blachy względem profili źródłowych: **Lewo** , **Środek** lub **Dobrze** .

Gdyby **PORADY** jest WŁĄCZONY, wyświetla się widżet podpowiedzi.

6 naciskać **Wejść** aby utworzyć część arkusza blachy.



Gdyby **FUNKCJEKOLORY** jest ON, można rozróżnić 2 kołnierze ( ) i wyciągniętego zakrętu ( ). Właściwości wyciągniętego gięcia są wyświetlane w **Przeglądarce Mechanicznej**:



Rysunek 132: Właściwości wyciągniętego gięcia

Opcjonalnie wybierz element wyciągniętego gięcia, aby edytować jego właściwości:

- **Liczba podpodziałów**: liczba płaskich regionów w rozwiniętej reprezentacji elementu wyciągniętego gięcia (im większa liczba, tym więcej linii gięcia zostanie dodanych do rozwiniętego widoku dla wyciągniętego gięcia).

- **Triangulacja sił** : większość wyciągniętych zagięć może być rozwinięta za pomocą inteligentnego algorytmu teselacji poczwórnej. Czasami jednak może to dawać nieoptymalne wyniki. W takich przypadkach ustaw tę właściwość na *Tak*, aby przełączyć się na bardziej prosty algorytm, który tworzy kwadraty przypominające trójkąt.

Samouczek: Wyciągnięte gięcia w BricsCAD Arkusz Blachy

21.5.17.2 Aby Podzielić Wyciągnięte Gięcie

Wyciągnięte Gięcia mogą zostać podzielone wzdłuż linii na wyciągniętej powierzchni w dowolnym punkcie:

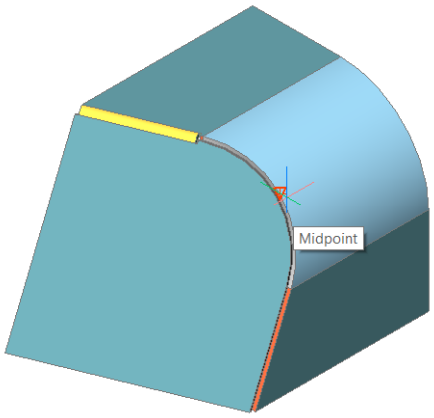
1 Wykonaj jedną z następujących czynności:
   - Kliknij narzędzie **Podziel** () na karcie wstążki **Arkusz Blach**.
   - Wybierz opcję **Podziel** z menu **Arkusz Blachy**.
   - Wpisz **smsplit** w pasku poleceń.

     Pasek poleceń wyświetli: Wybierz kołnierz lub wyciągniętą powierzchnię zagięcia:

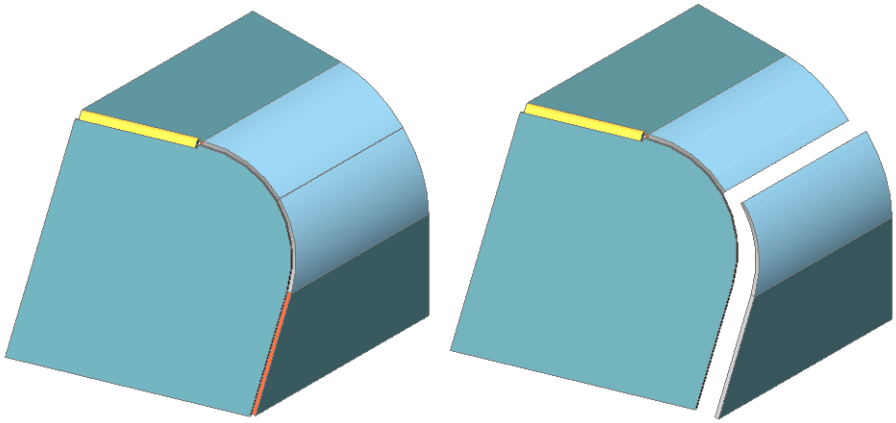
2 Wybierz powierzchnię wyciągniętego gięcia.

   Pasek poleceń wyświetli: Wybierz punkt na powierzchni wyciągniętego gięcia:

3 Wybierz punkt na powierzchni wyciągniętego gięcia.



Wyciągnięte gięcie jest podzielone. Jeśli połączone złącze jest rozłożone, możesz swobodnie przesuwać oddzieloną częścią.



21.5.18 Bend Relief and Corner Relief Feature Operations

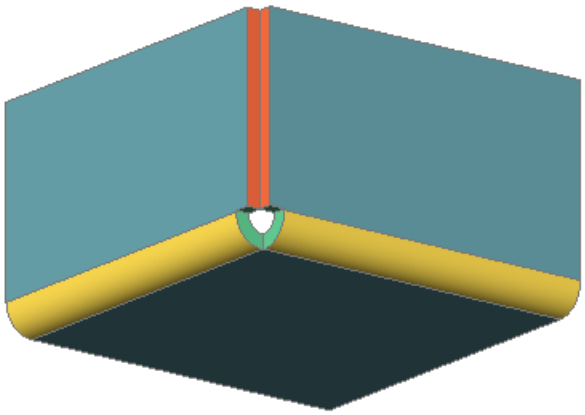
21.5.18.1 Overview

Use the *SMFLANGEEDGE*, *SMRELIEFCREATE*, *SMJUNCTIONSWITCH*, and *SMRELIEFSWITCH* commands.

Reliefs are needed in sheet metal parts to avoid material tear when bent on press breaks. Basically you must not care about reliefs, because BricsCAD automatically creates all needed reliefs when you create a bend (implicitly or explicitly). However, if you rework poor sheet metal parts, you may find it useful to create corner and bend reliefs with the dedicated tools.

21.5.18.2 Corner Reliefs and Junctions

When you create a flange using the *SMFLANGEEDGE* command, adjacent to a bend edge, a corner relief is created automatically in the corner where the 3 flanges meet. In addition, a junction is created between the 2 flanges, which are not connected with a bend.



21.5.18.3 Creating Missing Reliefs

1 Do one of the following:

- Click the **Create Relief** tool button ( ) on the **Sheet Metal** ribbon tab.
- Choose **Create Relief** in the **Sheet Metal** menu.

- Type *SMRELIEFCREATE* in the Command line.

  You are prompted: Select a hard edge, bend face, flange face or 3D solid [Entire model]:

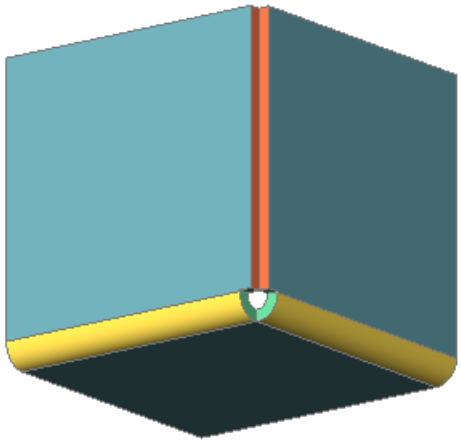
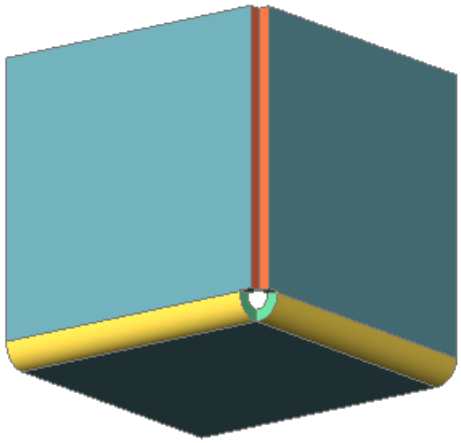
2 Select an item.

  Enter relief size by bend radius ratio or [force Bend reliefs/Auto] <Auto>:

3 Do one of the following:

  - Press **Enter** or right-click to accept the default bend radius ratio.
  - Type a value at the command prompt, then press **Enter**.

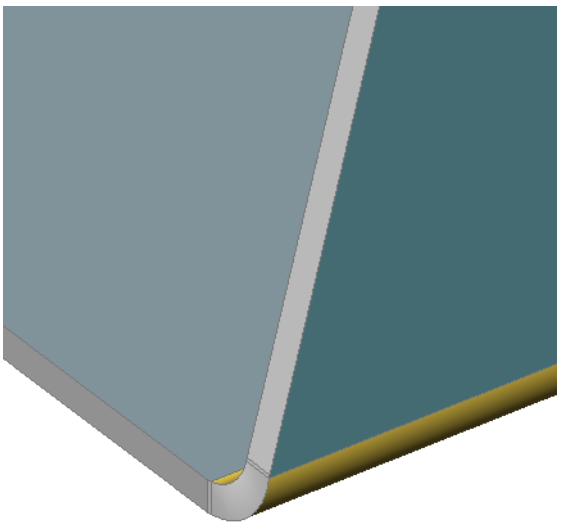
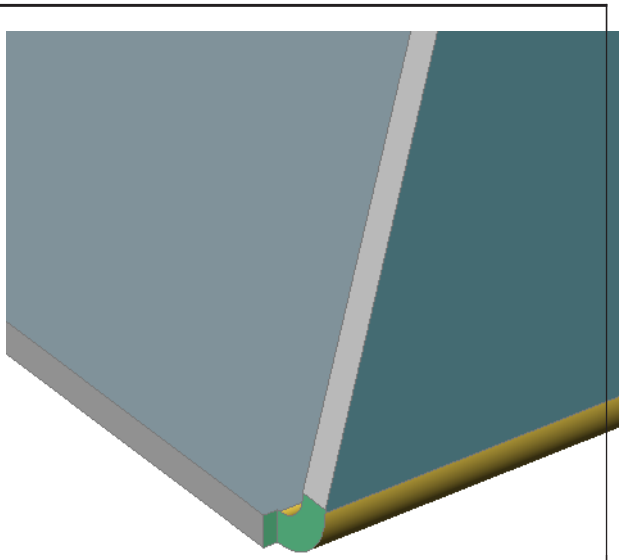
    The relief is created.

	
Selected bend faces	Created reliefs

Uwaga:

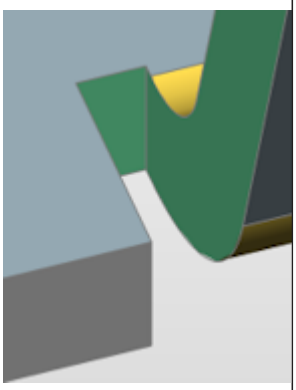
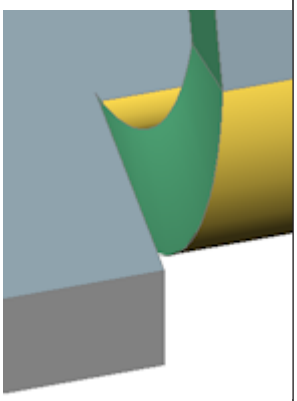
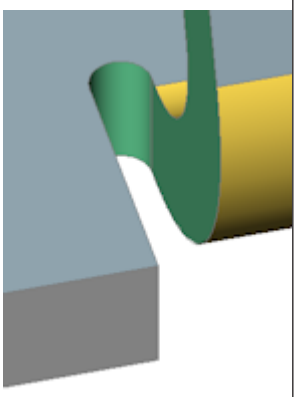
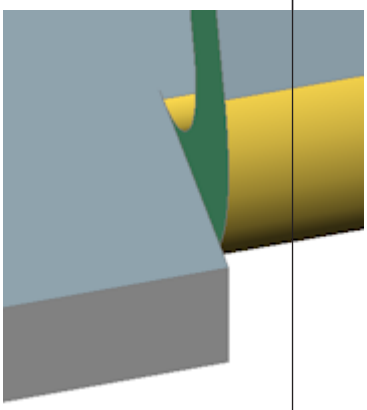
- The selected items can be a 3D solid with sheet metal features, a face or a hard edge of the solid.
- To create a relief locally, you can select 1 or 2 hard edges, 1 or 2 bends or a bend and a hard edge.
- If a 3D solid is selected, the command recognizes the places where corner or bend reliefs are needed and creates them automatically.
- If 2 hard edges or 2 bends or a bend and a hard edge at a corner are selected, the command tries to create a corner relief.

Option **force Bend reliefs** forces creation of the bend relief even if there is no need to create a bend relief. As shown in the image below, there is no need to create a bend relief, but the geometry is different from rectangular cut.

	
A bend relief is not needed because of a smooth transition of the geometry.	A rectangular bend relief is created only if the option **force Bend reliefs** is enabled.

21.5.18.4 Switching the Bend Relief Type

The bend relief type can be switched between 4 types:

			
Rectangular	Smooth	Round	Rip

Uwaga:

- A **Rip** relief has a very small width of 0.001 * thickness. If you need to remove the relief completely, change its width to 0. This operation removes both the bend relief geometry and the bend relief feature.
- **Smooth**, **Round** and **Rip** bend relief types can be applied only if the current bend relief type is **Rectangular**.

Using the ribbon, menu or toolbar

1 To choose the desired relief type, do one of the following:
- Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select **Switch to Rectangular**, **Switch to Smooth**, **Switch to Round** or **Switch to Rip**.
- Click a tool button on the **SM Relief** toolbar: **Switch to Rectangular Relief** (), **Switch to Smooth Relief** (), **Switch to Round Relief** () or **Switch to Rip Relief** ().
- Choose the relief type in the **Sheet Metal / Relief** menu.

  You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
- Press **Enter** to apply the selected relief type to the entire model.
- Click a neighboring flange or bend or a face of the bend relief.

  The bend relief is converted.

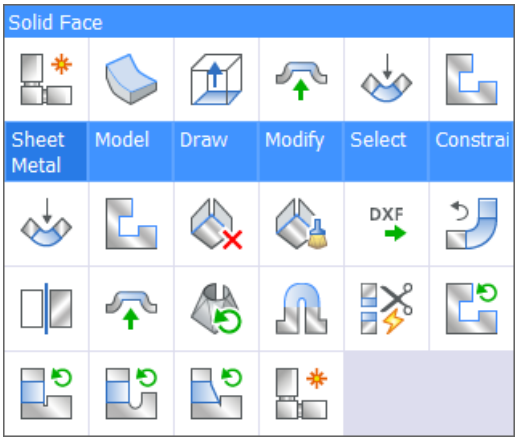
3 If the command has been called from the Ribbon, the command repeats the main prompt until you press the **ESC** key to stop.

Using the Quad

1 Hover over a neighboring flange or bend or a face of the bend relief.

If the **Select Faces** option of the **SELECTIONMODES** system variable is selected, the flange or face highlights; otherwise hold the **CTRL** key down when hovering until the face highlights.

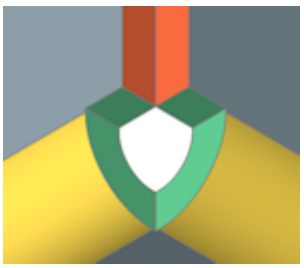
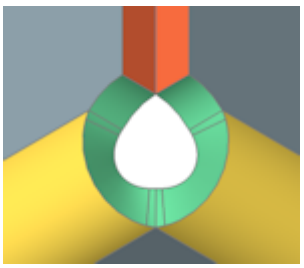
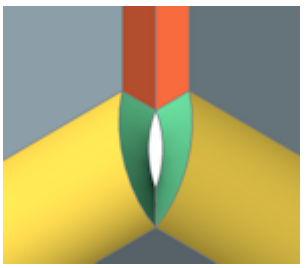
2 Select the desired relief type in the **Sheet Metal** tab in the **Quad** menu.



Rysunek 133: Sheet Metal

21.5.18.5 Switching the Corner Relief Type

The corner relief type can be switched between 3 types:

		
Rectangular	Circular	V-type

Using the ribbon, menu or toolbar:

1 To choose the desired relief type, do one of the following:
- Click the **Relief** drop-down button in **Sheet Metal / Modify** panel on the ribbon and select a relief type.
- Click a tool button on the **SM Relief** toolbar.
- Choose the relief type in the **Sheet Metal / Relief** menu.

  You are prompted: Select faces, 3d solids to switch reliefs to V-type [RECtangular/CIrcular/V-type/SMooth/RIp/ROund/relief EXtension/Entire model] <Entire model>:

2 Do one of the following:
- Press **Enter** to apply the selected relief type to the entire model.
- Click a neighboring flange or bend or a face of the corner relief.

  The corner relief is converted.

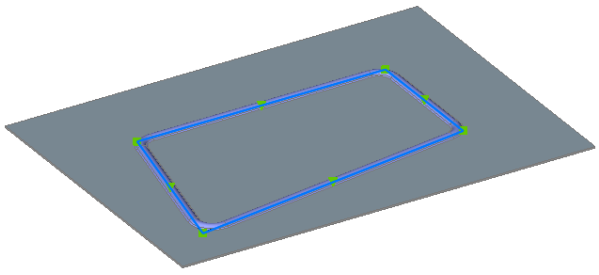
3 If the command has been called from the Ribbon, the command repeats the main prompt until you press the **ESC** key to stop.

Using the Quad:

1 Make sure the **Select Faces** option of the **SELECTIONMODES** system variable is selected.

2 Hover over a neighboring flange or bend or a face of the corner relief.

3 Select the desired relief type in the **Sheet Metal** command group in the **Quad** menu.



Rysunek 134: Sheet Metal

21.5.19 Working with Form and Rib Features

21.5.19.1 Overview

Form and Rib features are created by a forming tool. Rib features are produced by imprinting a wheel rolling over the flange following an arbitrary trajectory. We will use 'form' to refer to both form and rib features. Form features can be recognized in imported geometry or inserted from built-in or user-defined libraries. Form features can be edited directly or parametrically. Form features are listed in the **Mechanical Browser** with their parameters. Parameters of form features are displayed and can be edited in the **Properties Panel** when selected.

The following commands apply to form features:

Icon	Command	Description
	*SMFORM*	Convert a selected set of faces to a form feature. A drawing file with a user-defined form feature can be saved and then inserted with the *BMINSERT* command.
	*SMCONVERT*	Recognize form and rib features in imported geometry.
	*SMREPLACE*	Replace form features (including recognized ones) in sheet metal parts with form features from the built-in or user library.
	*SMRIBCREATE*	Create associative Rib form features on sheet metal parts from 2D profiles. The feature is updated automatically when the defining profile has been modified.
	*BMINSERT*	Insert an existing mechanical component or sheet metal form feature into the current drawing.
	*SMSELECT*	Select comparable form features. The command works in 2 modes: • **Similar**: Features which have the same set of faces, but possibly different sizes. • **Same**: Identical features.

A library of parametrized sheet metal form features is available from the Form Features tab on the **Tool Palettes** panel. The library contains the most commonly used form feature types. To insert a form feature to a sheet metal 3D solid, drag the form feature icon from the **Tool Palettes** panel or the **Components** panel onto a flange of the solid.

The *SMDEFAULTFORMFEATUREUNFOLDMODE* system variable controls the appearance of form features in 2D and 3D unfolded model representations.

21.5.19.2 To Insert a Form Feature

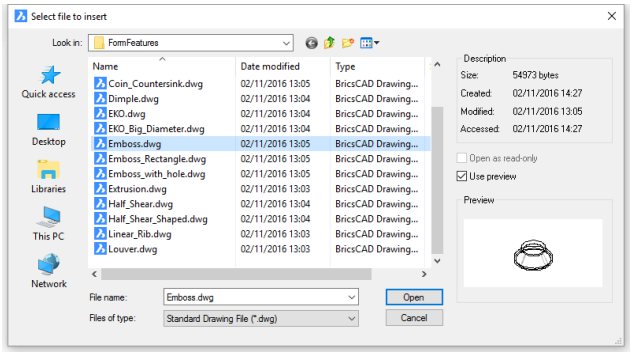
Form features are added to a sheet metal part by insertion on the existing flanges. The form features can be inserted by means of the *BMINSERT* command or by drag-and-drop from the **Form Features** tab of **Tool Palettes**.

1 Make sure **Dynamic Dimensions** (DYN) and **Dynamic UCS** (DUCS) are active.

2 Do one of the following:

- Click the **Form Feature** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Form Feature** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Insert Form Feature** in the **Sheet Metal** menu.
- Type *BMINSERT* in the Command line, then press **Enter**.

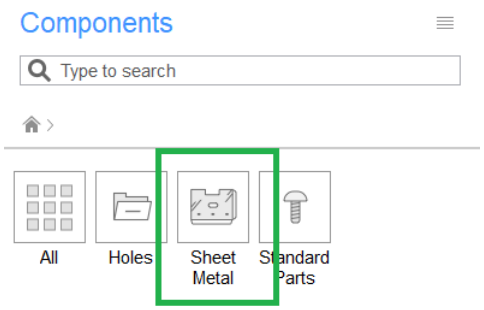
  The *Select file to insert* dialog box displays the **FormFeatures** folder [BricsCAD Installation folder]\Support\DesignLibrary\SheetMetal\FormFeatures:



Rysunek 135: Select file to insert

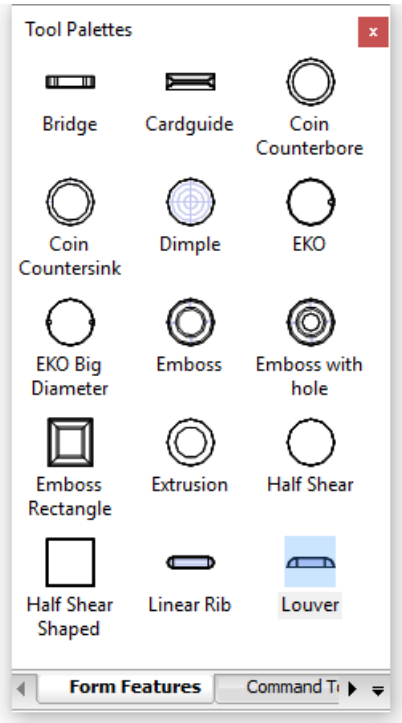
  Select the desired form feature, then double-click the feature or click **Open**.

- Select the form feature from Form Features tab of **Components** panel:



Rysunek 136: Components

- Select the form feature on the **Form Features** tab of the **Tool Palettes** panel:



Rysunek 137: Tool Palettes

3 Move the cursor over the flange face you want to insert the feature in.

The dynamic UCS aligns to the selected face.

The orientation of the UCS depends on the edge the cursor enters the face.

Dynamic dimensions display from the origin of the UCS to the insertion point of the form feature.

You are prompted:

Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple] <0, 0, 0>:

4 (option) Choose an option (see the *BMINSERT* command for more details):

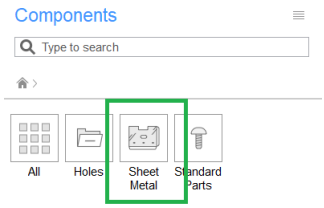
- **Edit**: Edit the parameters of the selected insert.
- **Rotate**: Define the rotation angle of the component.
- **Base Point**: Specify a different base point to insert the component.
- **Name**: Define the insert name of the component.
- **Insertion Type**: Specify the insertion type: Local or External.
- **Flip**: Insert the form feature in the opposite face of the flange.
- **Multiple**: Insert multiple inserts of the same component or create an array.

5 To insert a form feature do one of the following:

- Specify a point.
- Type a distance in the dynamic dimension fields. Hit the **TAB** key to go back and forth between the entry fields.

6 Repeat step 5 to insert another copy, or press **Enter** to stop.

In order to have Form Features in the **Components** panel, you must create or use the **Mechanical Profile** in the **Get Started** dialog box.



Rysunek 138: Components

21.5.19.3 Editing the Parameters of a Library Form Feature

Form features in the form feature library are parametric components. Several parameters control their dimensions. Parameters to control the Length, Width, Height and Radius of the form feature are correspondingly named L, W, H and R. The thickness of the form feature is set equal to the thickness of sheet metal part automatically.

The parameters of a single form feature can be edited either in the **Properties Panel** or in the **Mechanical Browser**. Parameters of multiple form features can be edited in the **Properties Panel** only.

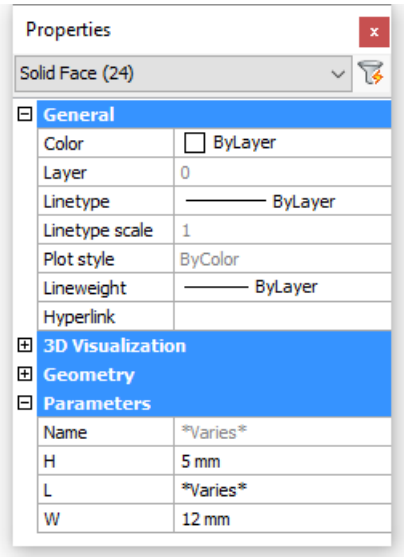
Methods to select form features

- Check the **Select Faces** option of the *SELECTIONMODES* system variable, then click the feature(s) in the model.
- Select the feature(s) in the **Mechanical Browser**, then press the **space bar** or right-click and choose **Select** in the context menu. Press and hold the **CTRL** key to select multiple form features.
- A sheet metal part can have lots of different form features, which might complicate operations such as deletion or replacement, especially if only a selection of features, that meet certain conditions, needs to be processed. To select form features more easily, the **SmSelect** command allows to select form features that are either identical to a selected feature or similar.

Editing parameters in the Properties Panel

1 Select the form feature(s).

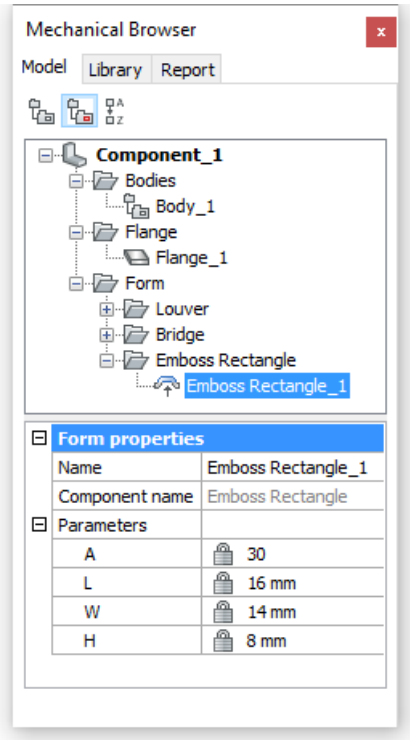
2 Select a parameter in the **Parameters** node in the **Properties Panel**.



Rysunek 139: Properties

Editing parameters in the Mechanical Browser

1 Select the form feature in **Form** node of the component structure tree.

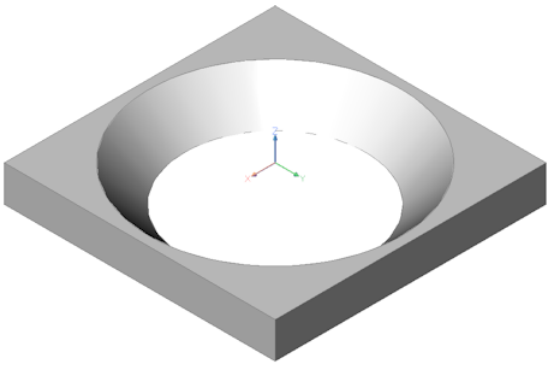


Rysunek 140: Mechanical Browser

2 Select a parameter in the lower panel of **Mechanical Browser** and type a new value.

21.5.19.4 Creating a custom form feature

1 Open a new drawing using **Mechanical-metric.dwt** as the template.



2 Create the 3D geometry of the form feature.

3 Run the *SMCONVERT* ( ) command.

4 Run the *BMMECH* ( ) command.

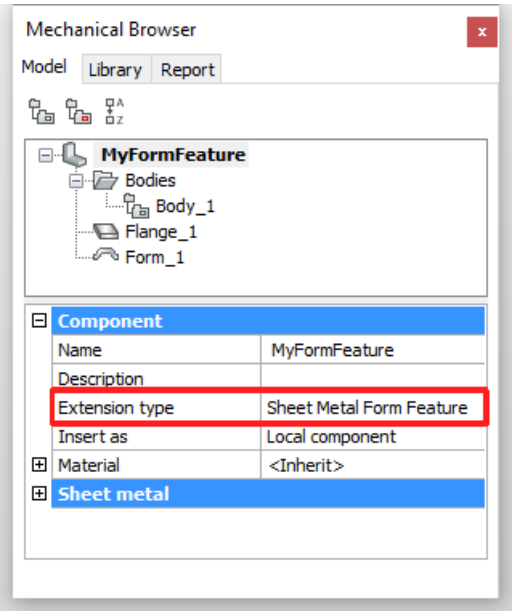
You are prompted: Name for mechanical component <Component_2>:

5 Type a name for the form feature.

You are prompted: Convert blocks and external references to mechanical components? [Yes/No] <Yes>:

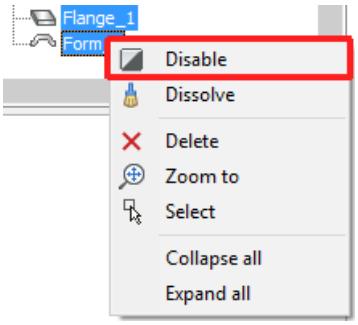
6 Select the **No** option.

7 In the **Extension type** field in **Mechanical Browser**, select **Sheet Metal Form Feature**.



Rysunek 141: Mechanical Browser

8 Select the Flange_1 and Form_1 features in the **Bodies** node, then right-click and select **Disable** in the context menu.

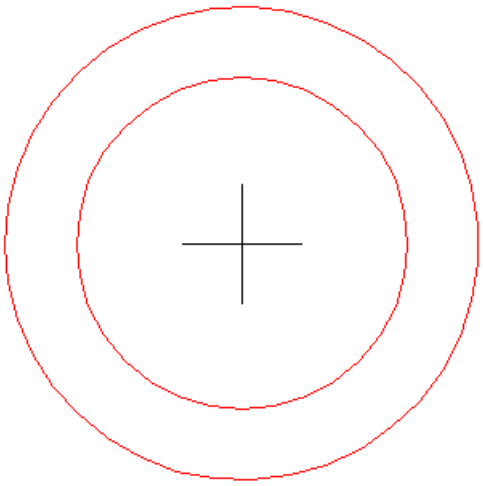


Rysunek 142: Diable

9 Create 2 new layers: UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN.

10 Turn layers 0 and UNFOLD_SYMBOL_DOWN **Off** and set layer UNFOLD_SYMBOL_UP **Current**.

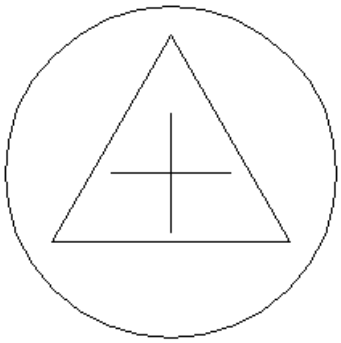
11 Create the unfold symbol, which will be placed on the face the form feature is placed



in.

12 Turn **Off** the UNFOLD_SYMBOL_UP layer and turn **On** the UNFOLD_SYMBOL_DOWN layer. Set the UNFOLD_SYMBOL_DOWN layer **Current**.

13 Create the unfold symbol, which will be placed on the opposite face the form feature is placed



in.

14 Turn **On** layer 0 and turn the UNFOLD_SYMBOL_UP and UNFOLD_SYMBOL_DOWN layers **Off**. Set layer 0 as **Current**.

15 Save the drawing.

21.5.19.5 Creating a custom parametric form feature

We will now parametrize the custom form feature we created in the previous procedure.

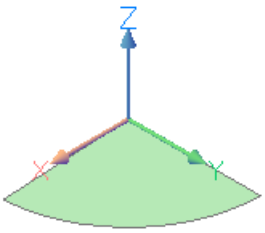
2 parameters are needed:

- **D** for a diameter of the bigger edge of the conical hole.
- **A** for an angle of the face of the conical hole.

In Sheet Metal, the default parameter **_SM_THICKNESS** controls the thickness of the sheet metal part.

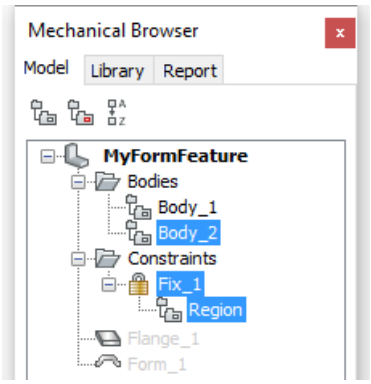
1 Create an auxiliary body for constraining on a new CONSTRUCTION layer.

Make this layer current and switch off all other layers.



2 Create a Region in the XY plane of the WCS on the CONSTRUCTION layer.

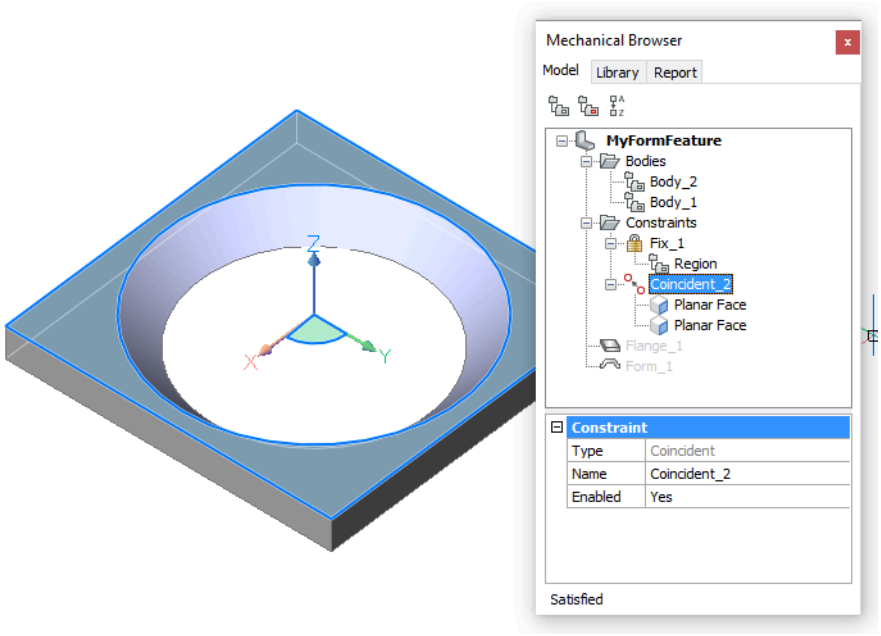
3 Apply a *DMFIX3D* constraint ( ) to the region.



Rysunek 143: Mechanical Browser

4 Switch **ON** layer 0.

5 Apply a *DMCOINCIDENT3D* constraint ( ) between the region face and the flange top



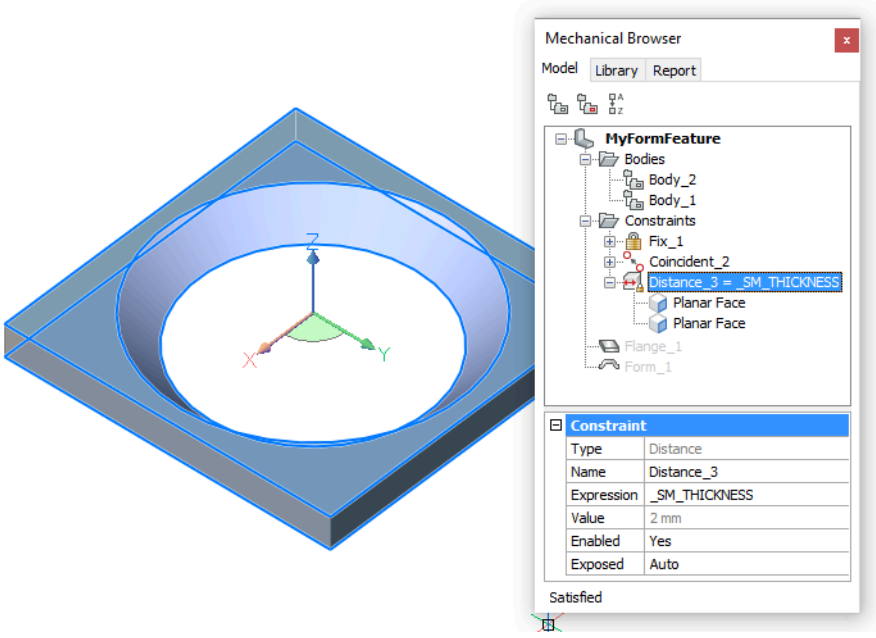
face.

6 Apply a *DMDISTANCE3D* constraint ( ) between the flange top and bottom faces.

First select the top face, then press the **TAB** key to select the bottom face.

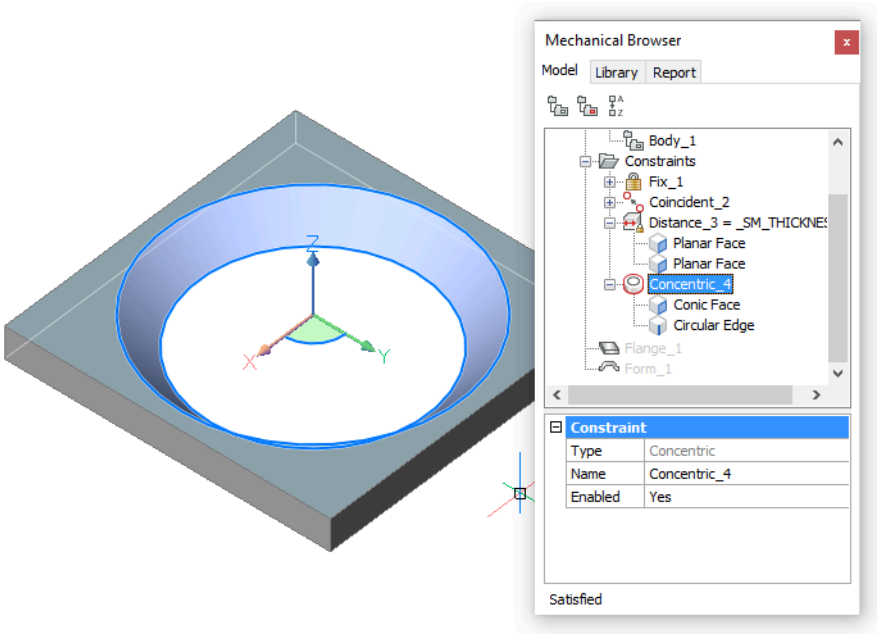
Accept the default value in the dynamic dimension.

7 Type _SM_THICKNESS in the **Expression** field of the Distance_3



constraint.

8 Since our Form Feature is conical we can modify the flange from parallelepiped to



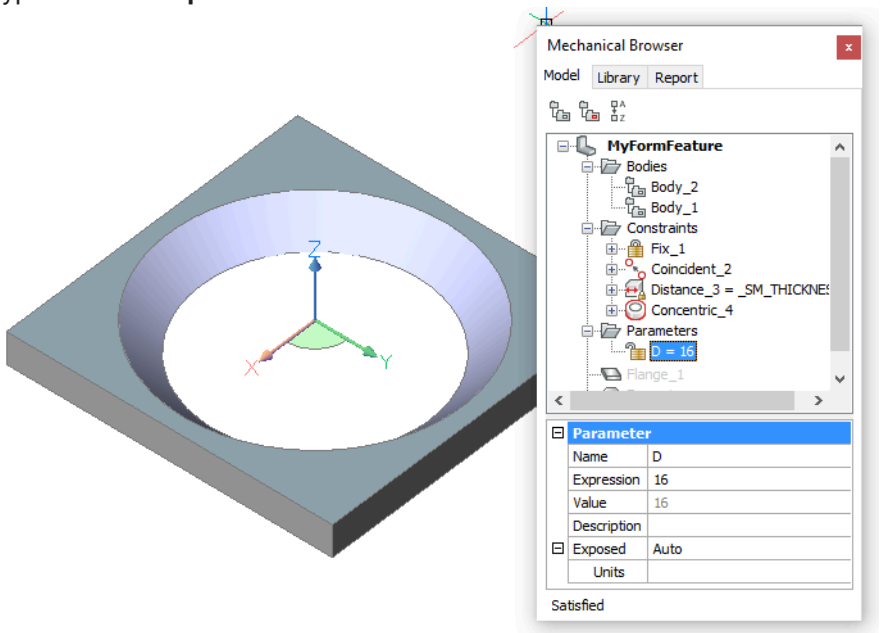
cylinder.

9 In the **Mechanical Browser** right-click the feature name and select **Add new parameter** in the context menu.

Rename new parameter to **D**.

Select mm for the **Units** field.

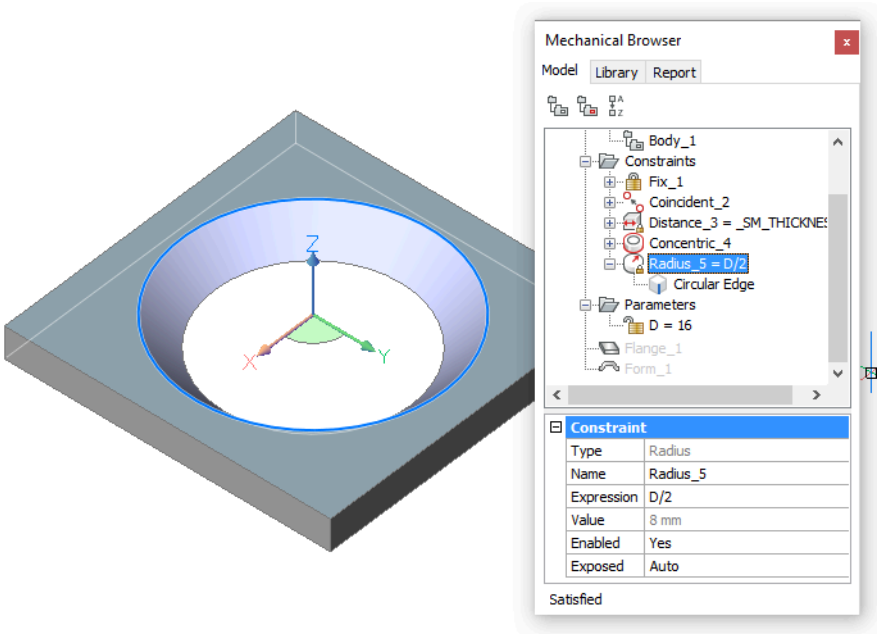
Type 16 in the **Expression** field.



10 Apply the *DMRADIUS3D* constraint ( ) to the top edge of the conical face.

Accept the default value in the dynamic dimension.

11 Type D/2 in the **Expression** field of the Radius_5



constraint.

12 In the **Mechanical Browser** right-click the feature name and select **Add new parameter** in the context menu.

Rename new parameter to **A**.
Leave the **Units** field empty.

Type 45 in the **Expression** field.

13 Do one of the following to create a **Cone Angle constraint** to the conical face:

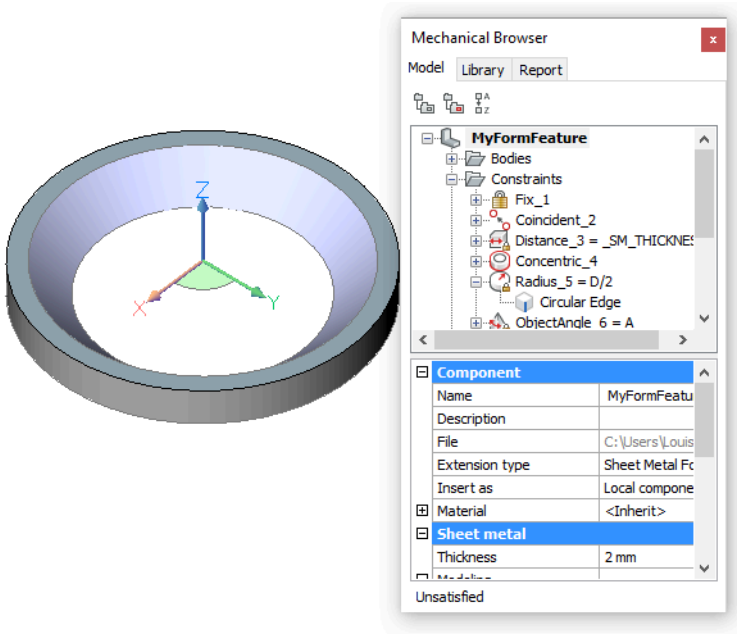
- Launch the *DMANGLE3D* command (), then choose the **Cone Angle constraint** option, then select the conical face.
- Hover over the conical face, then choose **Cone Angle constraint** () in the **Constraints** command group in the **Quad** menu.

Accept the default value.

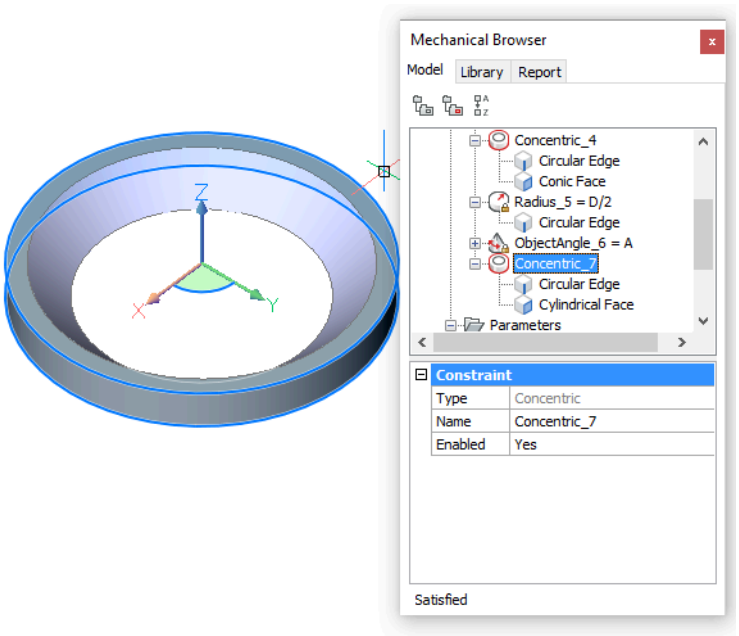
14 Type A in the **Expression** field of the ObjectAngle_6 constraint.

15 As our Form Feature is conical, modify the flange from parallelepiped to cylinder:

- Draw a circle on top of the flange. The radius of the circle is 9 mm.
- Select **Enable Boundary Detection** in the **Settings** panel on the **Home** tab in the Ribbon.
- Place the cursor over the area between the circle and the square edge of the flange, then choose **Extrude** () in the **Quad** menu and move the cursor downwards.



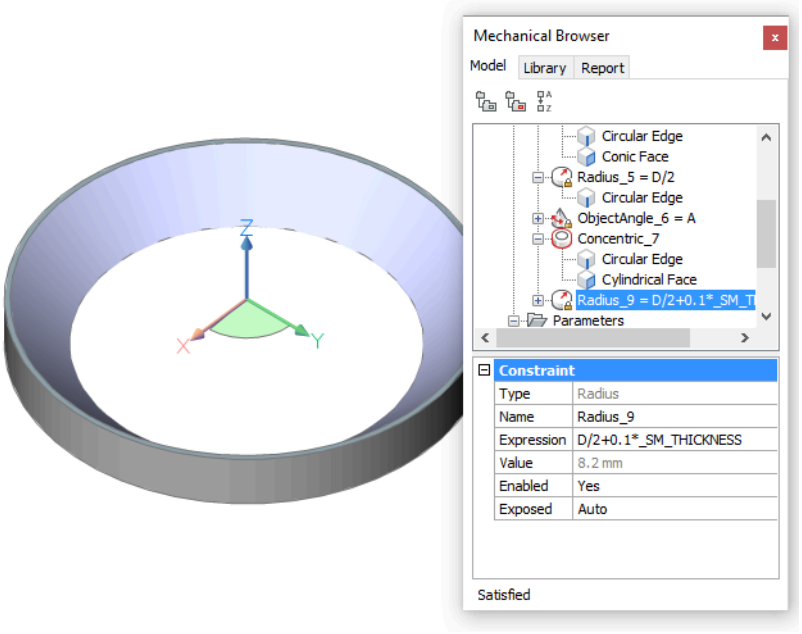
16 Apply the *DMCONCENTRIC3D* constraint ( ) between the circular edge of the region and the cylindrical face.



17 Apply the *DMRADIUS3D* constraint ( ) to the cylindrical face.

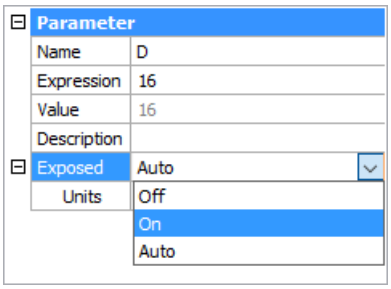
Accept the default value in the dynamic dimension.

18 Type D/2+0.1*_SM_THICKNESS in the **Expression** field of the Radius_8



constraint.

19 Set the **Exposed** field of both parameter A and D **On**.



Rysunek 144: Parameter

Now create 2D constraints for the unfold symbols.

20 Make layer UNFOLD_SYMBOL_UP **current** and switch **Off** all other layers.

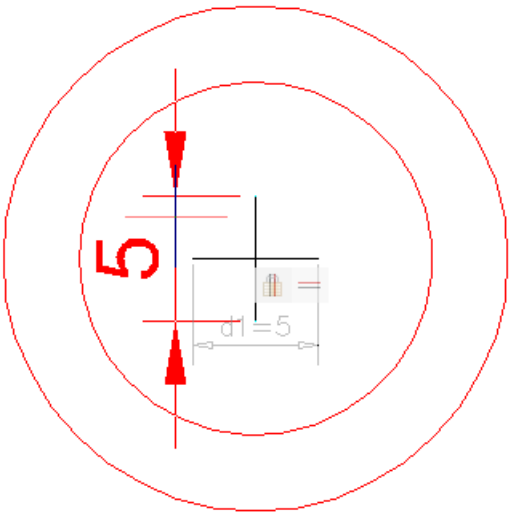
21 Apply the *GCFIX* constraint ( ) to both crossing lines indicating the center of the circles.

22 Apply the *GCHORIZONTAL* constraint ( ) to the horizontal center line.

23 Apply the *GCVERTICAL* constraint ( ) to the vertical center line.

24 Apply the *GCCONCENTRIC* constraint ( ) between the 2 circles.

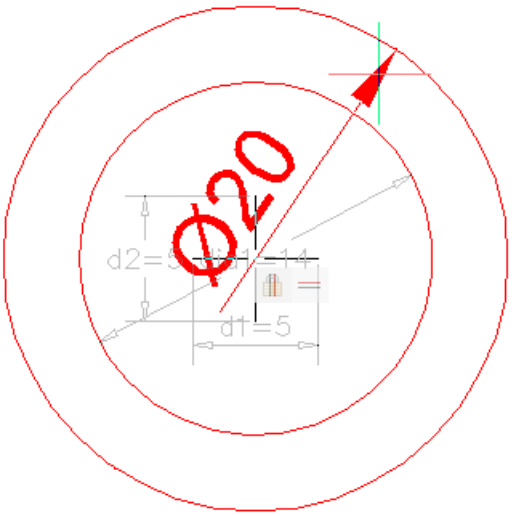
25 Apply the *DCLINEAR* constraint ( ) between the endpoints of the center lines. Accept the default



value.

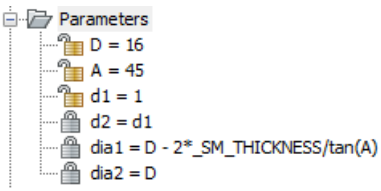
26 Apply the *DCDIAMETER* constraint ( ) to both circles.

Accept the default value for dia1 (=inner circle) and dia2 (= outer circle).



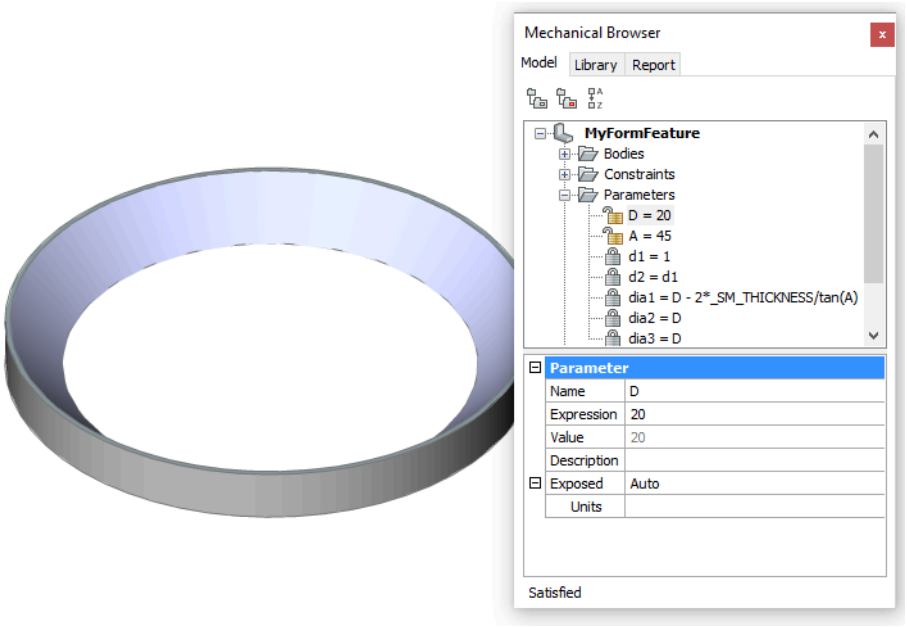
27 Adjust the **Expression** fields of the 2D constraints as follows:

- d1 = 1
- d2 = d1
- dia 1 = D - 2*_SM_THICKNESS/tan(A)
- dia 2 = D



Rysunek 145: Parameters

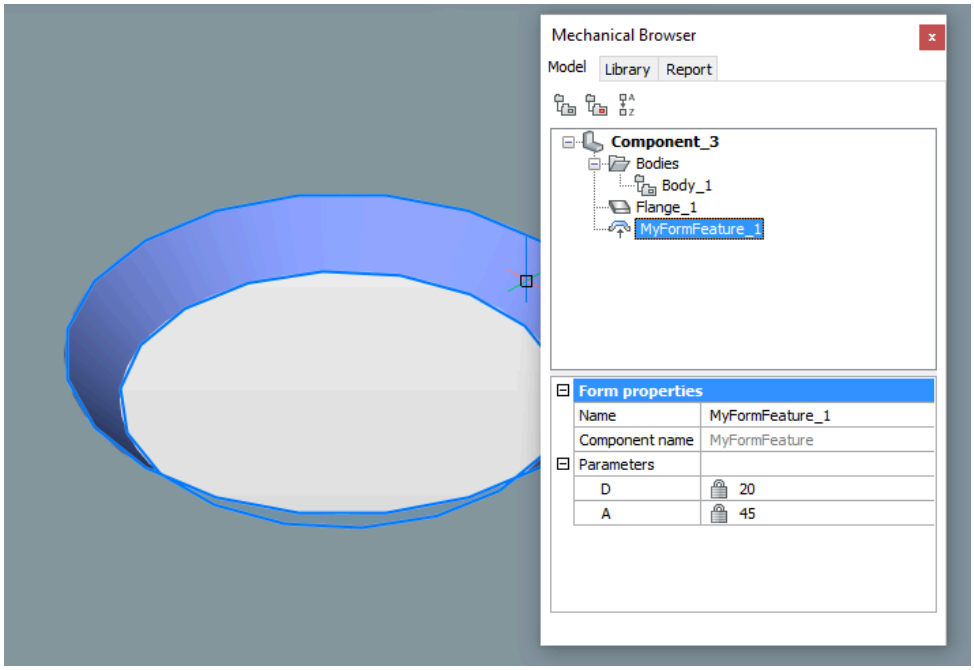
28 Switch **Off** the display of all layers, but layer 0, then save the



file.

The form feature can now be used to make conical parametric openings in a sheet metal flange.

Use the **Insert Form Feature** tool (), which launches the *BMINSERT* command.



Optionally: Add the form feature as a component to a tool palette.

21.5.19.6 To create a rib feature

Rib features are created by the *SMRIBCREATE* command from a 2D curve on flange face.

1 Do one of the following:

- Click the **Create Rib Feature** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Create Rib Feature** tool button ( ) on the **Modify** panel of the **Sheet Metal** tab on the ribbon.
- Type *SMRIBCREATE* in the Command line, then press **Enter**.

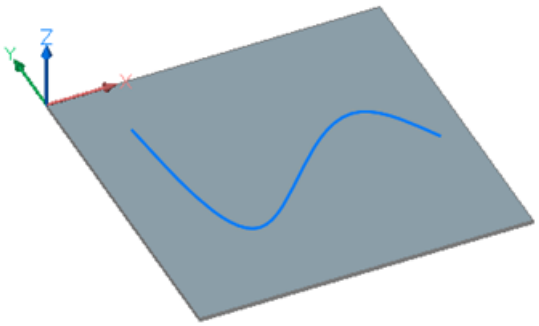
  You are prompted: Select flange face:

2 Select the flange face.

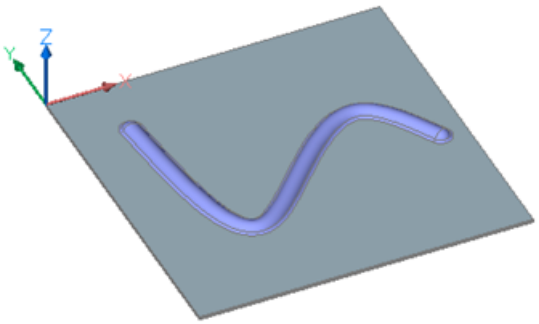
  You are prompted: Select curve or [Profile radius/Round radius]:

3 Select a curve on the selected flange face.

  Lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines are accepted. A 2D curve can be open or closed, but can not intersect itself or other form features.



After selection of the curve, a rib feature is created:



21.5.19.7 To modify a rib feature

To modify a rib feature:

- Modify the generatrix curve using its grip points.
- Edit the **Profile** and **Round** radius in the **Mechanical Browser**.

Modify the generatrix curve

1 Select the curve.

Because the curve is hidden by the rib feature, press the **TAB** key to select obscured entities, or select the curve in the **Structure** panel.

2 Grip edit the curve.

The Rib feature updates according to the changes of the curve.

3 After selecting the flange face, you are prompted:

Select curve or [Profile radius/Round radius]:

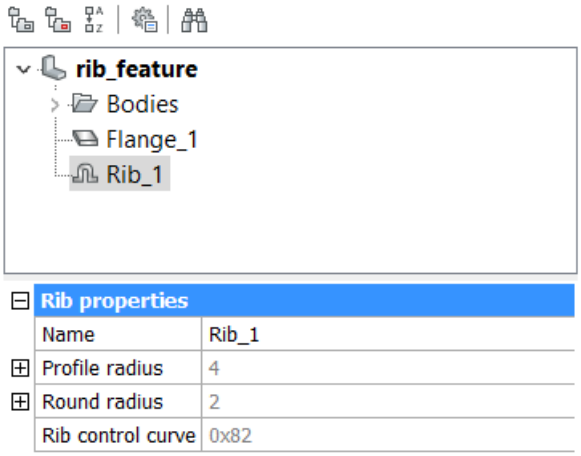
4 Select curve on selected flange side.

Edit the properties in the Mechanical Browser

1 Select the rib feature in the **Mechanical Browser**.

The rib feature highlights in the model.

2 Select the property in the **Rib Properties** grid.



Rysunek 146: Rib properties

3 Type a value.

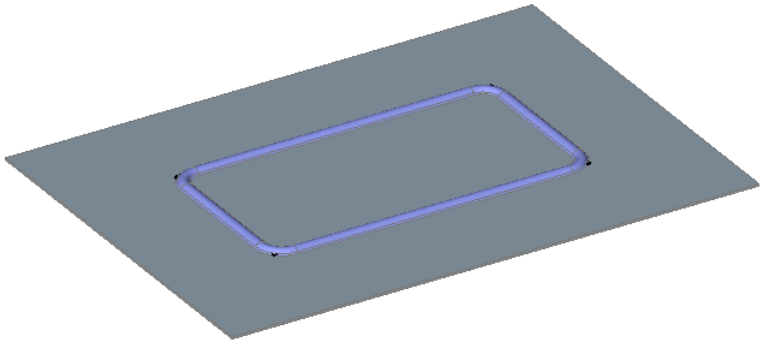
The selected rib feature is updated.

21.5.19.8 Understanding the difference between Control and Trajectory curves

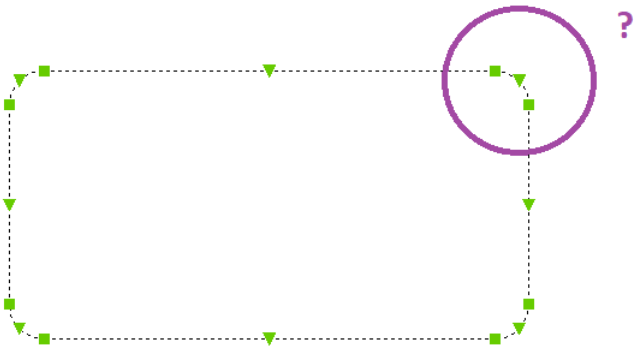
When creating a rectangular rib feature, we first draw a rectangle which is then used to create a rib feature by the *SMRIBCREATE* command. When this control curve is grip edited, the rib feature is updated accordingly.



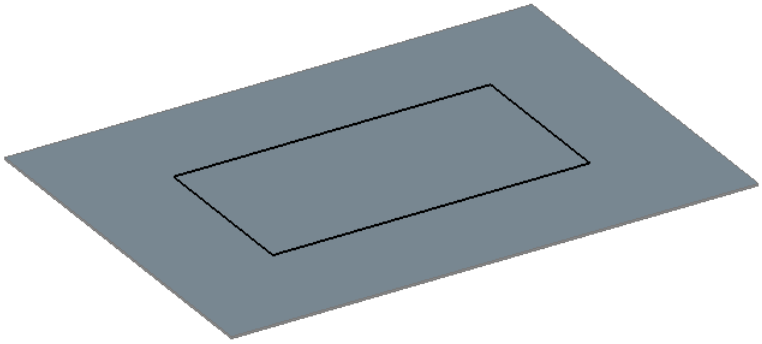
On the production side, the rectangular control curve is replaced by a filleted trajectory curve, according to the physical limitations of the wheel tool:



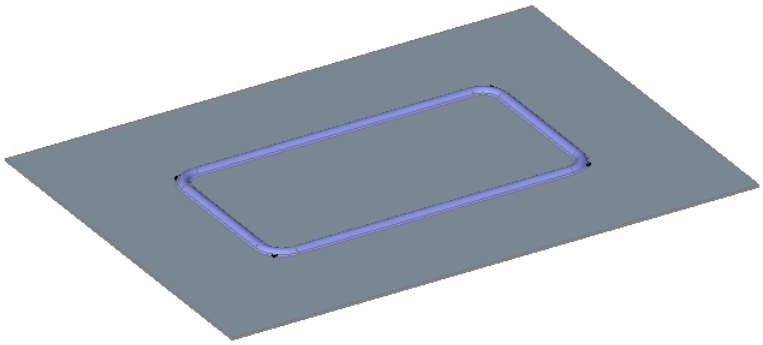
The trajectory curve, however correct, is hard to manipulate, for example when you want to move a corner.



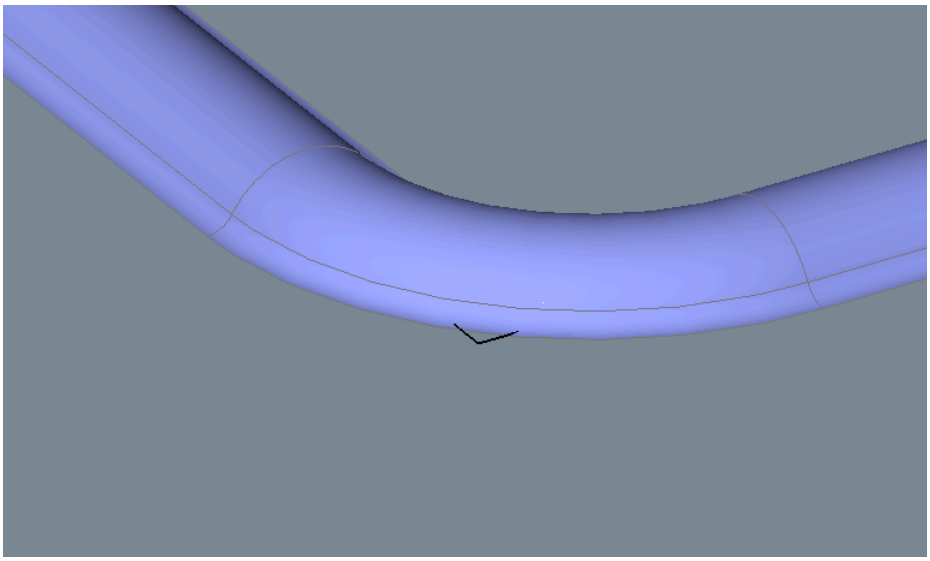
In the design stage we use the **Control Curve**:



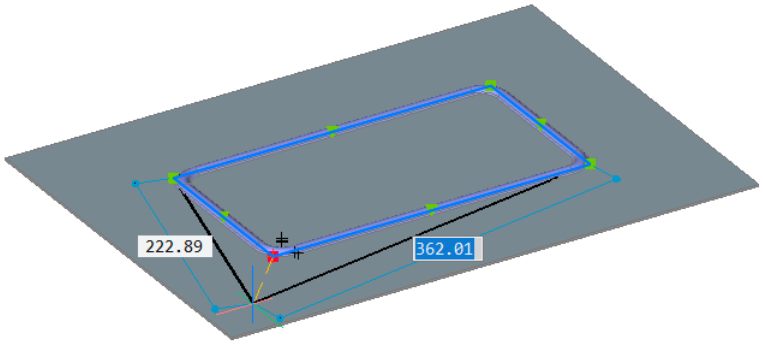
When creating the rib feature, *SMRIBCREATE* adds fillets according to the value of the **Rib fillet radius** parameter in **Mechanical Browser**. This parameter is not controlled per feature, since it relates to the properties of the physical wheel tool:



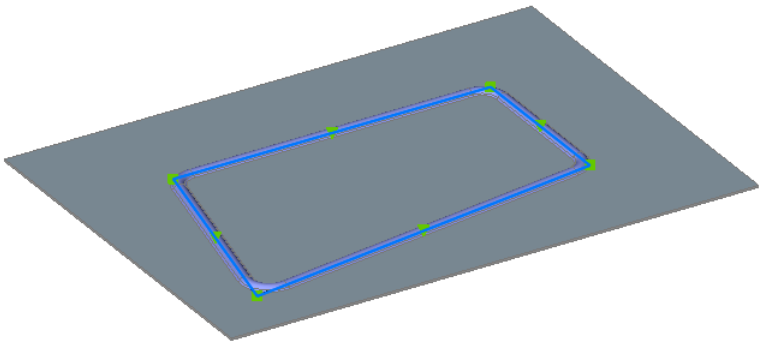
When zooming in closely, you notice the difference between the 3D geometry of the rib feature and the control curve:



When the Polyline is grip edited ...

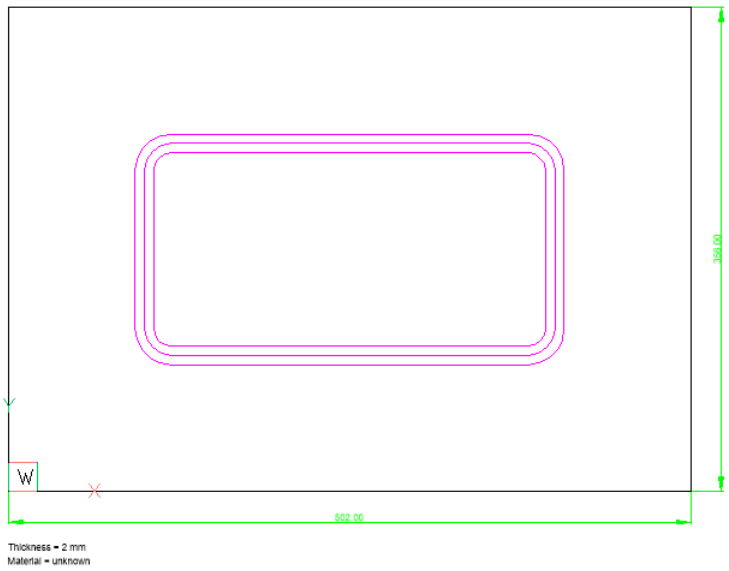


... the rib feature is adjusted accordingly:



This illustrates the associativity between the 2D curve and 3D rib feature. The recognition of rib features in *SMCONVERT* is controlled by the *SmConvertRecognizeRibControlCurves* user preference.

When the part is unfolded by the *SMUNFOLD* command, the curves follow the 3D design and have the fillets. The middle curve is called **Trajectory Curve**:

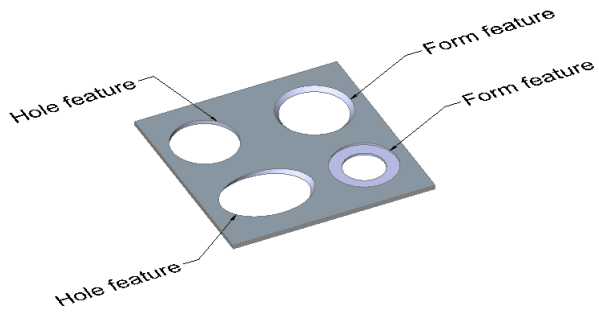


21.5.20 Working with Hole Features

21.5.20.1 Overview

Hole features are a simplified version of Form Features. In the image below you can see different features, which have only thickness faces (the *SMCONVERT* command respects these rules):

- Orthogonal holes are hole features.
- Simple non-orthogonal holes are hole features.
- Chamfered orthogonal holes are form features.
- If the hole thickness face consists of 2 different regions, it is a form feature.



Hole features are recognized by the *SMCONVERT* command if the *SMCONVERTRECOGNIZEHOLES* user preference is ON.

The following basic operations are allowed for hole features:

- *SMDELETE*, *SMDISSOLVE*, disable and *SMSELECT* them using commands or in the **Mechanical Browser for Sheet Metal**.
- **Form Feature Modes** in the *SMUNFOLD* command, including **Symbol** mode.

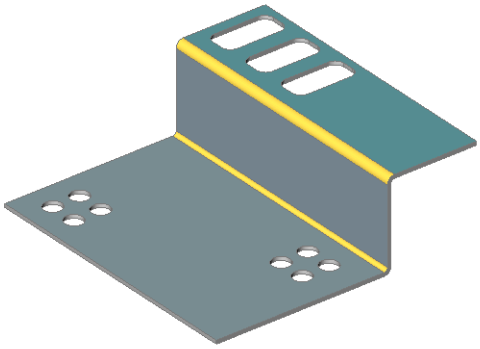
- Associate your own 2D geometry in a flattened layout.
- Hole features are preserved by the *SMREPAIR* command. Plain hole faces become orthogonal. A hole feature of the same geometry is not affected by the *SMREPAIR* command.

21.5.20.2 Recognition of Arrays of Hole Features

When hole features exist in a body, the *SMPARAMETRIZE* command detects rectangular arrays of holes on flanges, according to the following rules:

- Holes belong to the same flange.
- Holes are on a rectangular grid.
- There are no gaps (missing elements) in the array.
- Holes are equally oriented.
- The minimal array size is either 1x3 or 2x2.

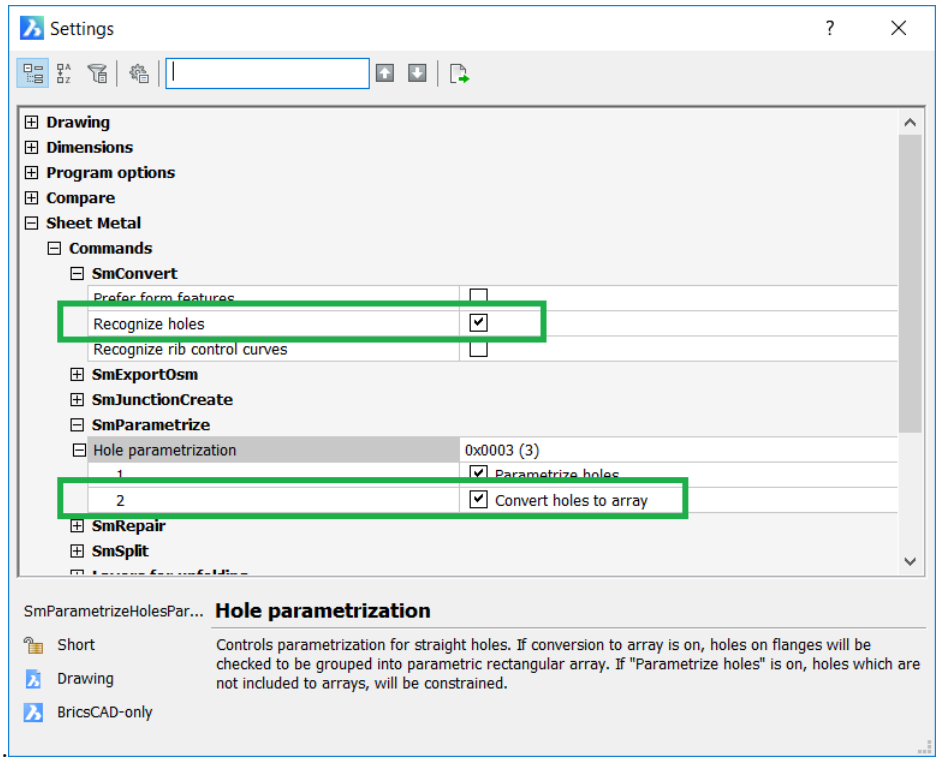
Let us illustrate the workflow on a part with Flange and Bend features.



1 In the *Settings* dialog box, go to **Sheet Metal/Commands**.

2 Enable the **Recognize holes** option for the *SMCONVERT* command.

3 Enable the **Convert holes to array** and **Parametrize holes** options for the *SMPARAMETRIZE*



command.

4 Run the *SMCONVERT* command.

5 Do one of the following:

- Click the **Convert to Sheet Metal** tool button () on the **Sheet Metal** toolbar.
- Click the **Convert to Sheet Metal** tool button () on the **Sheet Metal** ribbon bar.
- Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

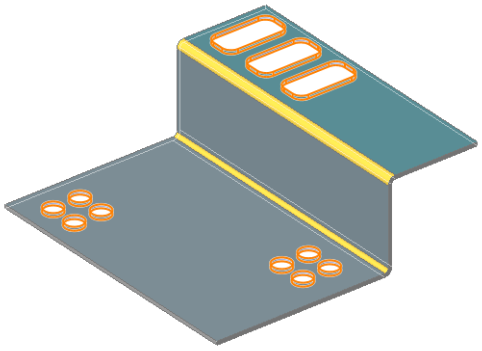
  You are prompted: Select 3D solids or [Entire model] <Entire model>:

6 Select the 3D solid to convert.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

7 Press **Enter** to convert the 3D solid to sheet metal part.

  11 hole features are recognized on the part.



8 Run the *SMPARAMETRIZE* command.

9 Do one of the following:

- Click the **Parametrize** tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Parametrize** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Parametrize** in the **Sheet Metal** menu.

  You are prompted: Select 3D solids to parametrize or [Entire model] <Entire model>:

10 Press **Enter** to process the sheet metal part.

The command reports must look like the one below:

Created 4 distance constraints

Created 3 fix constraints

Created 0 coincident constraints

Created 0 tangent constraints

Created 0 rigid set constraints

Created rectangular array 2x2 basing on Hole_3 feature

Created rectangular array 2x2 basing on Hole_7 feature

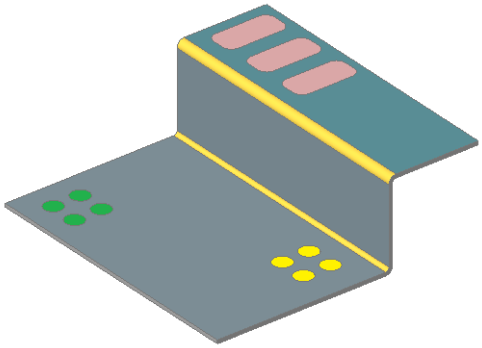
Created rectangular array 3x1 basing on Hole_9 feature

Total: 7 constraints and 3 array(s) created

Uwaga: Feature names and constraint distribution may differ depending on the BricsCAD version.

To examine the arrays, you can temporarily unfreeze the **BC_SUBTRACT** layer.

In the image below the arrays are colored manually.

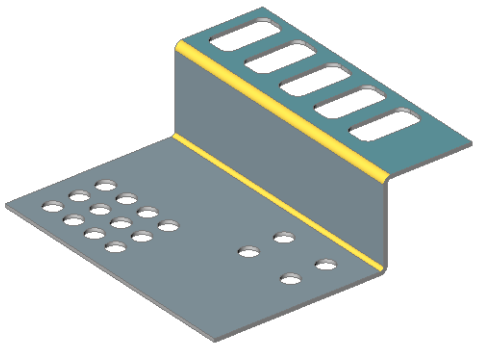


Uwaga: The green and yellow arrays are not unified. Otherwise the array would have missing holes.

11 Edit the **Array Properties** in the **Mechanical Browser**.

In the image below the following has been modified:

- Green array: number of rows and columns from 2x2 to 3x4.
- Yellow array: X and Y spacing.
- Red array: number of rows and columns from 3x1 to 5x1.



21.5.21 Converting to Sheet Metal and Repairing

Use the *SMCONVERT* and *SMREPAIR* commands.

21.5.21.1 To convert 3D geometry to a valid Sheet Metal part

A sheet metal part can be created from any 3D solid geometry, whether created in BricsCAD or imported from another CAD system. However, the geometry might not be a valid sheet metal model: missing bends and reliefs or thickness faces which are not orthogonal to the flange faces. BricsCAD Sheet Metal contains a dedicated toolset to convert such geometry to a valid sheet metal part and add all missing sheet metal features.

1 Do one of the following:

- Click the **Convert to Sheet Metal**tool button ( ) on the **Sheet Metal** toolbar.
- Click the **Convert to Sheet Metal** tool button ( ) on the **Sheet Metal** ribbon bar.
- Choose **Convert to Sheet Metal** in the **Sheet Metal** menu.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

2 Select the 3D solid to convert.

  You are prompted: Select 3D solids or [Entire model] <Entire model>:

3 Press **Enter** to convert the 3D solid to a sheet metal part.

The properties of the sheet metal part display in the **Mechanical Browser for Sheet Metal**.

Uwaga: Converted bodies can be unfolded or exported to .DXF or .OSM file formats for processing by a CAM system.

In some cases it might be necessary to fix converted bodies before unfolding/exporting:

- Create missing reliefs: *SMRELIEFCREATE* command.
- Convert hard edges to bends or junctions: *SMBENDCREATE* and *SMJUNCTIONCREATE* commands.
- Split flanges and lofted bends: *SMSPLIT* command.
- Repair sheet metal bodies: *SMREPAIR* command.

21.5.21.2 To repair a Sheet Metal part

A sheet metal part may have inaccuracies in geometry and topology or sheet metal design issues. This is often the case for imported parts which are designed in geometrical kernels different from ACIS.

Examples of such problems are:

- Thickness faces, which are not perpendicular to the flange/bend faces.
- Lofted bends are not tangential to the adjacent flanges/bends.
- Wrong bends.
- General ACIS issues.

Such problems can prevent editing operations from being executed successfully.

The recommended workflow to fix such problems is:

- Run the *DMAUDIT* command to fix general modeling issues.
- Run the *SMREPAIR* command.

21.5.21.3 To repair a Sheet Metal body

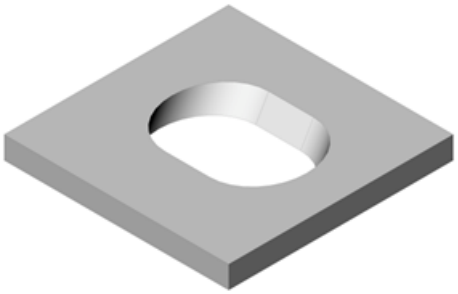
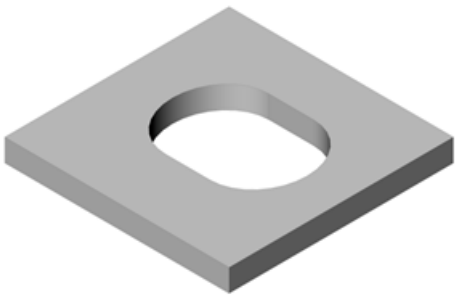
Sheet metal bodies with thickness faces, which are not perpendicular to the flange/bend faces, can not be correctly manufactured using traditional tools.

1 Do one of the following:

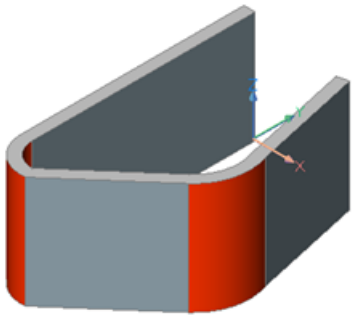
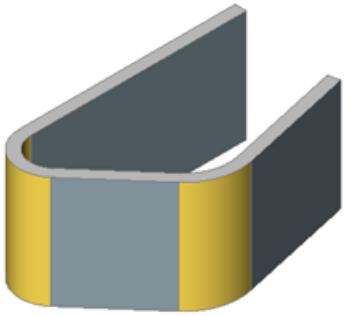
- Click the **Repair Sheet Metal** tool ( ) on the Heal panel of the Sheet Metal ribbon tab;
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar;
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu;
- Type *SMREPAIR* in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/ Scan for lost details]:

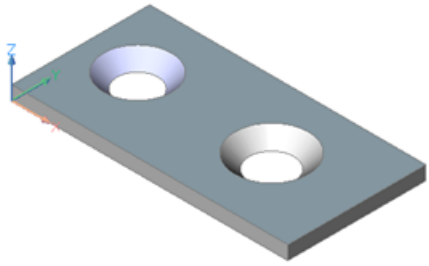
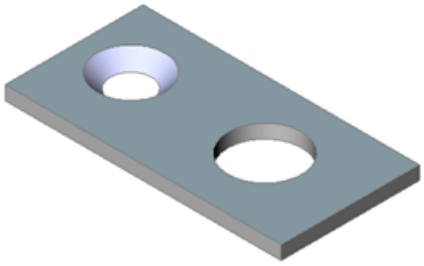
2 Select the face to be repaired.

	
The same model before ...	... and after applying a repair operation:

When the model has wrong bends *SMREPAIR* transforms them to regular bends:

	
Wrong bends, which have a non-tangential connection with the flanges.	Correct bends after repairing.

Form features are translated to the new repaired model:

	
Model before repairing.	The form feature has not been changed, while a hole, which is not a form feature, is orthogonalized.

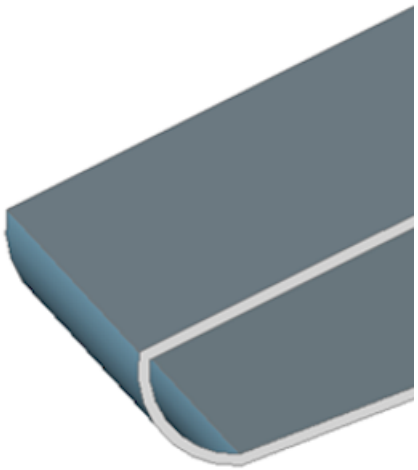
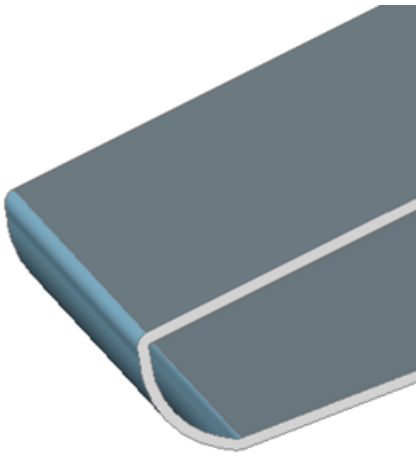
21.5.21.4 To repair a Sheet Metal body with lofted bends

1 Do one of the following:

- Click the **Repair Sheet Metal** tool ( ) on the **Heal** panel of the **Sheet Metal** ribbon tab.
- Click the **Repair Sheet Metal** tool ( ) on the **Sheet Metal** toolbar.
- Choose **Repair Sheet Metal** in the **Sheet Metal** menu.
- Type *SMREPAIR* in the Command line.

  You are prompted: Select flange face or lofted bend face to repair or [switch on Local repairing/ Scan for lost details]:

2 Select a face of a flange or a lofted bend. The selected face provides the surface to be thickened.

Before	After
	
The lofted bend is not tangent to the flange. Unfolding is impossible.	Notice the difference in the connection of the blue lofted bend with the top flange, which is smooth now. Unfolding is possible.

Option: If the **SmRepairLoftedBendMerge** user preference is ON, adjacent lofted bends with a tangential connection are merged to a single lofted bend.

21.5.22 Unfolding Sheet Metal Parts and Exporting to CAM

21.5.22.1 Overview

Use the *SMUNFOLD* command.

Sheet metal parts can be designed in many different ways. To manufacture the sheet metal part, a flat pattern with bend annotations is needed, which is then used in a CAM system to program CNC machines, such as laser cutters and press brakes. This flattening operation is called unfolding.

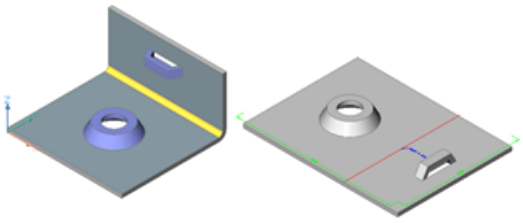
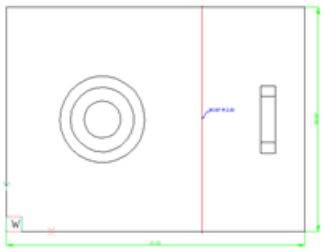
21.5.22.2 Unfolding a sheet metal body

The *SMUNFOLD* command unfolds sheet metal parts. BricsCAD automatically creates a solid body corresponding to the flat metal sheet needed to manufacture the sheet metal part using bending techniques. This sheet is placed on the XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in Z axis, run the *SMUNFOLD* command again and select the opposite face of the initial body.

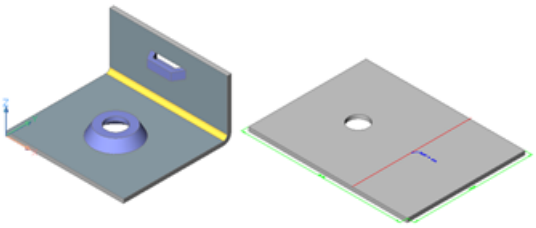
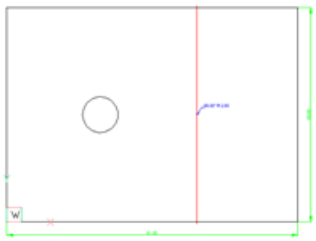
Form features are unfolded according to the **Unfolding Mode** setting in the **Mechanical Browser**.

The options are:

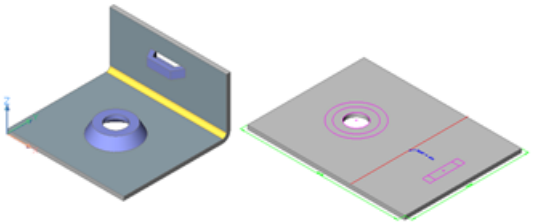
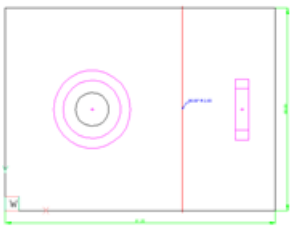
- **Keep** (0): allows to keep form features geometry on the unfolded representation.

3D model and unfolded body	2D
	

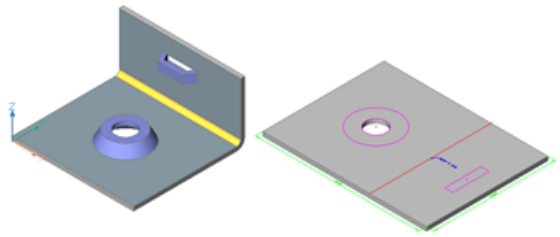
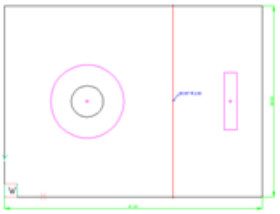
- **Remove** (1): removes the form feature geometry from the unfolded representation.

3D model and unfolded body	2D
	

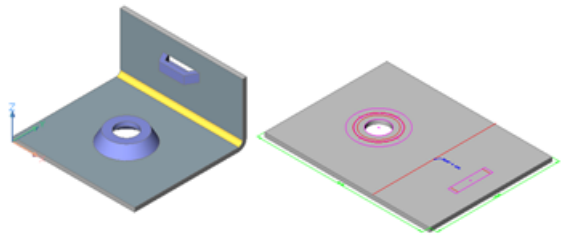
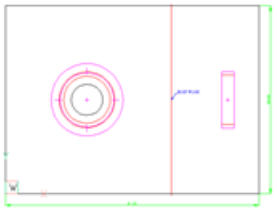
- **Project** (2): generates a form feature symbol by projecting the form feature edges onto the flange plane.

3D model and unfolded body	2D
	

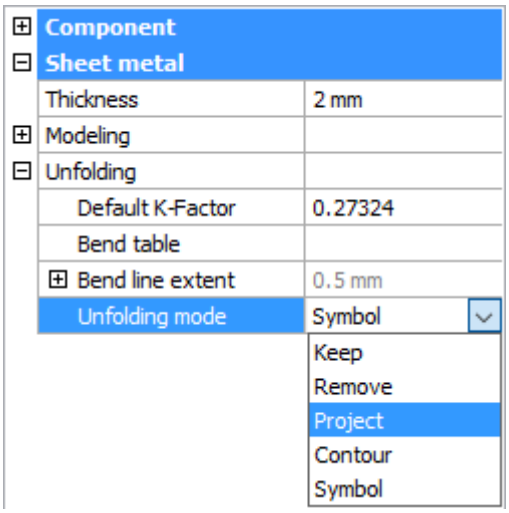
- **Contour** (3): generates a form feature symbol by projecting the external contour and holes of the form feature onto the flange plane.

3D model and unfolded body	2D
	

- **Symbol** (4): creates the unfold symbol on the **UNFOLD_SYMBOL_UP** or **UNFOLD_SYMBOL_DOWN** layer depending on the direction of the form feature.

3D model and unfolded body	2D
	

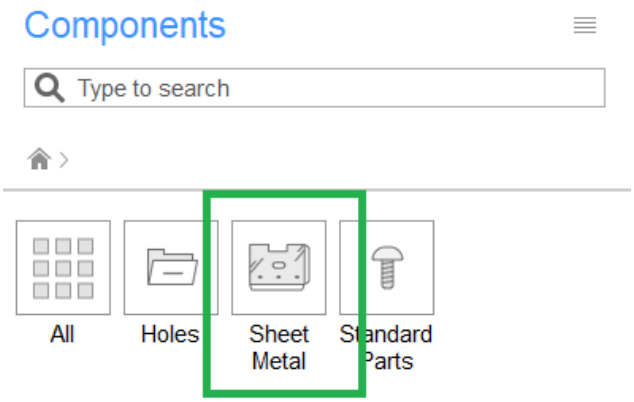
The unfold mode for the current sheet metal part can be set in **Mechanical Browser**.



Rysunek 147: Unfolding mode

Lofted bends lying on cylindrical and conical surfaces can be unfolded analytically: the dimensions and shape of the unfolded lofted bend will be as if its middle surface was mathematically unrolled on the

plane. This shape can also be achieved by a highly increased number of samples for the lofted bend. You must set the **Unfolding mode** individually for each lofted bend feature. If analytical unfolding is allowed, the option will be present in a drop down list:



Rysunek 148: Analytical

21.5.22.3 To unfold a sheet metal body

1 Launch the *SMUNFOLD* command.

You are prompted: Select a flange or lofted bend face to start unfolding [Associative]:

2 Select a face on sheet metal part.

You are prompted: Select position of the unfolded body:

3 Specify a point.

You are prompted: Validate the unfolded body and select an option [save 2D geometry/save 3D geometry/Optimize bend annotations/Keep] <Keep>

4 Do one of the following:

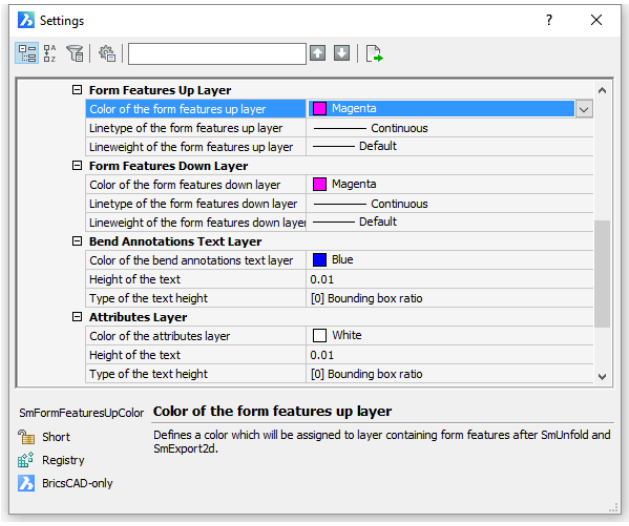
- Save the unfolded 3D solid in a separate drawing file.
- Convert the unfolded 3D solid to a 2D drawing file (*.DWG or *.DXF).
- Rearrange the bend annotations.
- Keep the block with the unfolded part in the sheet metal part model.

The *SMUNFOLD* command considers the deformation of the sheet metal material during bending. When a flat sheet of metal is bent into a 3D part with a bending tool (for example a press brake), the material is plastically deformed, which means it is compressed at the inside of the bend and stretched at the outside. As a result, the length of the part measured along its surface is different in the flat and bend states. BricsCAD can automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of **K-Factor** parameter or by attaching a Bend Table.

21.5.22.4 Managing layer properties of the 2D geometry

The 2D geometry generated by the *SMUNFOLD*, *SMEXPORT2D*, and *SMASSEMBLYEXPORT* commands has different colors for contours, bend lines up, bend lines down, overall dimensions, form features up, form features down, bend annotations and attributes.

You can set the colors differently in the **Sheet Metal/Commands/Layers for unfolding** settings group on the *Settings* dialog box. Each entity type is placed on a dedicated layer of which the color is controlled through a series of user preferences:



Rysunek 149: Settings

If the 2D geometry of a form feature was taken from a library component, its properties are controlled by the properties of this component at the moment of insertion. There are 2 other modes of **Unfolding Mode** which lead to generation of 2D geometry for form features: **Project** or **Contour**. In these modes 2D geometry is generated automatically from 3D geometry and its appearance is controlled by **Form Features Up Layer** and **Form Features Down Layer** settings groups.

21.5.22.5 To export a sheet metal body

Sheet metal parts created or edited in BricsCAD can be processed by different CAM systems (such as JETCAM or CADMAN-B) to generate NC code for cutting and bending machines.

The user preference controls the **save 2D geometry** option of the *SMUNFOLD* command to customize the .DXF file to be compatible with the list of popular CAM systems.

The *SMEXPORT2D* command exports the unfolded representation of a sheet metal body as a 2D profile in *.DXF or *.DWG file format.

The *SMEXPORTOSM* command exports a sheet metal solid to the *.OSM (Open Sheet Metal) file format (native for the CADMAN-B CAM system).

21.5.22.6 To export an assembly with sheet metal parts to .DXF

The *SMASSEMBLYEXPORT* command searches the assembly tree for sheet metal design and exports it to a *.DXF file. As a result, a set of *.DXF files, sorted by thickness in different subfolders is created

in the output folder. The command generates a HTML report which allows to control the output. Most of the sheet metal parts will be processed automatically. Poor sheet metal parts will be reported by the command to be reworked manually.

21.5.23 Associative Unfolded Representations of 3D Solids

21.5.23.1 Overview

BricsCAD offers bidirectional associativity between the 3D Sheet Metal part and its flattened appearance. In this mode both representations exist in separate documents: the Original part is stored in its native document and the Flattened part is stored in a temporary document. Associativity is temporary and exists while both documents are opened. In order to work in associativity mode in the next BricsCAD session or once the document was closed, you must repeat the steps again.

21.5.23.2 What does associativity mean?

Associative Unfolded allows you to make cuts on flattened geometry and propagates the changes back to the 3D model. This helps simplify 2 scenarios:

- Creating complex, non-standard Bends and Corner Relief cuts.
- Avoid self-intersections in unfolding.

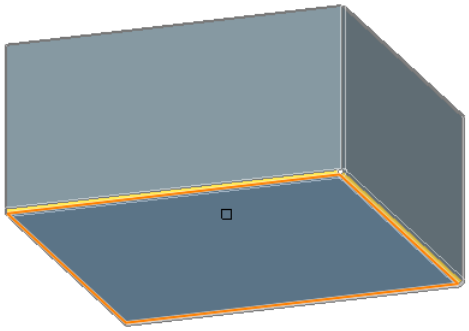
In these 2 scenarios both are easy to do in 2D rather than in 3D. In flattened mode Sheet Metal part is represented as set of separate flat solids (per-feature). Use the *SMDELETE* command to delete solid flanges and bends on flat patterns.

Associativity is bidirectional; changes can be propagated from flattened parts back to 3D models and vice versa. Changes made on one of the representations are propagated to the document with other representations once it becomes active.

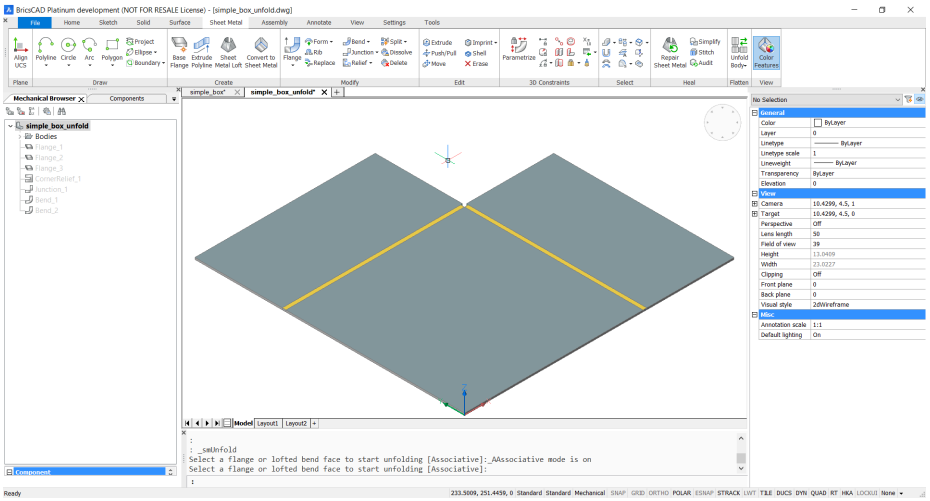
21.5.23.3 Switching to Associativity Mode

The associativity mode is turned on using the *SMUNFOLD* command. Make sure there are no faces selected.

From the **Sheet Metal** section of the Ribbon, select the **Associatively Unfold Body** icon ( ) and select **Flange** face.



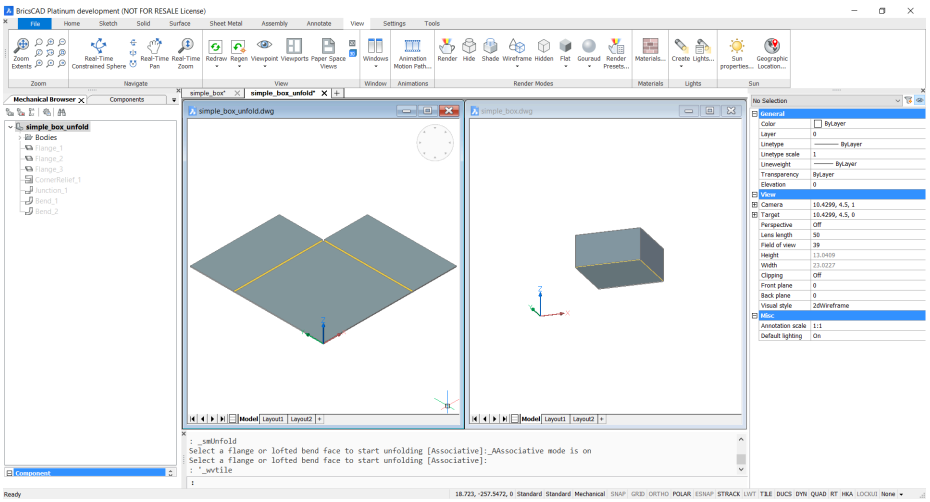
The application will compute flattening of the given part and switch it to associative mode. Condition of success of given operation is the same as for regular unfolding. Continue working in associativity mode only if no errors or warnings appeared during the operation. A new temporary document containing the flattened part will be created.



Uwaga: There is no need to save this document. It is used to propagate editing operations from the flattened solid back to the 3D model.

Use **Vertical Tile** to view both the 3D model and associative unfolding documents.

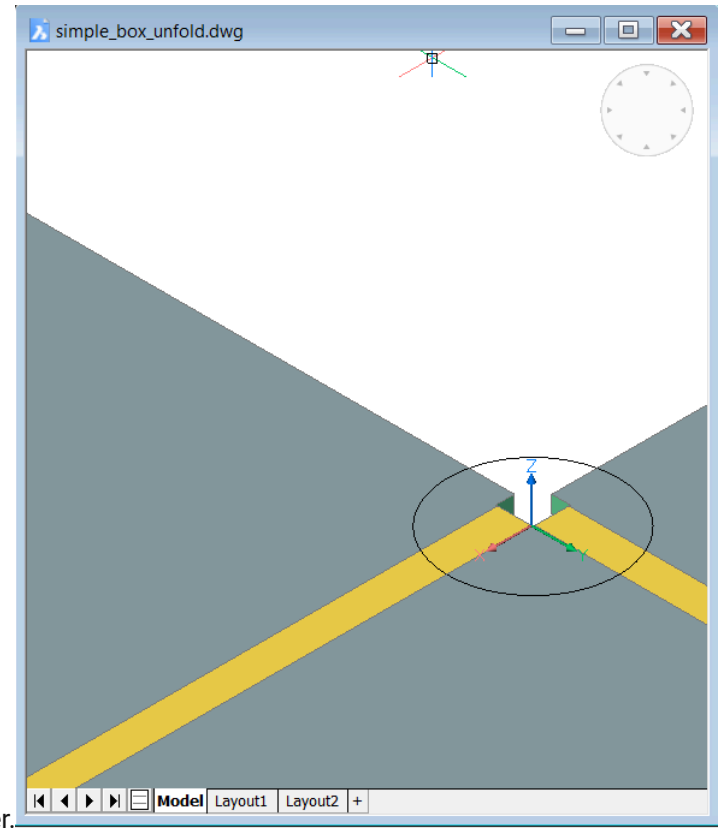
Click the **Vertical Tile** button on the Toolbar or Ribbon.



Uwaga: In the **Mechanical Browser**, when **flattened geometry** is selected, it lists features are disabled.

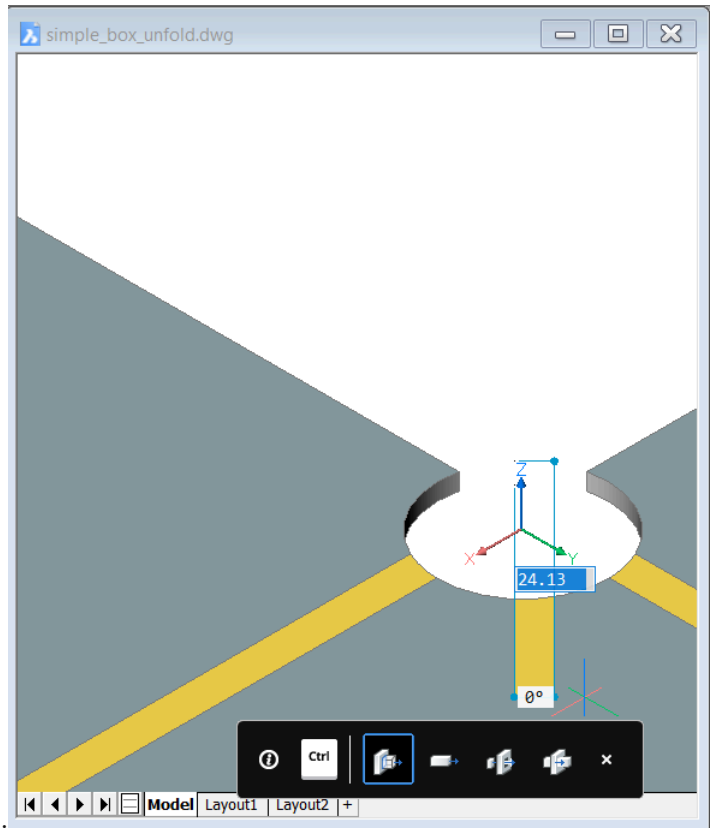
21.5.23.4 Editing the Geometry

1 Working with flattened design document, draw a circle in the



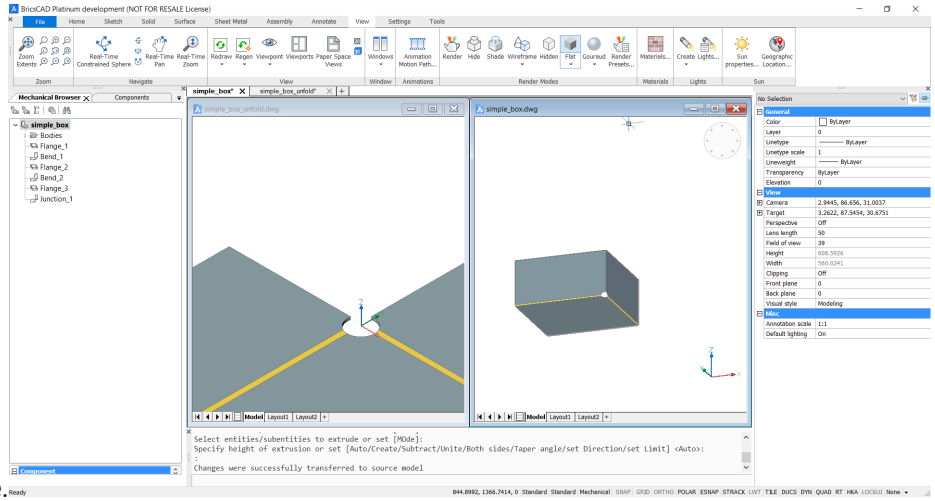
corner.

2 Apply *DMEXTRUDE* command to the circle, cutting the



volume.

3 Click on the original document again to make it

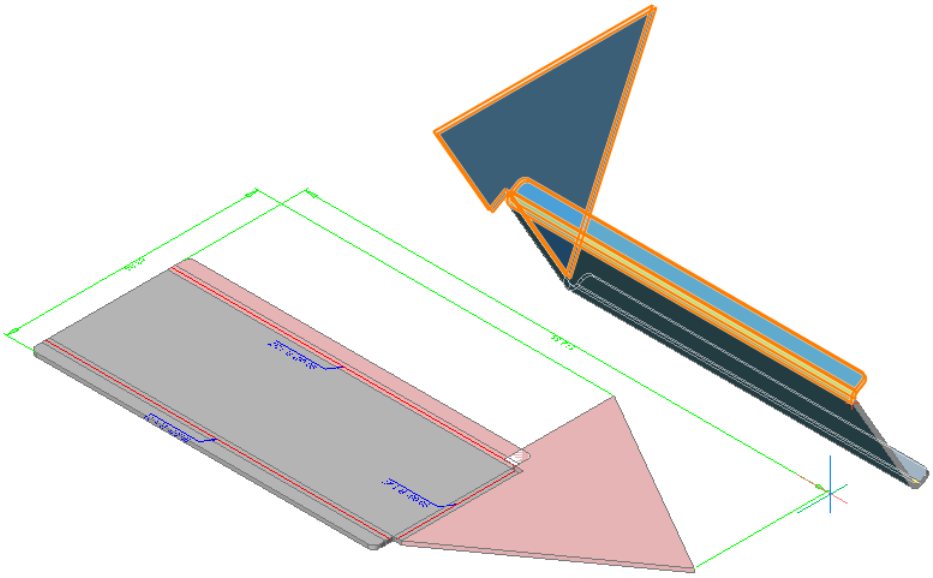


active.

4 The model will be updated and the custom corner relief appears. Application writes to the Command line: Changes were successfully transferred to source model.

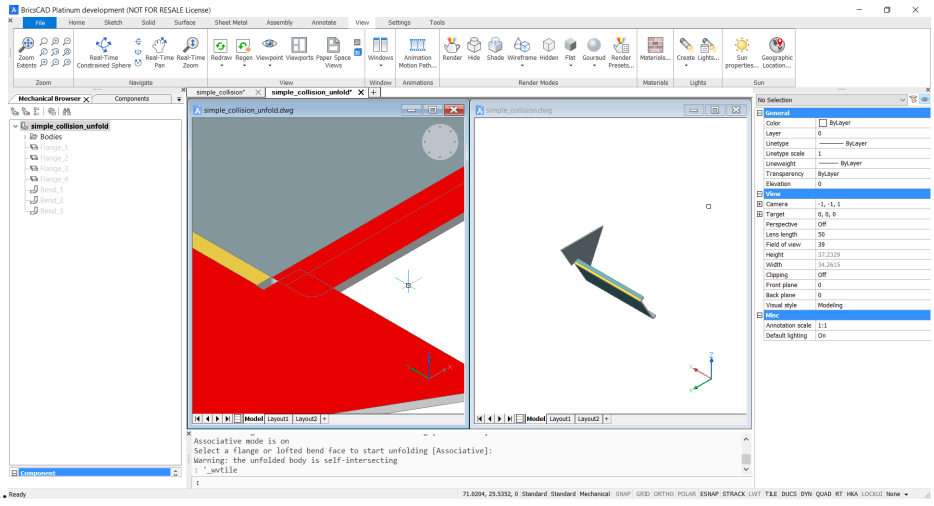
21.5.23.5 Avoiding Collisions in Flat Pattern

Another useful application of associative unfolding is fixing the not so obvious material collisions.

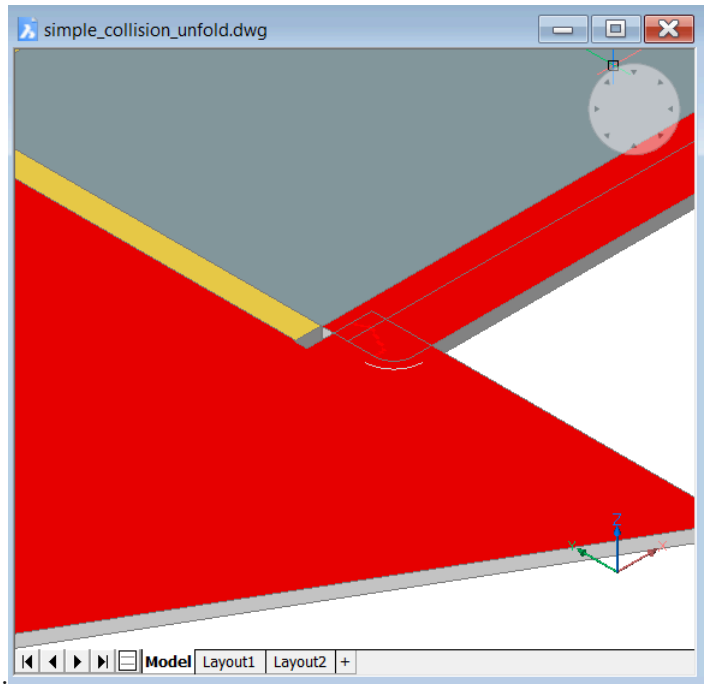


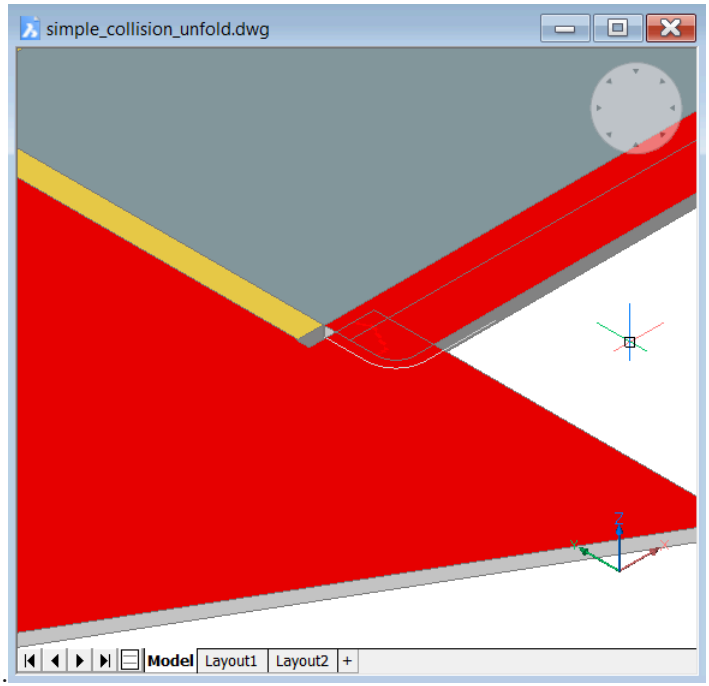
In this example, we do not know what changes to apply to the 3D model to avoid the collision when flattened. Let us rework the model with Associative Unfolding.

1 From the **Sheet Metal** section of the Ribbon, select the **Associatively Unfold Body** icon (), select **Flange** face, unfold and click the **Vertical Tile** button () on the Toolbar or Ribbon to tile the opened documents vertically and zoom to the collision area.

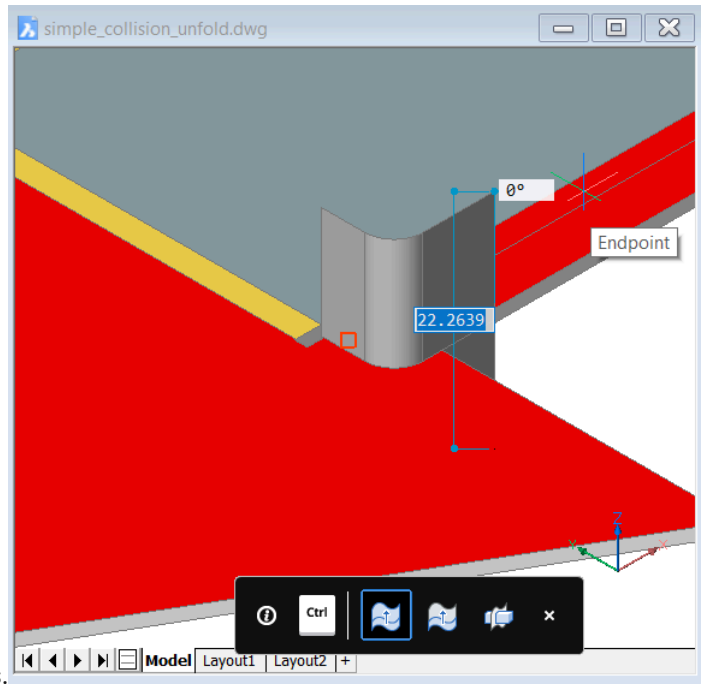


2 Select boundary circular edge and offset it by

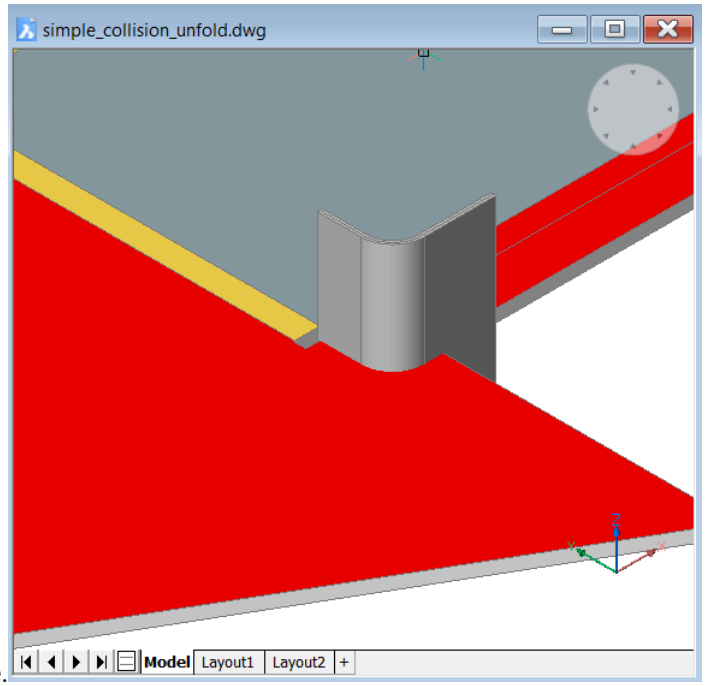
1. 

3 Draw extra straight lines adjacent to the edge and join them. 

4 *DMEXTRUDE* the polyline on both sides.

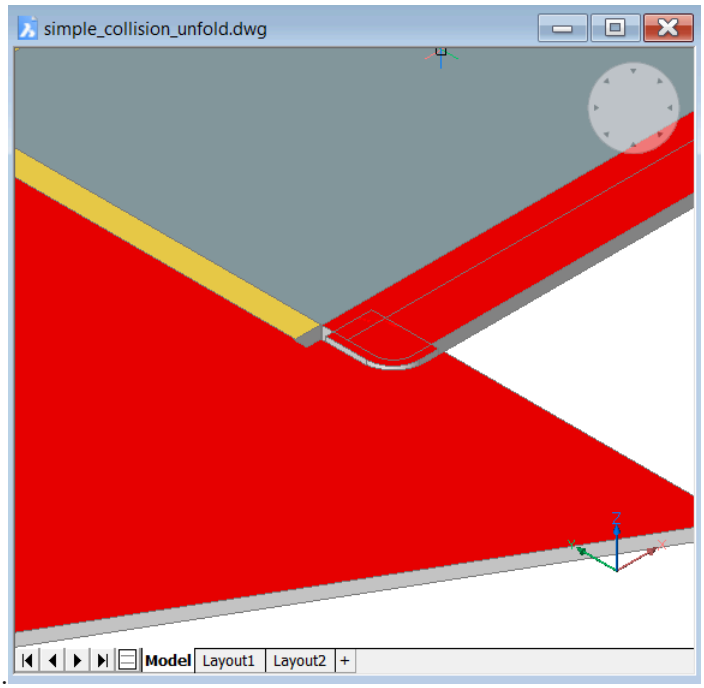


5 *DMTHICKEN* the surface by a bigger value.

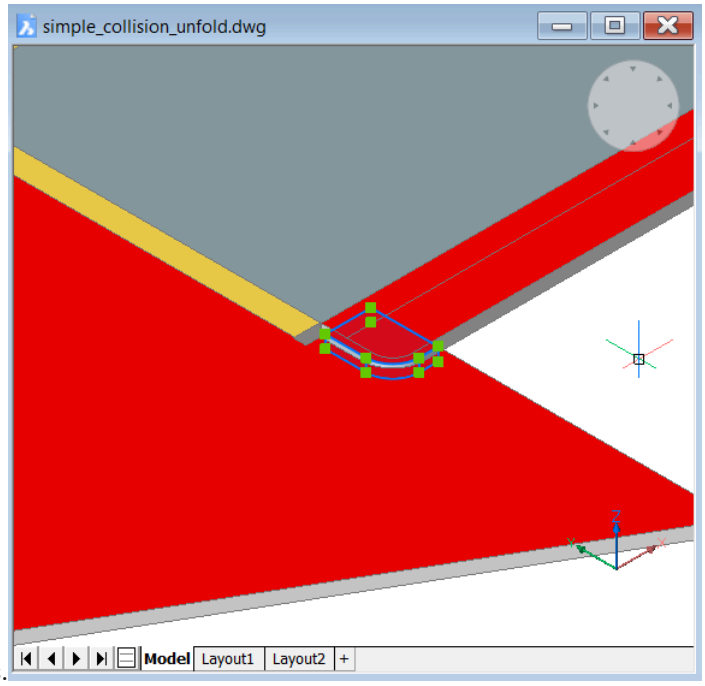


6 *SUBTRACT* the thickened body from unfolded

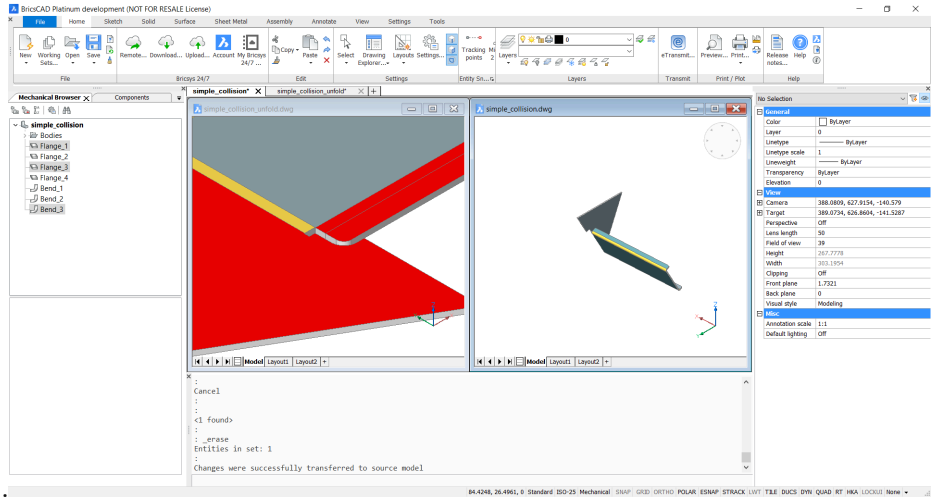
pattern.



7 *ERASE* the leftover pieces.



8 Switch back to the document with the 3D

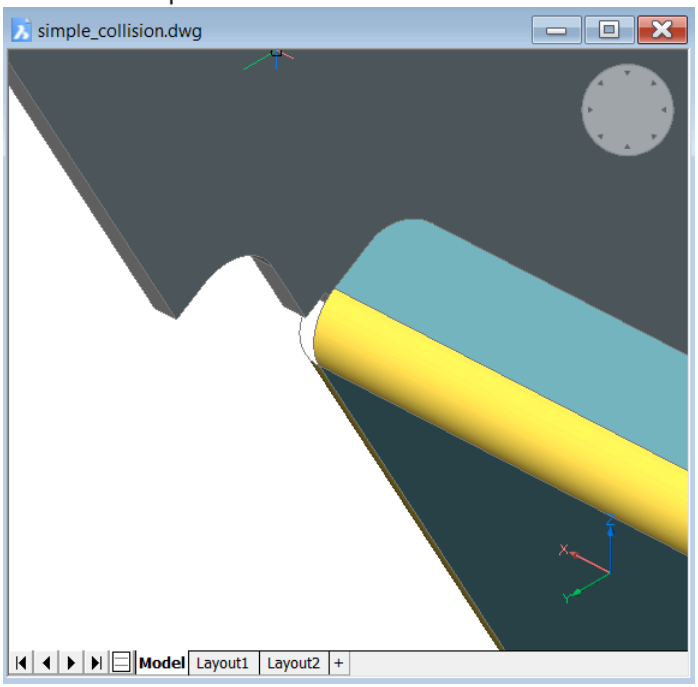


model.

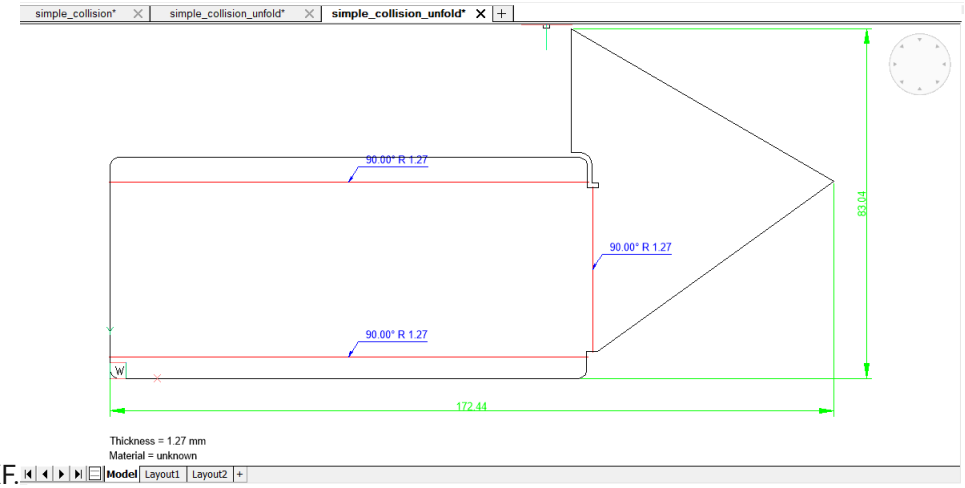
9 Make sure that the changes have been propagated.

Changes were successfully transferred to the source model.

Zoom to the updated corner.



10 To test the 2D unfold, export the file as



a .DXF.

21.5.24 Inteligentne funkcje

21.5.24.1 Przegląd

Technologia Smart Features zarządza lokalną inteligentną aktualizacją geometrii po wykonaniu poleceń Arkusza Blachy. Zachowanie jest kontrolowane za pomocą preferencji użytkownika **SmSmartFeatures**.



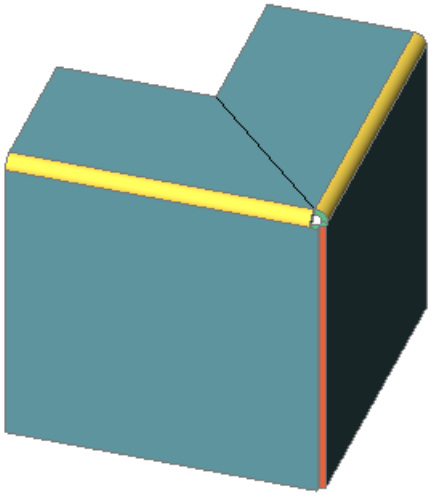
Dostępne są 3 opcje:

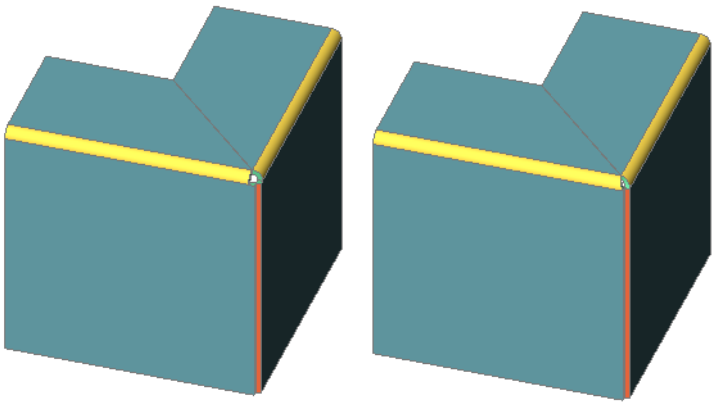
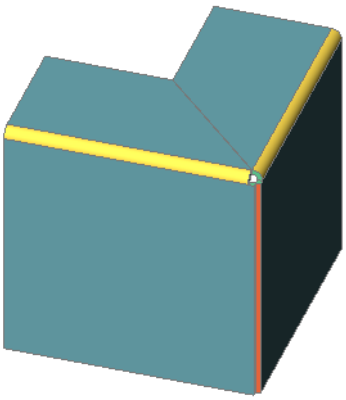
- Pozwól na przebudowanie arkusza blachy:
  - Podcięcie naroża po wykonaniu jest przekształcane w 2 podcięcia pod gięcie *SMSPLIT* na kołnierzu.
  - Niepotrzebne wycięcia są usuwane.
- Zezwalaj na automatyczny odcisk krawędzi po przebudowaniu: automatyczne odciskanie krawędzi na grubości ścian.
- Zezwalaj na automatyczne tworzenie złącz po utworzeniu gięć.

Uwaga: Zawsze miej włączone opcje Smart Features, chyba że wystąpi niespójne zachowanie.

21.5.24.2 Przekształcenie wycięcia narożnego w wycięcie gięcia

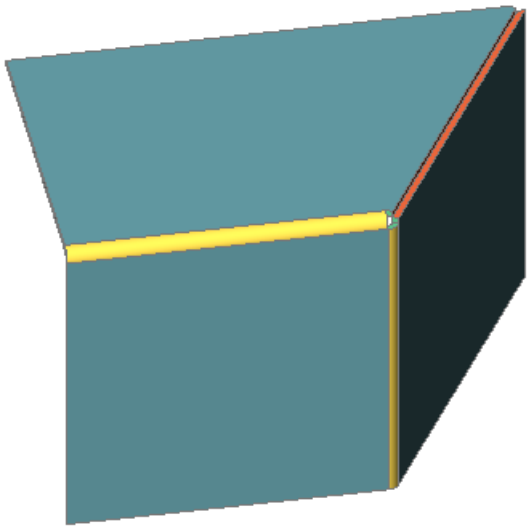
Podczas dzielenia górnego kołnierza linią na 2 oddzielne części wycięcie narożne jest przekształcane w 2 wycięcia gięć.



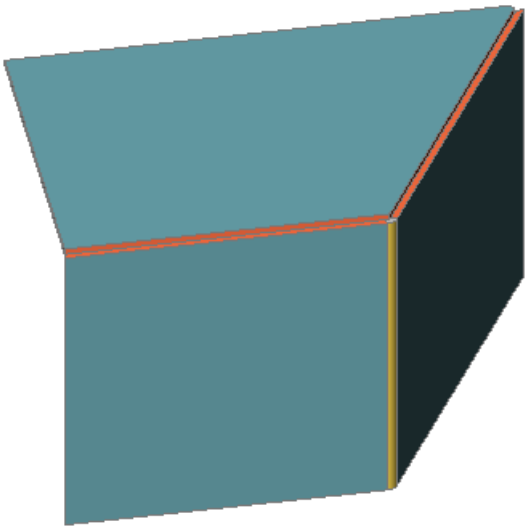
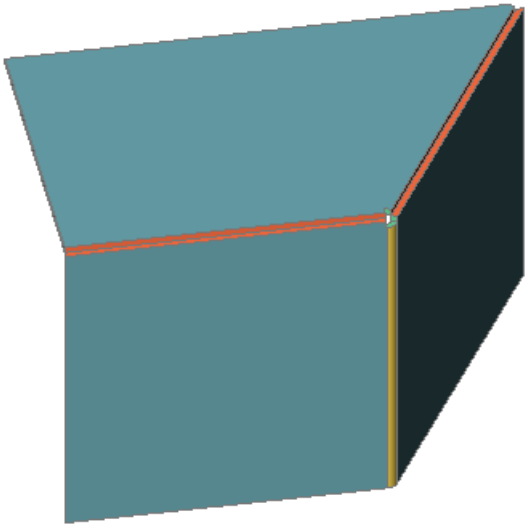
	
Inteligentne funkcje automatycznie tworzą wycięcia gięć, które można następnie przełączyć na łagodne.	Bez inteligentnych funkcji wycięcie narożne jest zachowane i nie można go zmienić.

21.5.24.3 Usuwanie niepotrzebnych cięć

W poniższym modelu istnieje wycięcie narożne, które jest potrzebne, aby zapobiec rozciąganiu materiału.



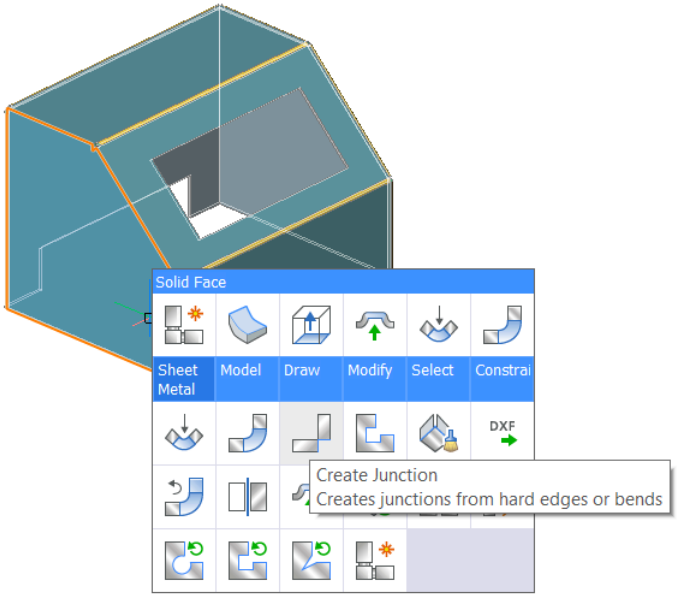
Kiedy górne gięcie zamienimy w złącze, dochodzimy do oddzielnych części. Inteligentne funkcje automatycznie usuwają niepotrzebne wycięcie.

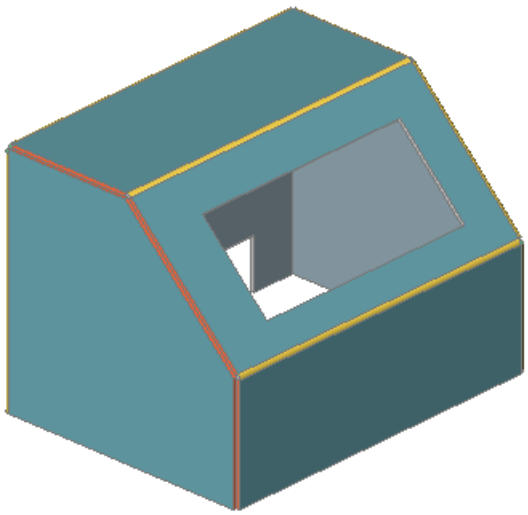
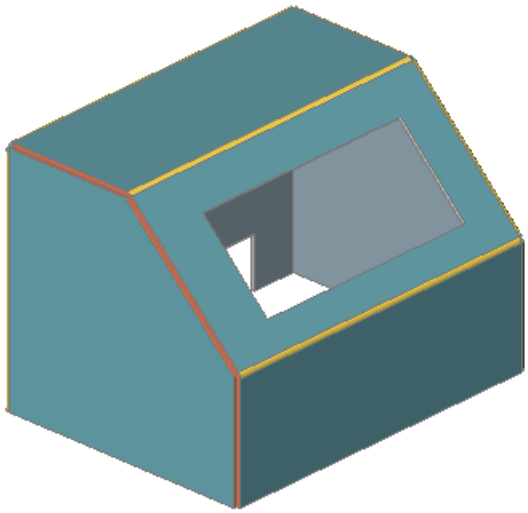
	
Inteligentne funkcje automatycznie wypełniają niepotrzebne cięcia, co upraszcza geometrię.	Bez inteligentnych funkcji wycięcie narożne nadal istnieje.

21.5.24.4 Automatyczne odciskanie krawędzi

Nadrukowane krawędzie na grubych powierzchniach dzielą je na obszary, które można automatycznie przerobić. Oznacza to, że zlokalizowane skrzyżowania można przekształcić w zakręty. Bez automatycznie odciskanych krawędzi potrzebne jest tutaj polecenie *SMIMPRINT*.

W poniższym przykładzie oddzielimy kołnierz za pomocą złącz. Włączamy funkcję **Przebudowa elementów arkusza blachy**, a następnie używamy **Automatyczny odcisk krawędzi po przebudowaniu**.

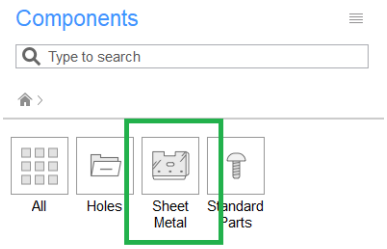


	
Zezwalaj na automatyczny odcisk krawędzi po przebudowaniu = WŁ. Powierzchnie złączeń są prawidłowo podzielone.	Zezwalaj na automatyczny odcisk krawędzi po przebudowaniu = WYŁ. Elementy złączeń nakładają się na siebie.

21.5.24.5 Automatyczne tworzenie złączeń

W pewnych sytuacjach, gdy ostre krawędzie spotykają się w narożniku i na wszystkich poza jednym tworzymy gięcie, które musi stać się złączem, aby część mogła się rozkładać.

Załóżmy, że tworzymy gięcia na górnej powierzchni skrzynki z blachy:



Następnie pozostałe twarde krawędzie przylegające do powierzchni muszą stać się złączami. Gdy **SmSmartFeatures**= WŁ, odbywa się to automatycznie:



Aby być precyzyjnym, obszar automatycznego tworzenia złączeń znajduje się we wspólnym wierzchołku twardych krawędzi (lub wycięciu narożnym podczas tworzenia gięć). Ta opcja nie optymalizuje układu rozwiniętej części.

21.6 Wygenerowane rysunki

21.6.1 Drawing Views

21.6.1.1 Overview

BricsCAD Mechanical provides a toolset to generate drawing views from 3D models of parts and assemblies. Drawings views are created in the paper space of the same DWG file. The 3D model is created in model space. Drawing views can be annotated using dimensions, leaders, text and tables. Views and annotations are updated when the associative 3D model is changed.

The tools to generate, update and annotate drawing views are located on the **Annotate** tab of the **Mechanical workspace**:



Rysunek 150: Annotate

Tabela 45: Annotate

Drawing view	Description
Base Views	Generate associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
Projected Views	Generate additional projected views from an existing drawing view.
Section View	Create a cross-section view based on a standard drawing view generated by command *VIEWBASE* in a paper space layout.
Detail View	Create a detail view of a portion of a standard generated drawing at a larger scale.
Edit View	Change the scale and the hidden line visibility of drawing views.
Update Views	Updates a selection of drawing views obtained by *VIEWBASE* and *VIEWSECTION* when *VIEWUPDATEAUTO = 0*.
Section Style	Predefined and user-defined styles.
Detail Style	
Export Views	Export the content of drawing views, obtained by command *VIEWBASE* and *VIEWSECTION*, to the Model Space of the drawing, or to a new drawing.

21.6.1.2 Draft-quality drawing views

By default, BricsCAD generates precise (high-quality) drawing views for any 3D model. The edges of such views (including silhouette edges) are represented by 2D entities; lines, circles, arcs and splines. The generation of precise views for a large number of 3D entities can take a significant amount of time. This is because every silhouette edge must be approximated with an analytical 2D curve.

You can reduce the time needed to generate drawing views by changing the *DRAWINGVIEWQUALITY* setting to *0*. This enables draft-quality drawing views. Such drawing views are internally represented as

3D solids, which makes it impossible to put annotations on their silhouette edges. However, they look very similar to a precise (high-quality) drawing view and you can use them to quickly create layouts.

A draft-quality drawing view can be switched to a high-quality drawing view using its *Geometry* property (change it from *Draft* to *2D*).

21.6.2 Opisy rysunków

Tworzenie i edytowanie adnotacji, przejrzyj Omówienie narzędzi wymiarowania .

Wymiary i wielolinia odniesienia, które są w widoku rysunku, są aktualizowane po zmianie podstawowego modelu 3D.

Rysunki montażowe można opisywać za pomocą numerów pozycji BOM. Każdy numer pozycji odnosi się do wiersza w tabeli BOM wygenerowanej na podstawie złożenia.

Tabelę BOM można umieścić w obszarze papieru (układach) z widokiem rysunku złożenia.

Reprezentacje zespołów rozstrzelonych mogą być używane do generowania widoków rysunkowych i opisywania ich odnośnikami.

22. Communicator 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

Uwaga: 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

Uwaga:

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.

Uwaga:

- 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.

Uwaga: 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.

Uwaga:

- 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

Uwaga: 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!

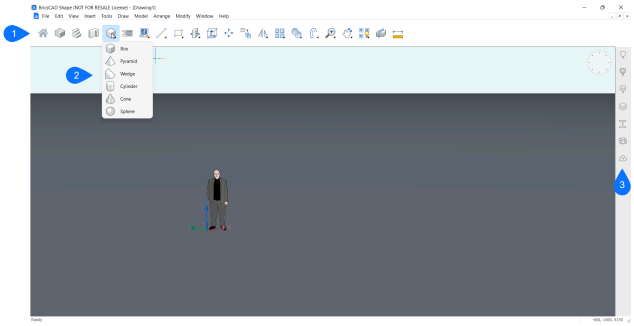
Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "10 Minutes to BricsCAD Shape" on YouTube, Bricsys channel.

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.



Uwaga: 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.

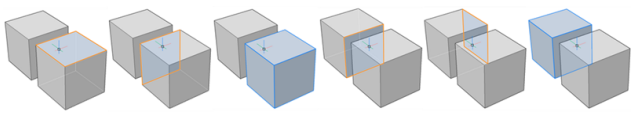
When hovering over an entity, a part of the entity is highlighted. To select the highlighted part of the entity, left-click the edge or face.

Uwaga: 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.

Uwaga: 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.

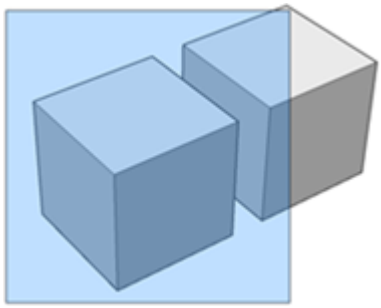
Selection windows

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.

Uwaga: 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.

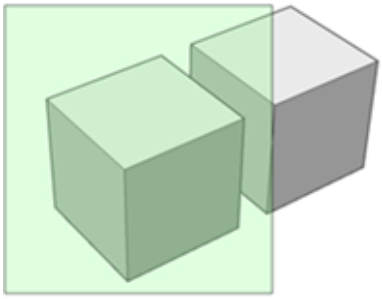
Blue selection window

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'.



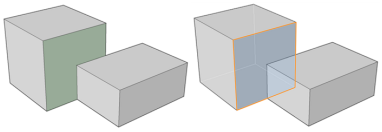
Green selection window

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.



Uwaga: 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:

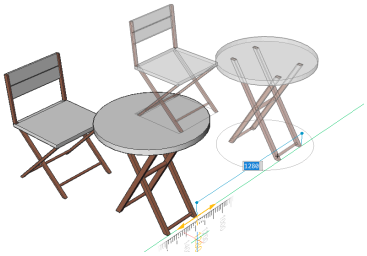
- Select the entity or subentity and choose **Manipulator** in the **Modify** tab on the **Quad**.
- Hold down the left mouse button a little bit longer when selecting.

Uwaga: 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

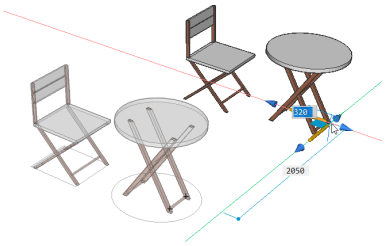
By selecting an axis of the manipulator you are able to move the entity. Specify the value of displacement by entering a value or picking a point. The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

Uwaga: To copy the entity, hold down the **CTRL** key before clicking the axis.



Select a plane

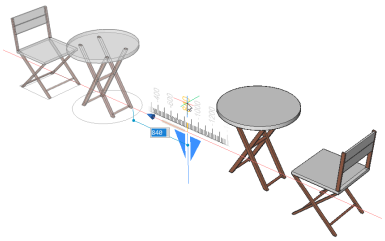
By selecting a plane of the manipulator you are able to move the entity in the selected plane after specifying the value of displacement by entering a value or picking a point.

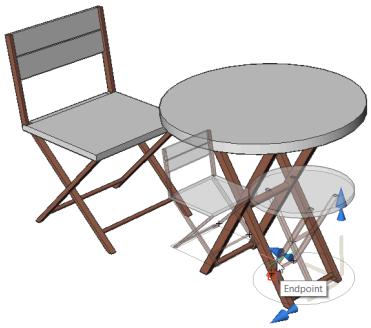


Select an arrowhead

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.

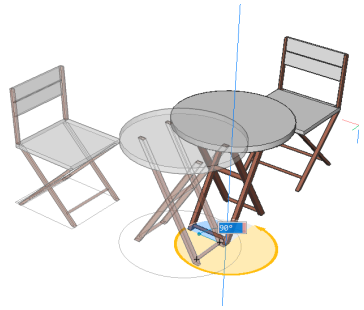
Select one of the arrowheads and specify the mirror axis or scale.





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.

Uwaga: **Zoom Extents** displays the drawing to the extents of entities

23.4.4 Look From control

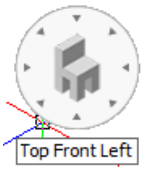
The **Look From** control allows you to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen.

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.



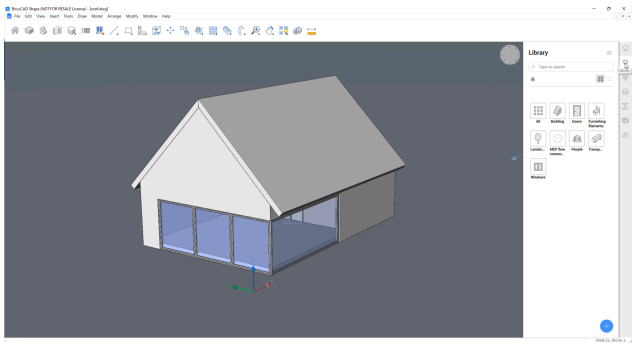
3 Click to confirm.

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*

Uwaga: 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.

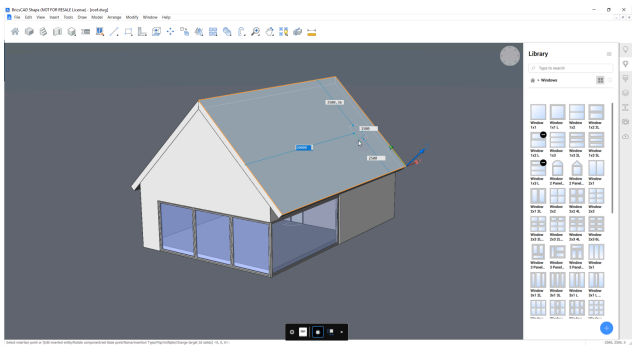
Uwaga: The selected component is attached to the cursor. The X and Y distances from the origin of the current coordinate system display dynamically.

3 Do one of the following:

- Click to place the component.
- Type a value in the X and Y distance fields.

Uwaga: 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.

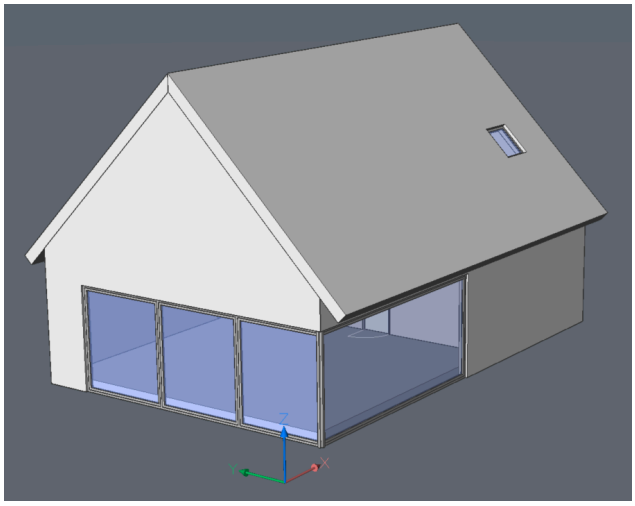
Uwaga: The window aligns to the face under the cursor. Distance fields from the sides of the face display dynamically.

3 Do one of the following:

- Click to insert the window.
- Type a value in distance fields.

  Uwaga: 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.



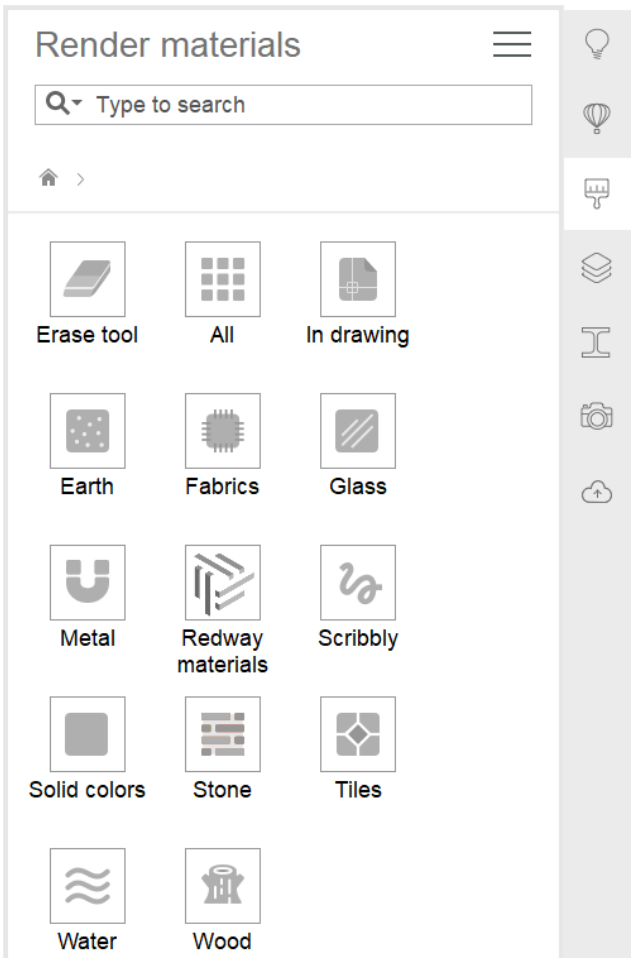
Uwaga: When you delete a window, the opening is removed.

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*.

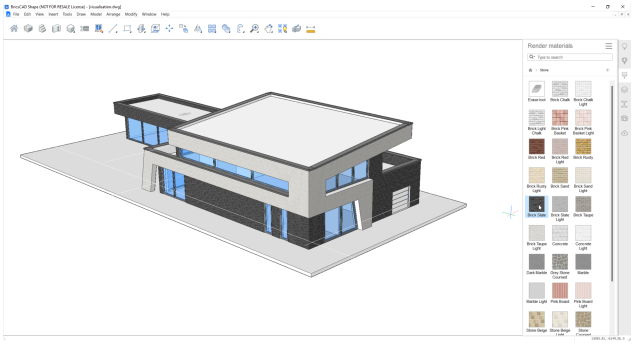
Open the **Render materials** panel by clicking the **paintbrush** icon on the right-hand edge of the screen. Several categories are available. To view all materials that are used in the drawing, select the **"In drawing"** category.



To assign a material to an entity in your model:

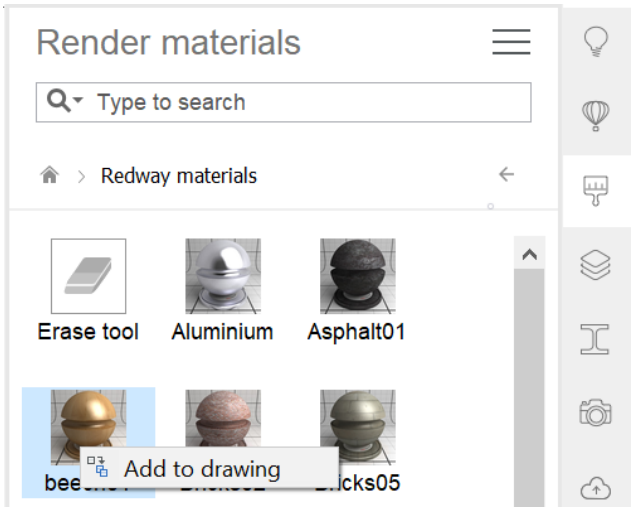
- Select the material in the **Render materials** panel.
- Select the entity you want to assign the material.

Uwaga: You can apply a material to multiple entities at once by preselecting them.



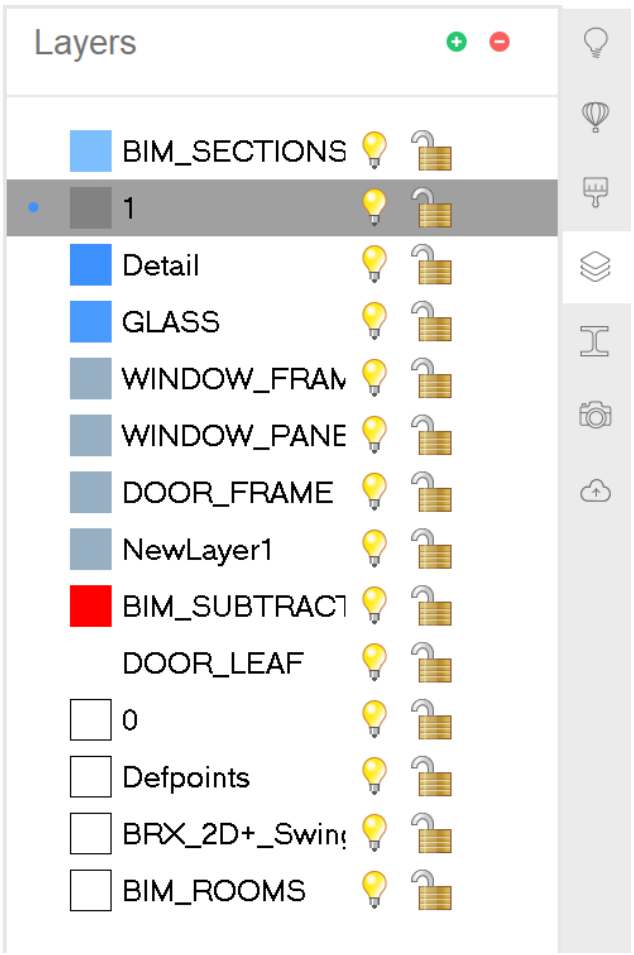
Uwaga: To remove a material from an entity, use the erase tool and click any entity to remove the material from.

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.



Uwaga: 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

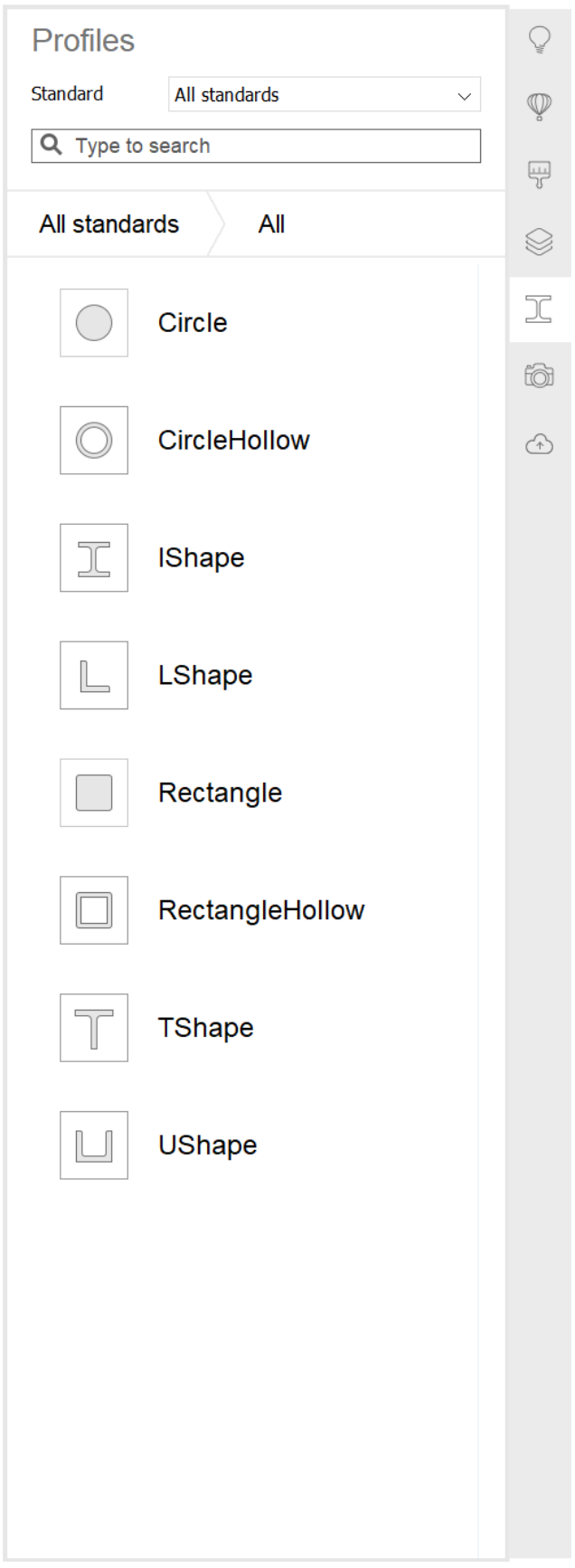
BricsCAD Shape offers standard structural steel profiles which enable you to easily create structural elements. Multiple standards are available, including American, British and European standards.

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.

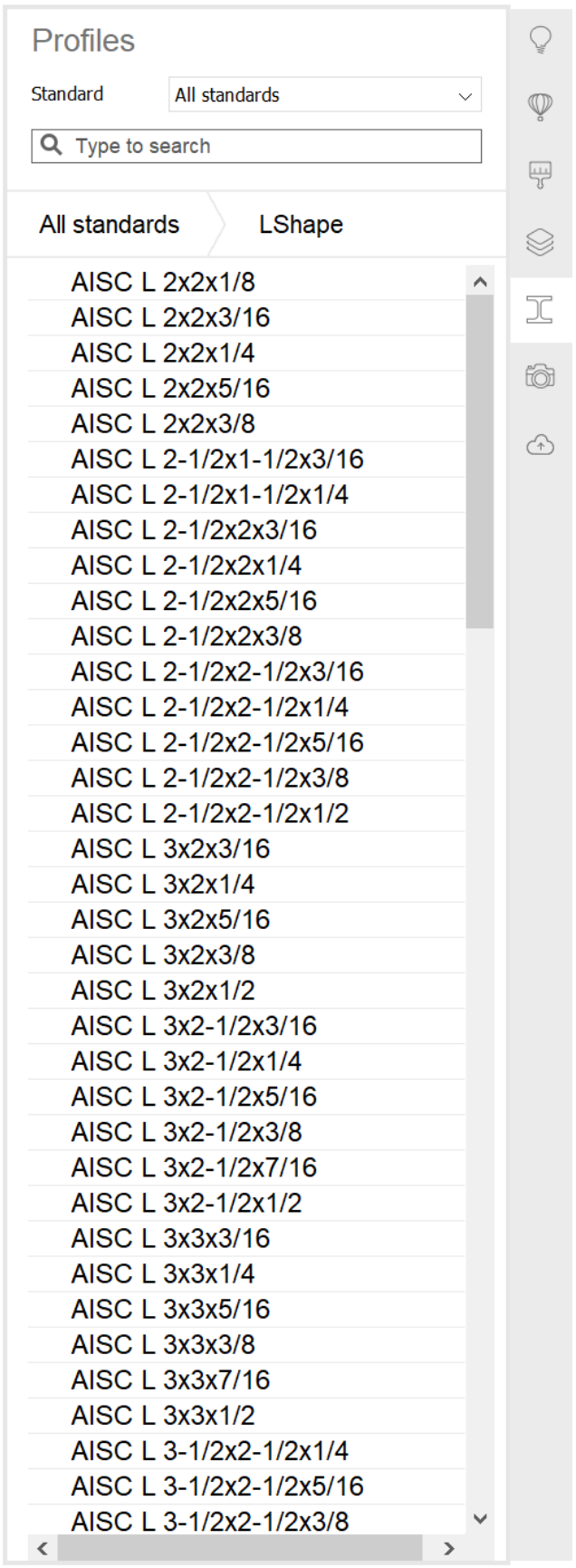
The **Profiles** panel displays.


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.

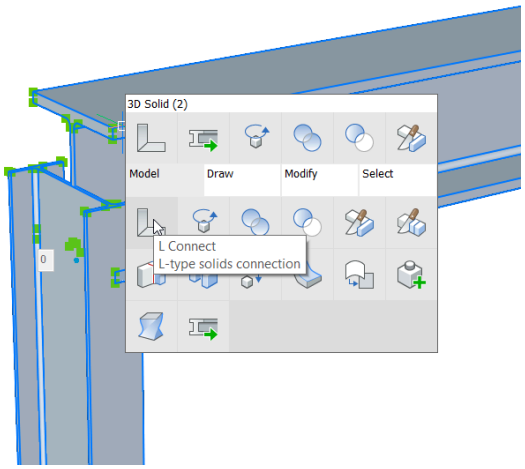
10 Repeat the previous steps to create more profiles or right click to stop.

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.

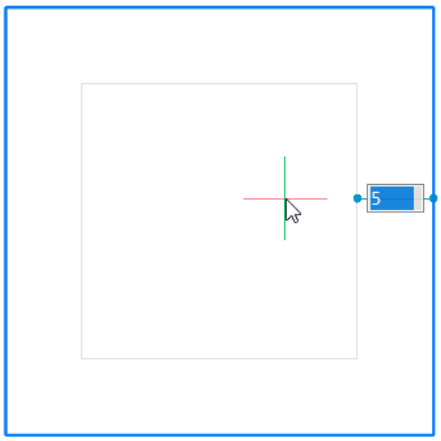
2 Hold down the left mouse button, then drag the profile onto the linear solid.

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.

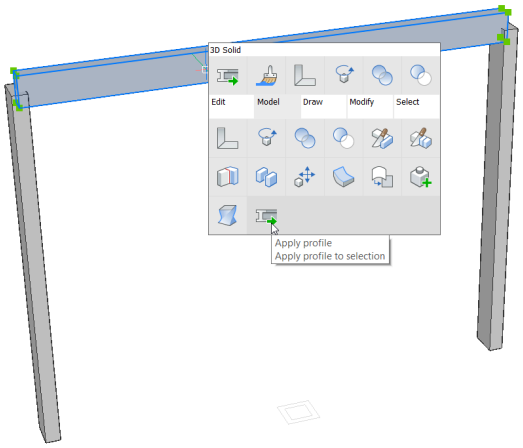
2 Optionally, use the **Offset** tool to create a hollow profile: select the outline, then specify the offset.



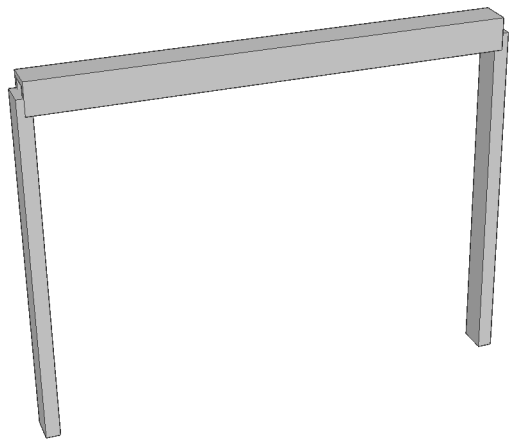
3 Hold down the **Ctrl** key, then select one or more linear solids. The linear solid(s) highlight(s).

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>:



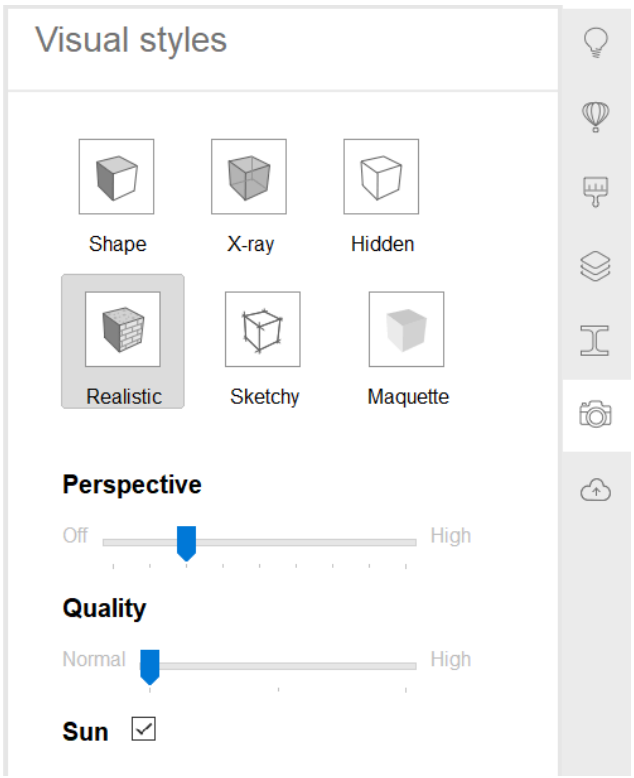
6 Do one of the following:
- Right click to apply the profile.
- Choose an option.
- Press the Esc key to quit.

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.

Uwaga: 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.

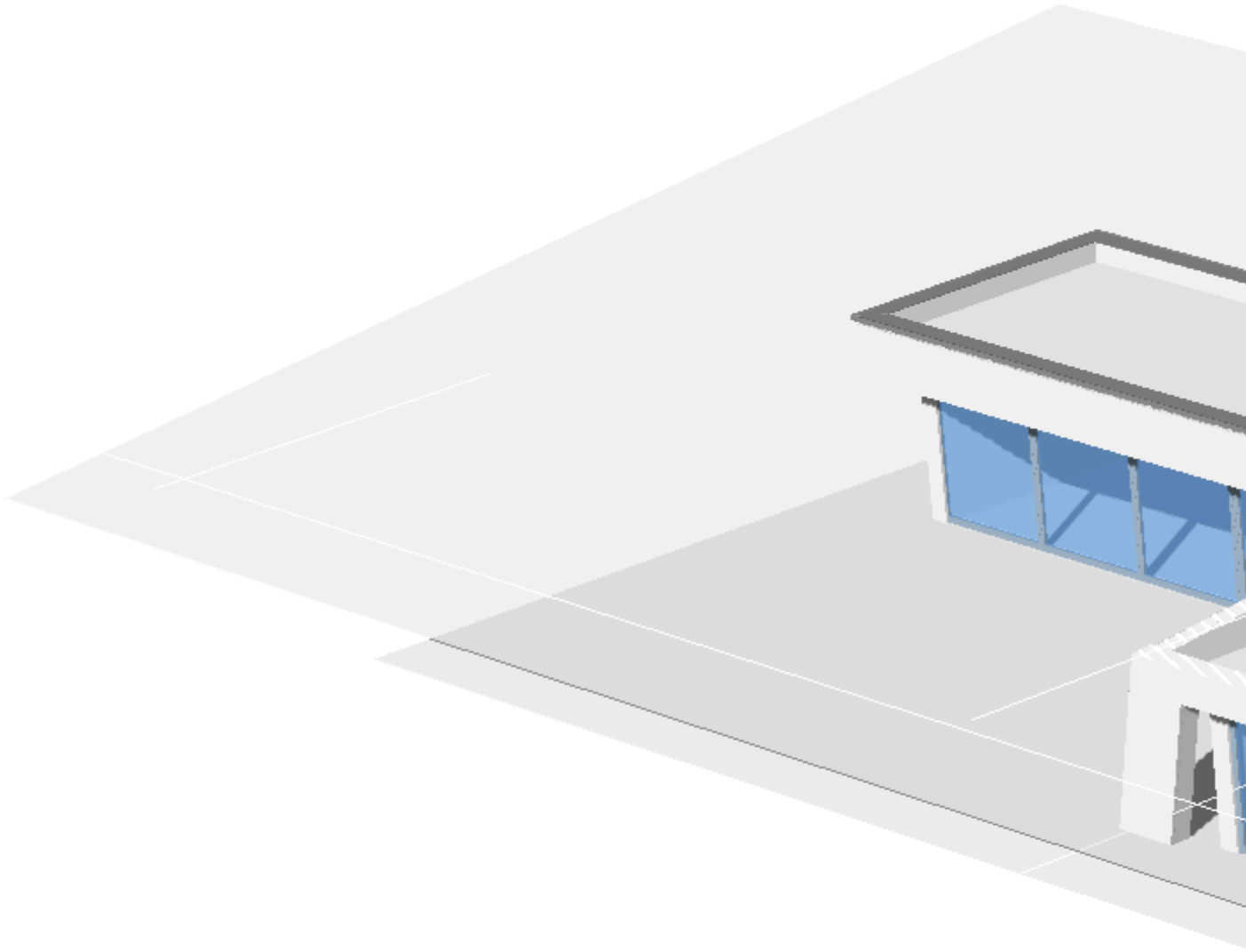
Uwaga: If you don’t choose a surface to attach the Section, a horizontal Section plane will be created.

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.

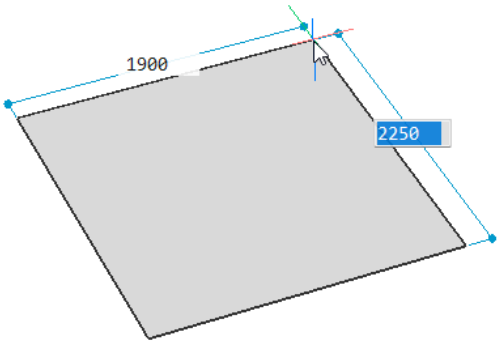
2 Specify a corner for the base of the box.

3 Specify the opposite corner for the base of the box to apply both the length and width.

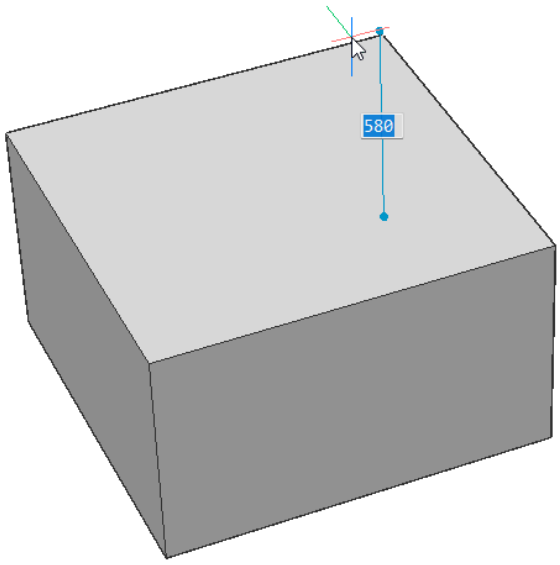
4 You can do this dynamically, by hovering the cursor at the opposite corner, or by entering the desired values in the dynamic input fields.

The box is created parallel to the x and y axes.

Uwaga: The blue dynamic field is the active one.



5 Specify the height of the box.



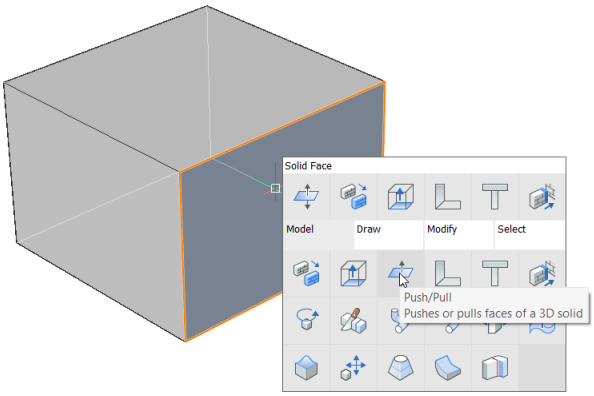
23.10.2 Manipulate a solid face

To manipulate a solid face:

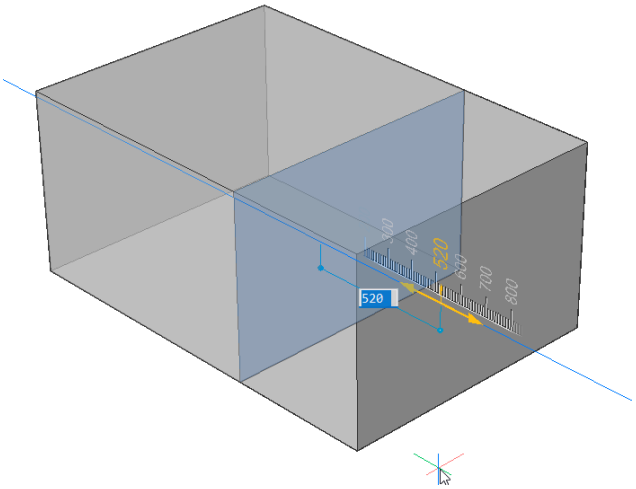
1 Hover the cursor over a face until highlighted.

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.



Uwaga: When selecting multiple faces and accessing the command, both faces will be pushed or pulled over same distances.

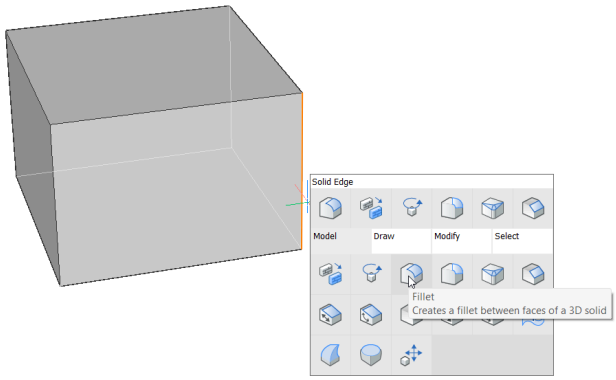
23.10.3 Manipulating a solid edge

To manipulate a solid edge:

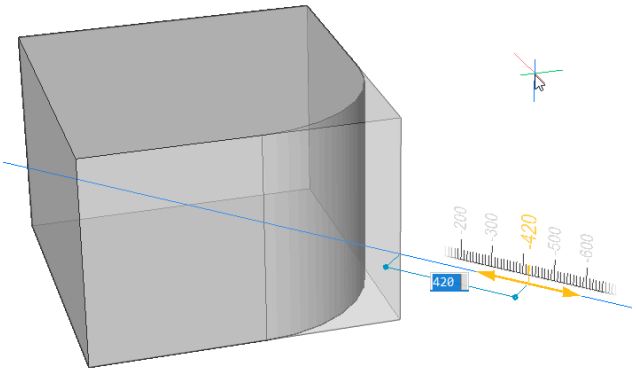
1 Hover your cursor over an edge while holding the **CRTL** key until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *FILLET* command can be accessed.

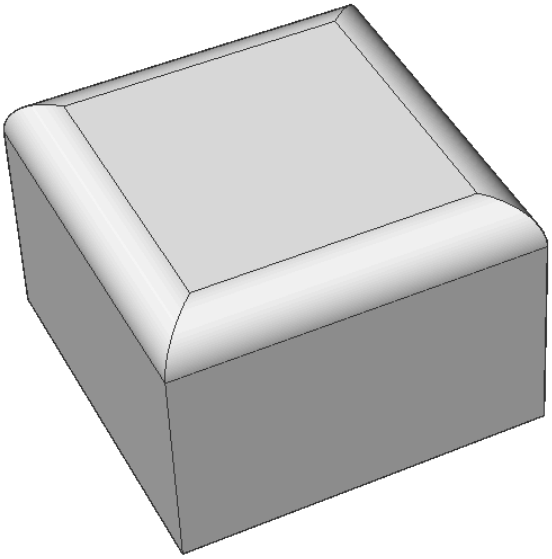


4 Specify the displacement value or move with your cursor to define the end position of the face.



Uwaga: When selecting multiple edges and accessing the command, all selected edges will have a fillet of the same radius.

Uwaga: 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.

Uwaga: 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.

Uwaga: 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:

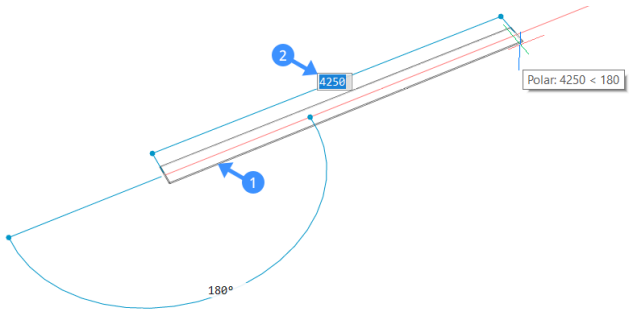
- Click the **Polysolid** icon

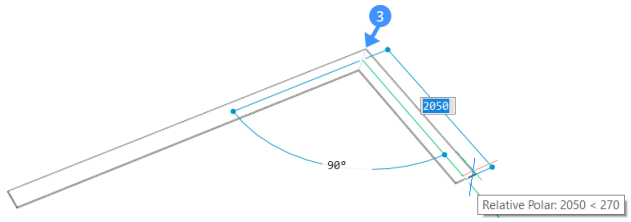
  in the toolbar
- Type *POLYSOLID* in the command line.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

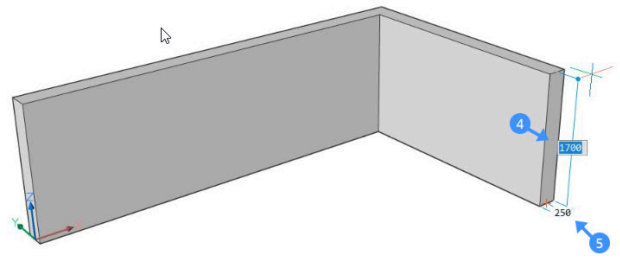
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



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.



Uwaga: 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.

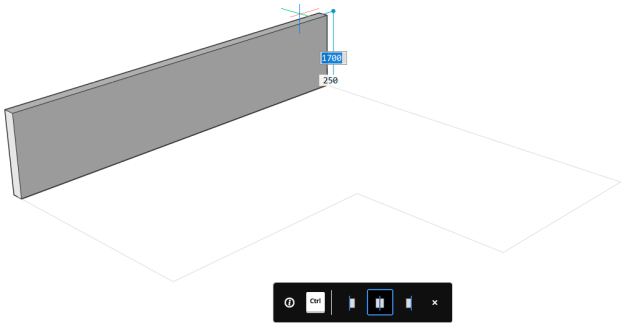
2 Type E in the command line or select **Entity** in the prompts menu.

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.

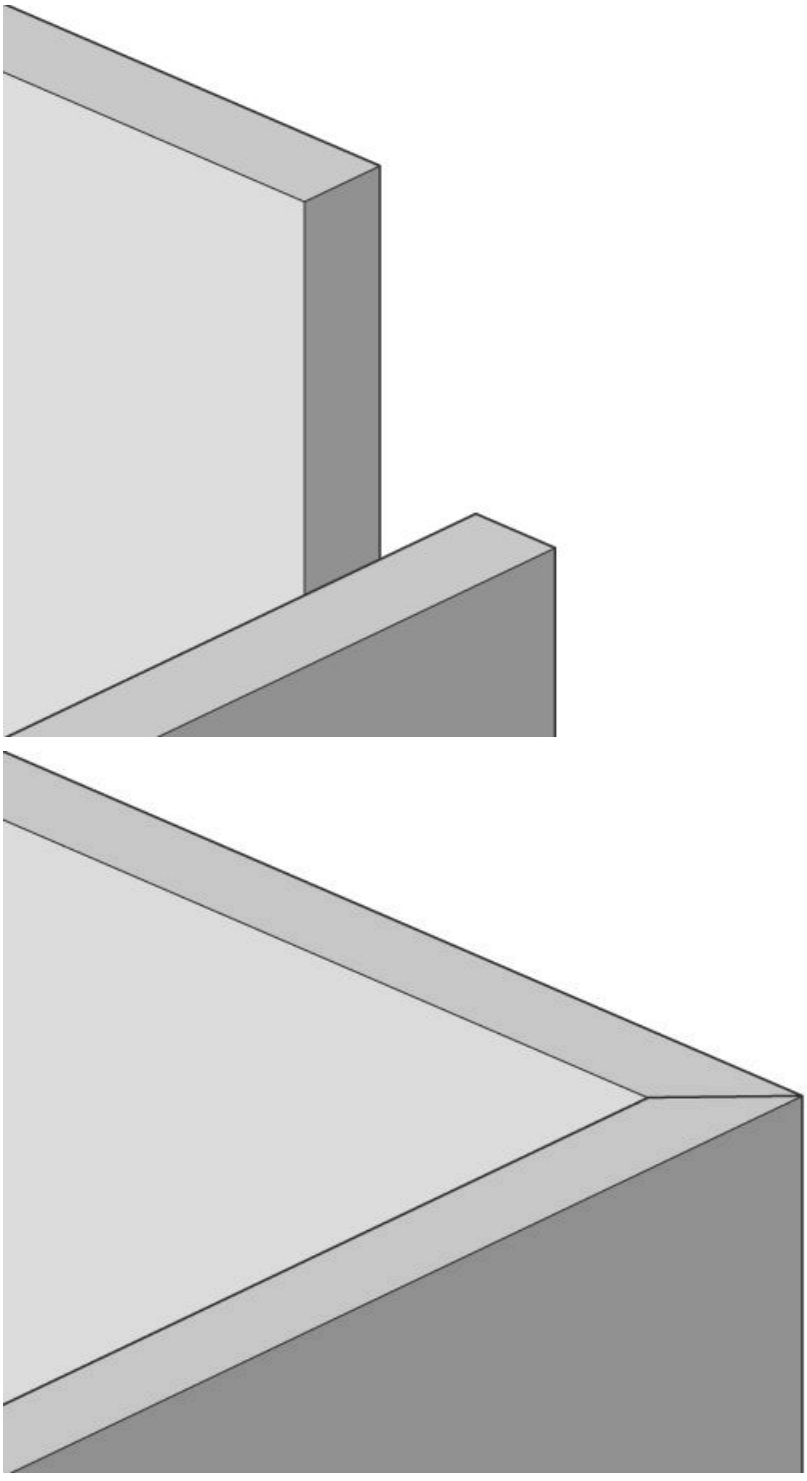
6 The **Hotkey Assistant** appears and displays the possible justification options. Press **CTRL** to cycle through the wall justification options.



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.



Uwaga: 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.

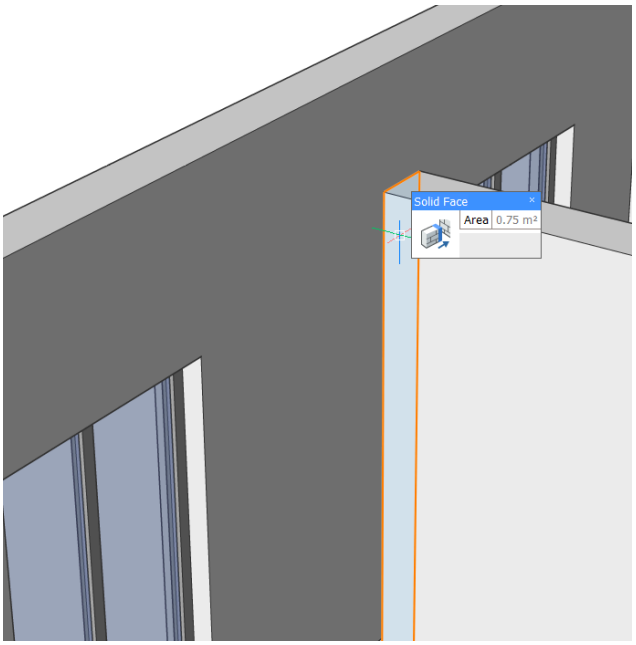
The tool allows you to connect the minor face of a wall to the major face of another element or wall. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or side face of a slab to a wall.

2 Select the faces of the objects that you would like to connect and press **Enter**.

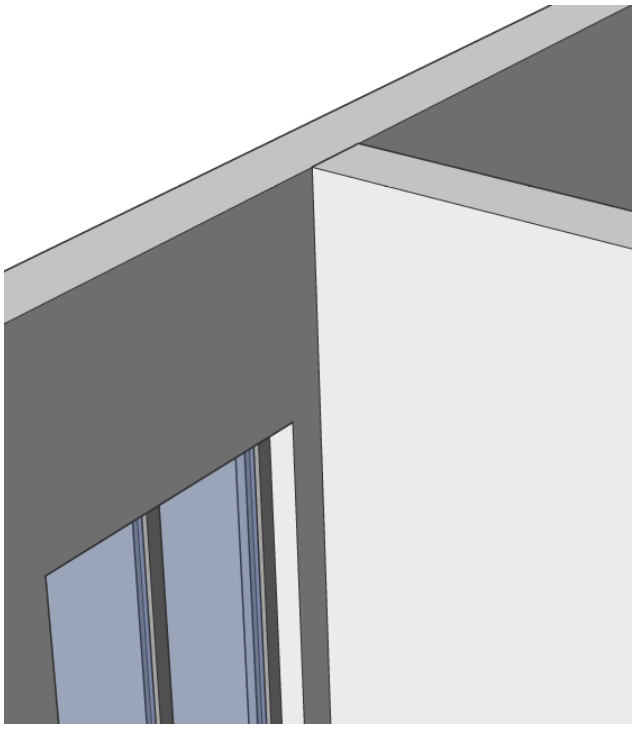
3 Select **Connect to nearest** in the prompt menu.

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.



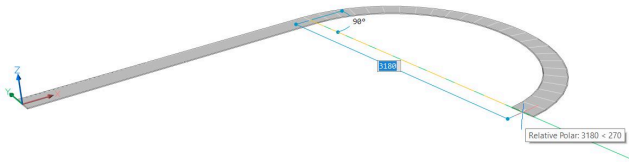
23.11.6 Procedure: Creating a curved wall

1 Launch the *POLYSOLID* command.

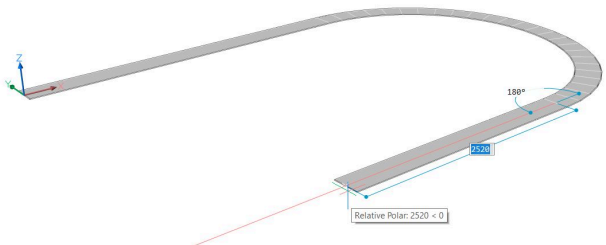
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

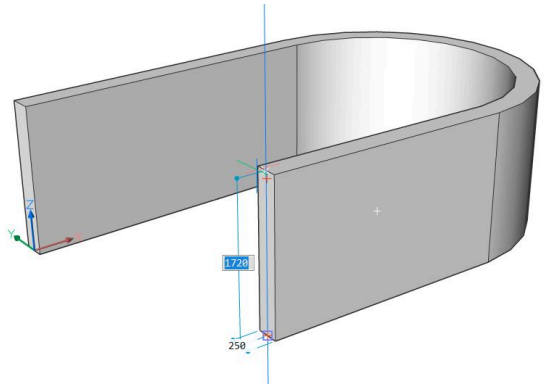
4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press **Enter**.



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.

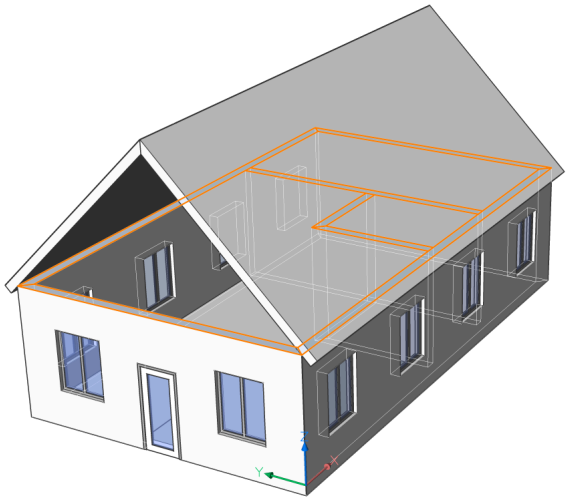


6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

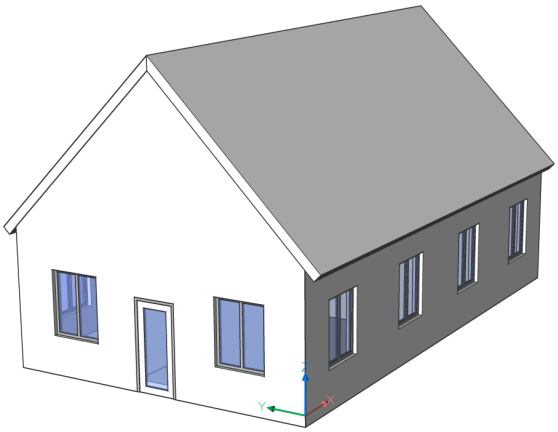


23.11.7 Procedure: Connecting multiple faces of the wall to the roof slab

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

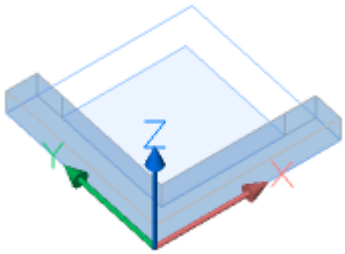
23.12.1 About the Quickdraw tool

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

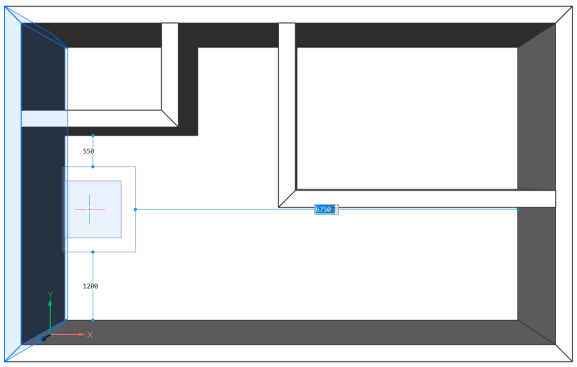
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object. You have the following options:

1 Align to the origin

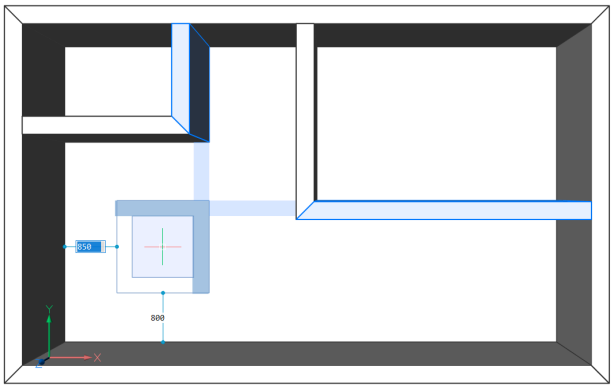


2 Align to existing walls

- 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.

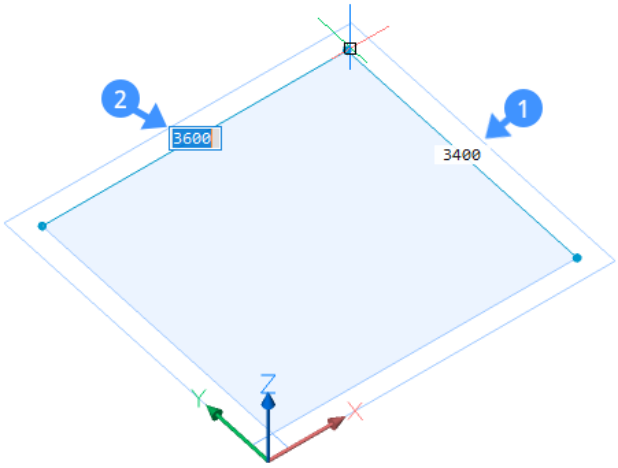
You are prompted: Select first point or [floorHeight/wallWidth/slabThickness/autoAdoptsizes]:

Uwaga: 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.

Uwaga: 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.

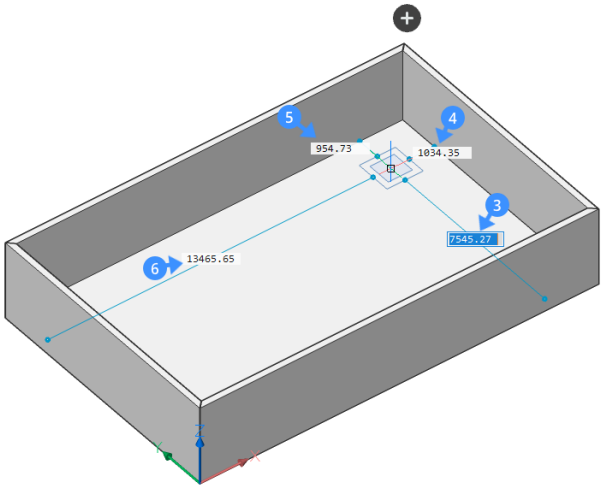
3 Drag the mouse cursor to define the footprint of the building.



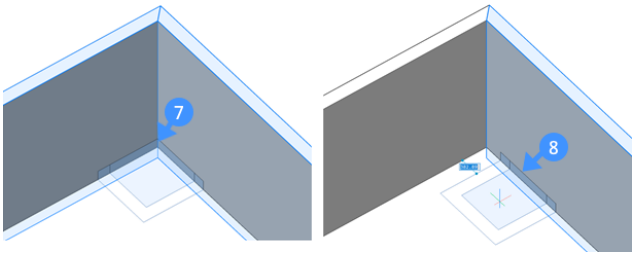
Specify the values of the room width and length in the Dynamic dimension field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

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).



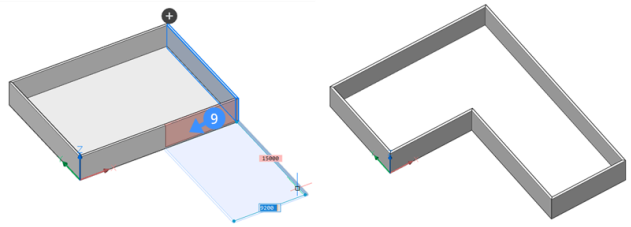
If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



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.

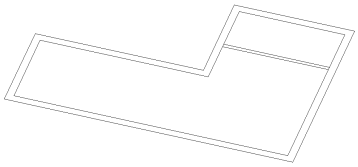


Uwaga: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

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

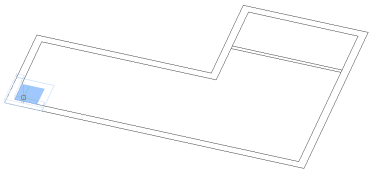
1 Open a 2D floorplan of a rectangular or L-shaped building.



Uwaga: 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.

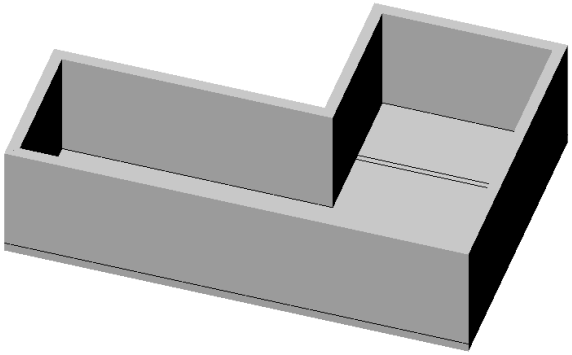
2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

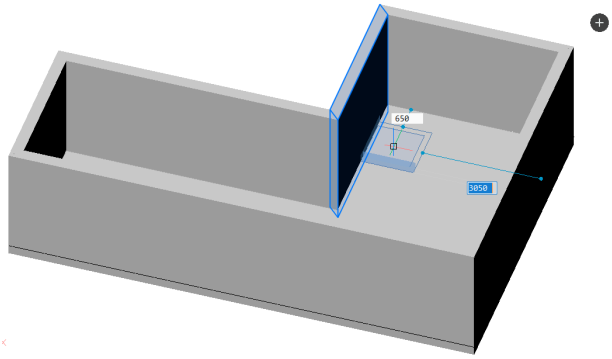


Uwaga: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

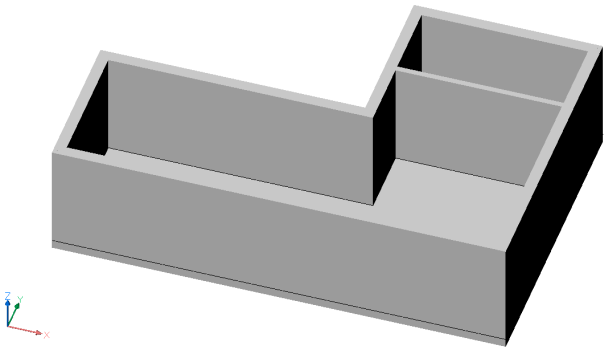


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Uwaga: 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

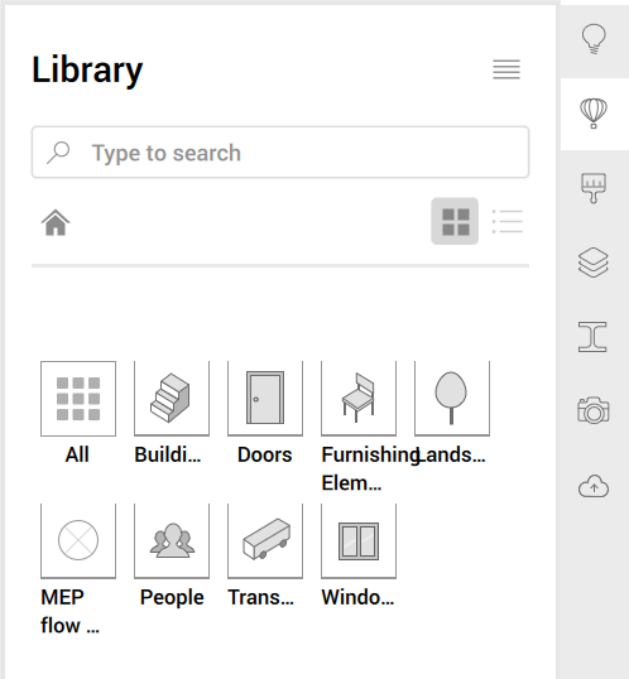
There are two ways to create windows or doors in BricsCAD Shape.

- 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.

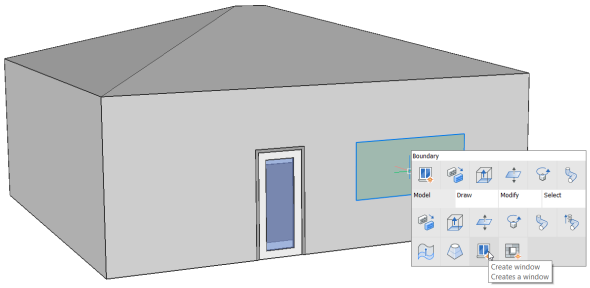
6 Press the **CTRL** key to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the **Tab** key. When you're done sizing and placing the door, press the **Enter** key.

Uwaga: 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.

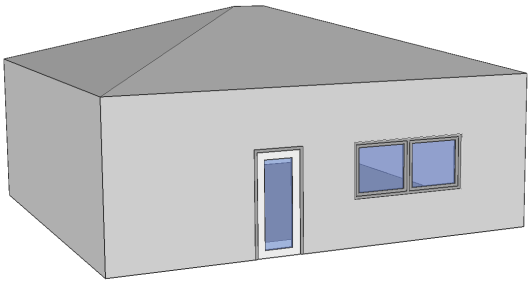
23.13.2 Procedure: Create a custom window

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:

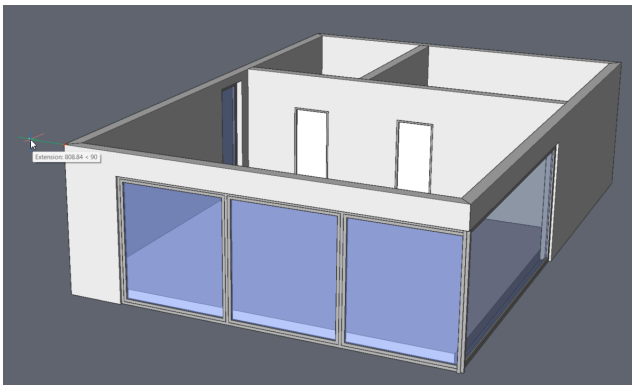
1 Select face of the outer wall and use the **Offset** tool from the Quad.

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

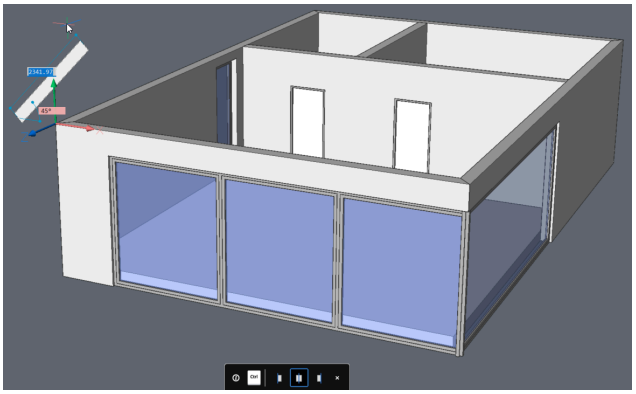
Launch the *POLYSOLID* command trough the toolbar or the "No selection" Quad.

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.

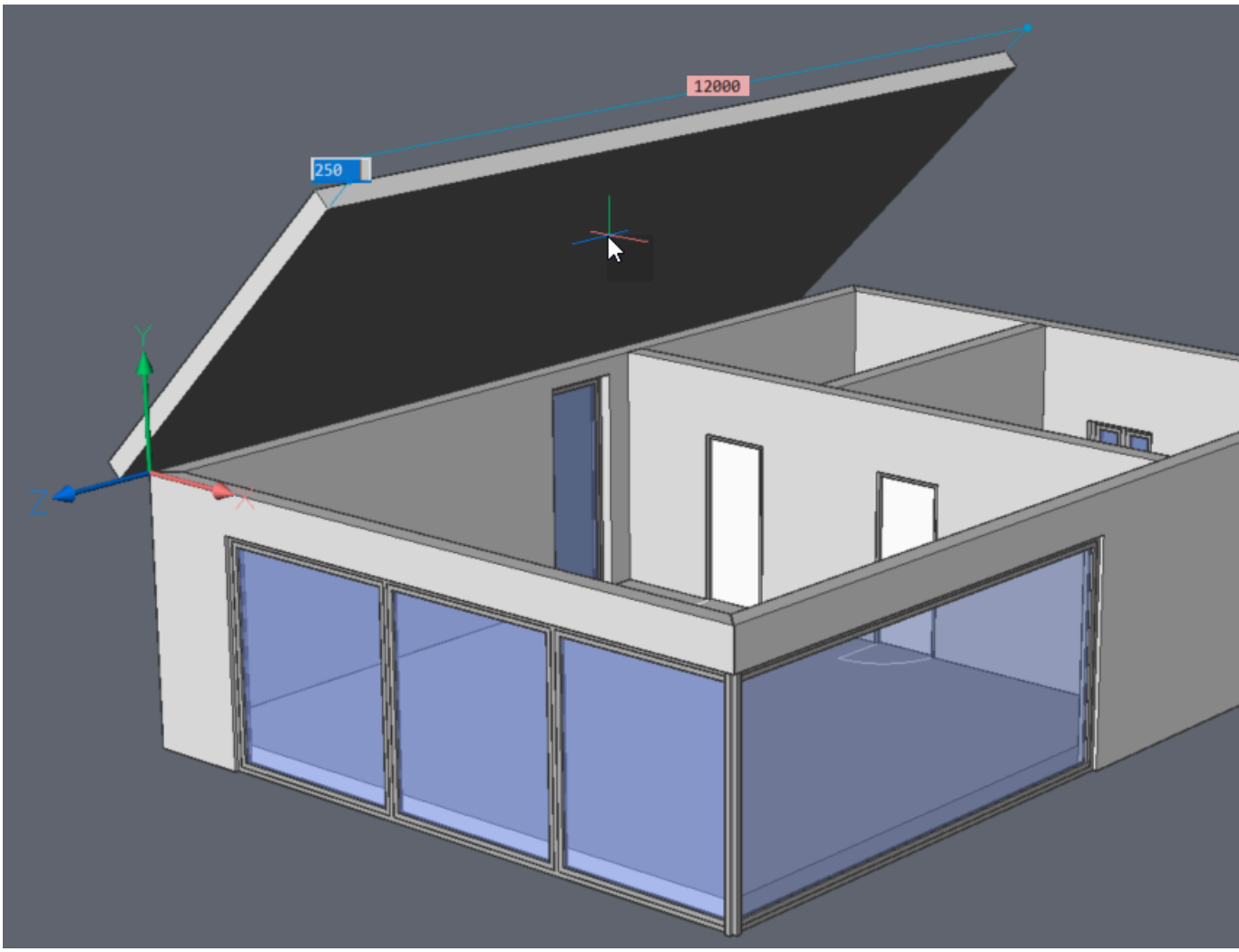


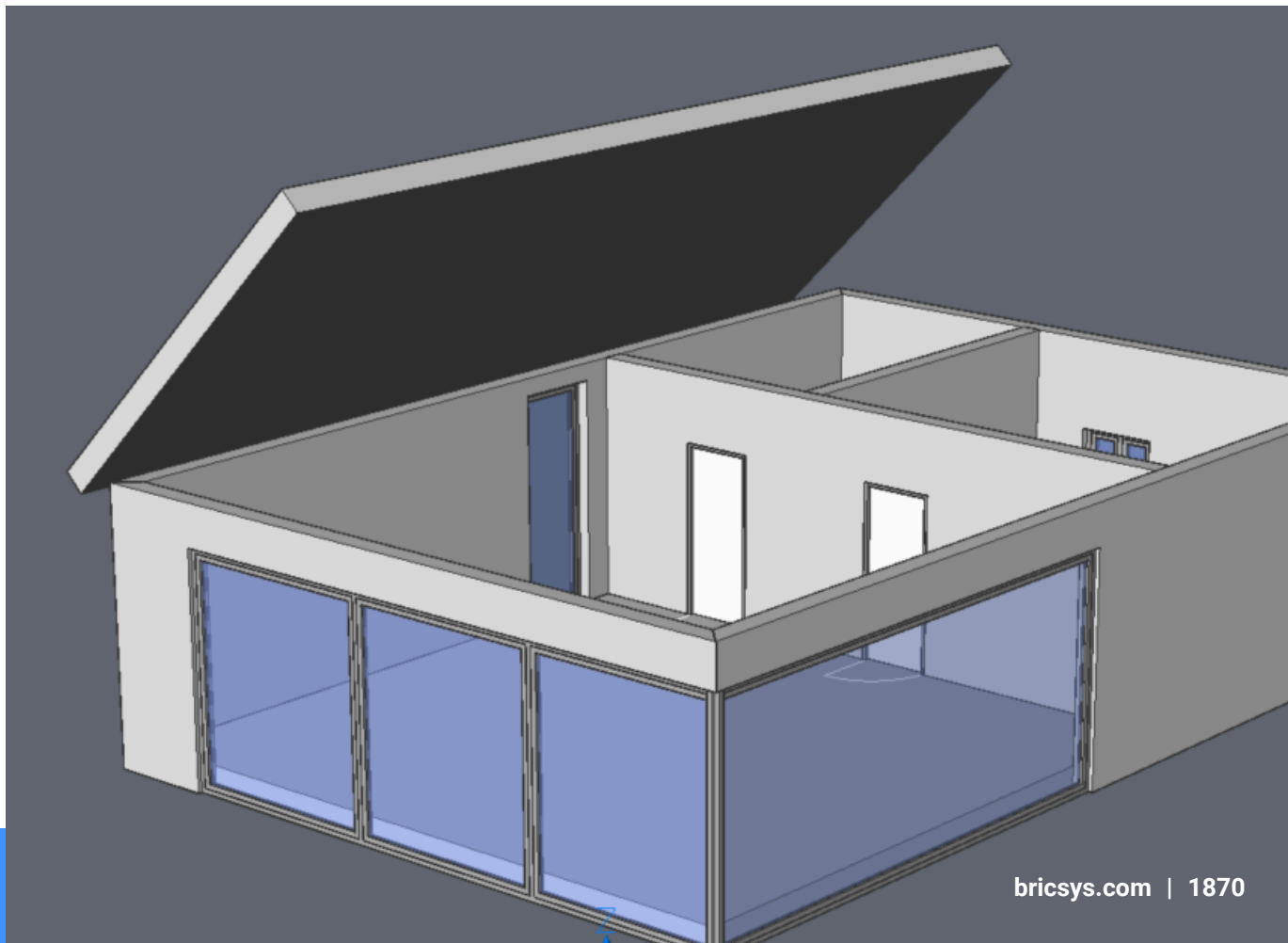
2 Pick an origin point to start drawing.

3 To lock the inclination, press TAB until the dynamic dimension of the inclination turns blue. Enter a value and press Enter.

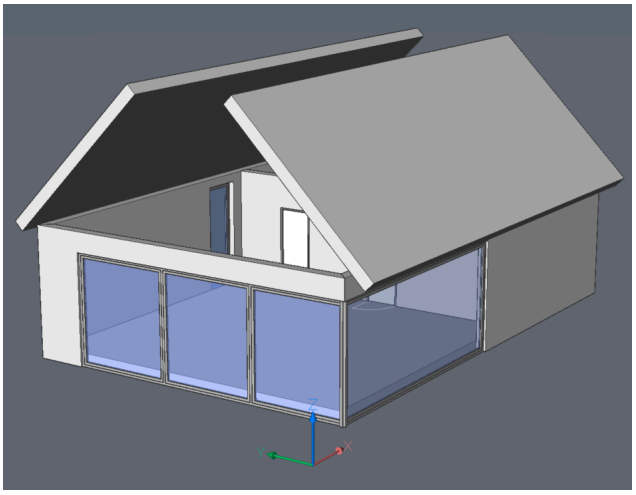


4 The profile thickness can be changes by pressing the TAB key and entering the desired value.

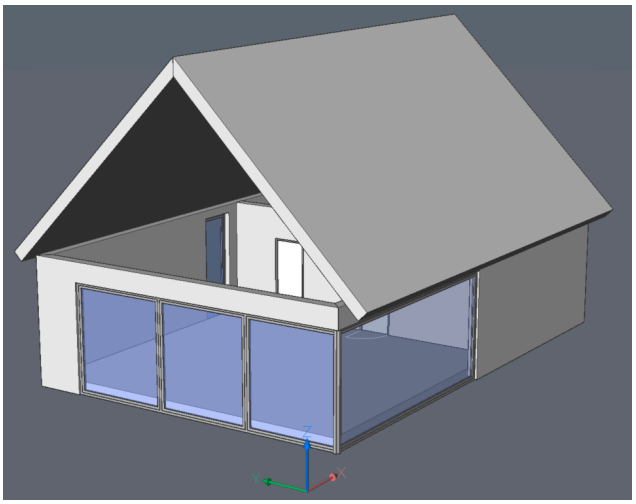

12000
250
Y
Z
X



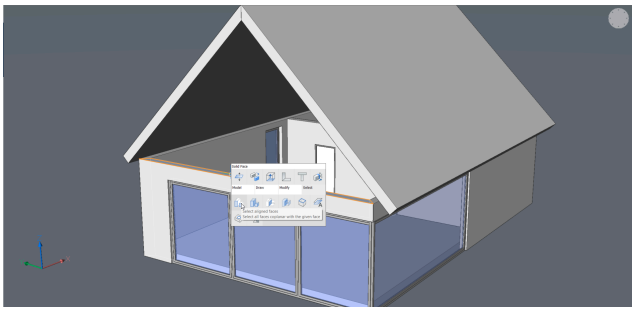
5 Repeat the same process on the other side of the building. Another way to have identical slabs on both sides is by 3D mirroring the first slab.



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.

