Bricsys®

BricsCAD V21

Documentazione del Prodotto

Bricsys®

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1. Cosa c'è di nuovo

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 alla pagina 32

Productivity alla pagina 35

Drafting alla pagina 49

Modeling alla pagina 51

Civil tools alla pagina 60

Performance and compatibility alla pagina 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 alla pagina 967

Modeling techniques alla pagina 968

Building data alla pagina 975

Project collaboration alla pagina 984

Design documentation alla pagina 985

Point clouds alla pagina 994

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:

Panoramica alla pagina 1538

Gestore BOM alla pagina 1539

Librerie di parti alla pagina 1542

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Lamiera alla pagina 1543

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

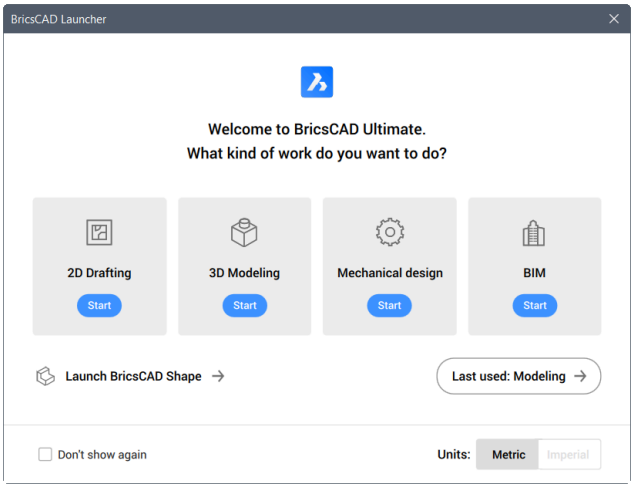
BricsCAD® V21 offers a variety of user interface enhancements to help increase your productivity without disrupting your existing workflow.

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.

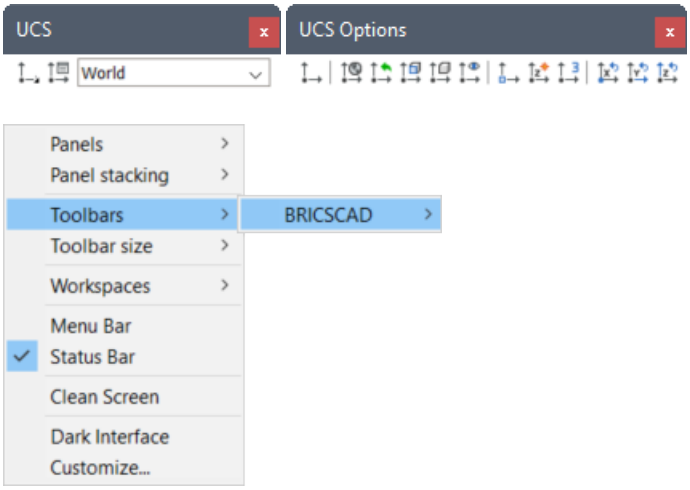
1.2.3 Ribbon

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

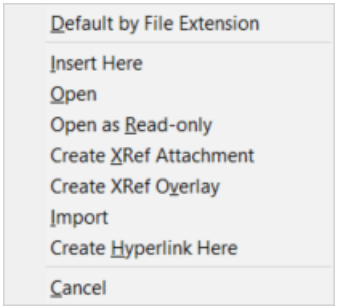
New UCS and UCS Options toolbars offer easy access to UCS commands and options. You an access these toolbars by right-clicking on a toolbar, a panel, or the ribbon and expanding the **Toolbars** > **BricsCAD** menu.



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

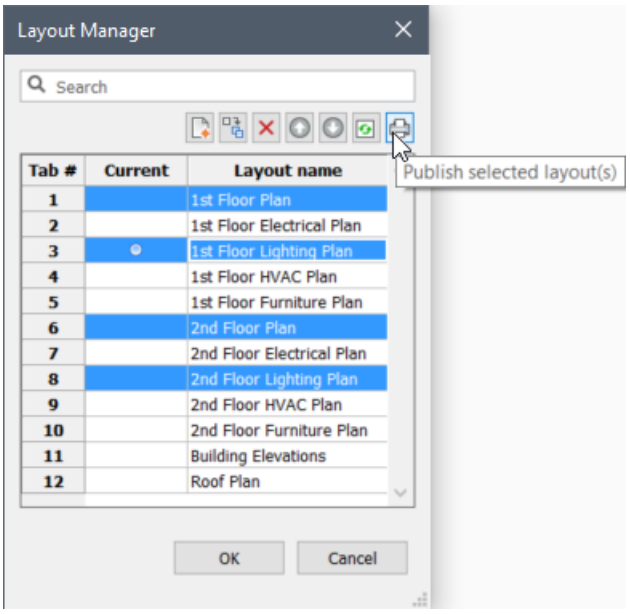
1.2.5 Menus

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**.

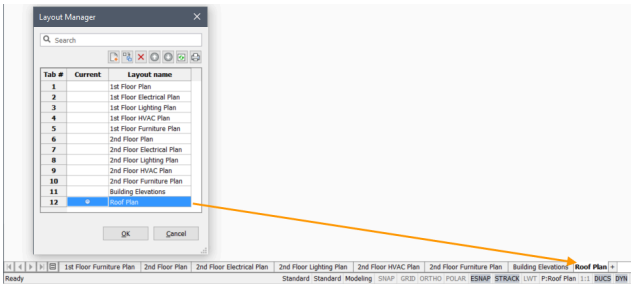


1.2.6 Layouts

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

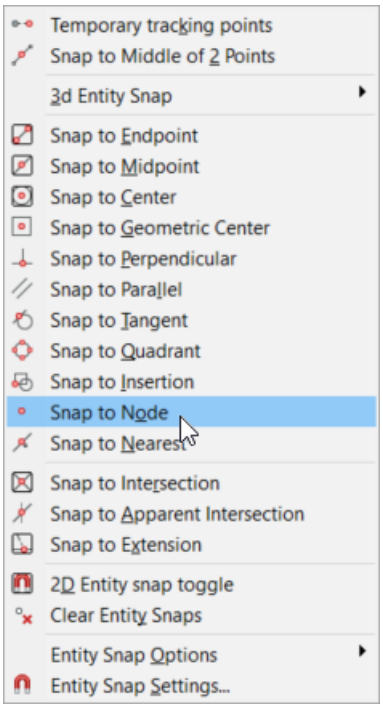


When you set a layout current in the Layout Manager, the active layout tab automatically scrolls to remain visible in the layout tab list.



1.2.7 Entity Snaps

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

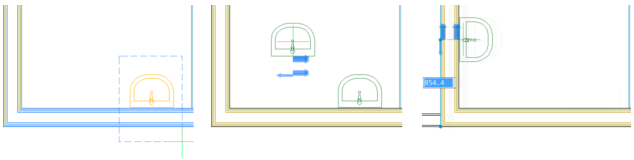


1.3 Productivity

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!

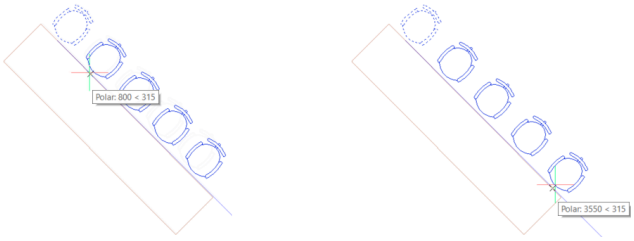


For more flexibility, you can first select the entities you want to move and then launch the MOVEGUIDED command. The pre-selected entities are used for the Move selection set and you are prompted to select entities to use as reference curves.

1.3.2 Copy in a linear array

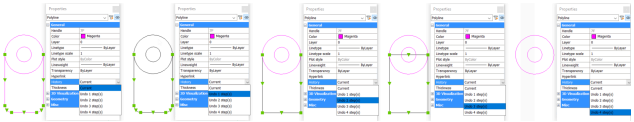
The COPY command now includes a new **Array** option enabling you to copy selected entities in a linear array. The new Array option is available after you specify the base point for the copy operation.

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.



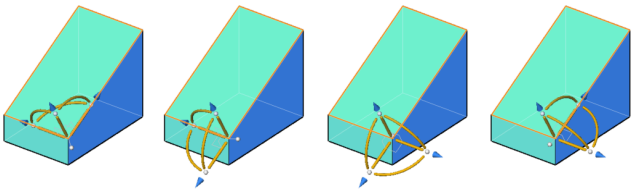
1.3.3 Undo per entity

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

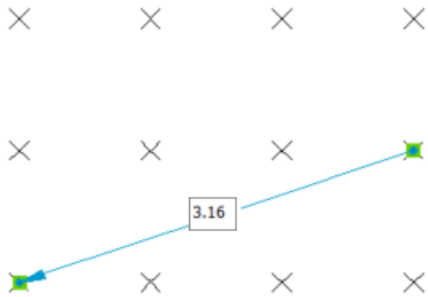
Press the SHIFT key to rotate the Manipulator 90 degrees about its normal axis.



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

1.3.5 Nearest Distance

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

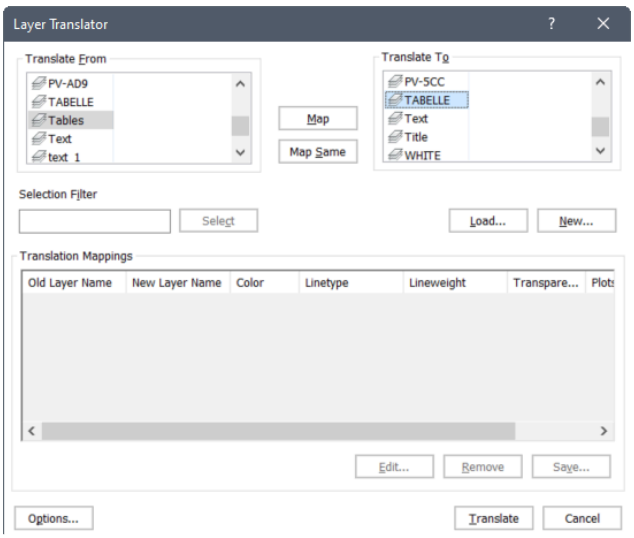


1.3.6 Layers

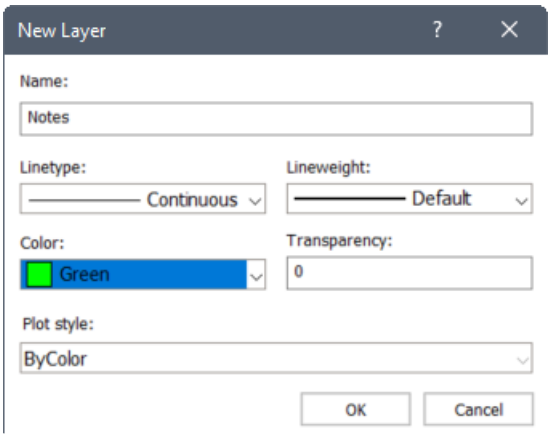
BricsCAD® V21 offers several layer enhancements to help increase your productivity.

Layer Translator

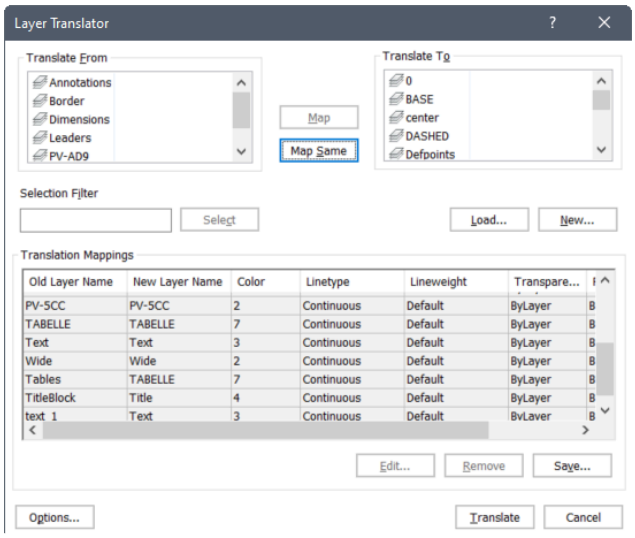
The new LAYTRANS command enables you quickly apply layer names and properties from another drawing file to the current drawing. When you launch the LAYTRANS command, the Layer Translator dialog displays. All the layer names in the current drawing are displayed in the Translate From list. You can specify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

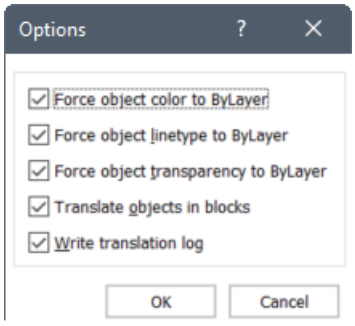


You can select one or more layers from the current drawing, the list on the left, to map to a layer from the list on the right. If you choose **Map Same**, any layer names in the current drawing with corresponding names in the list on the right will inherit the properties from the list on the right.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

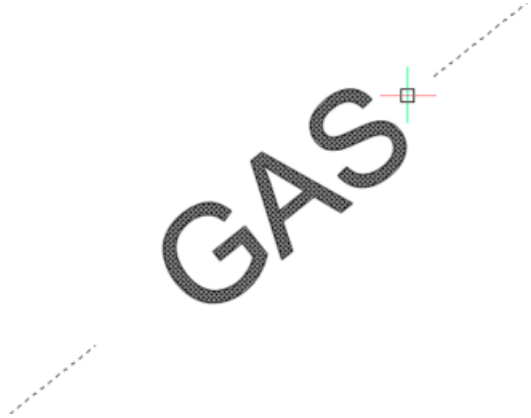
- 128: Transparency

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.

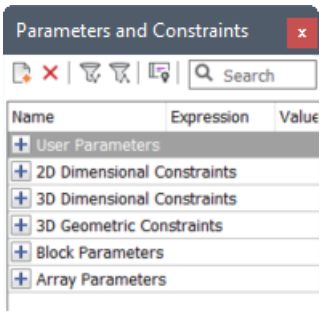


1.3.8 Parameters & Constraints

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

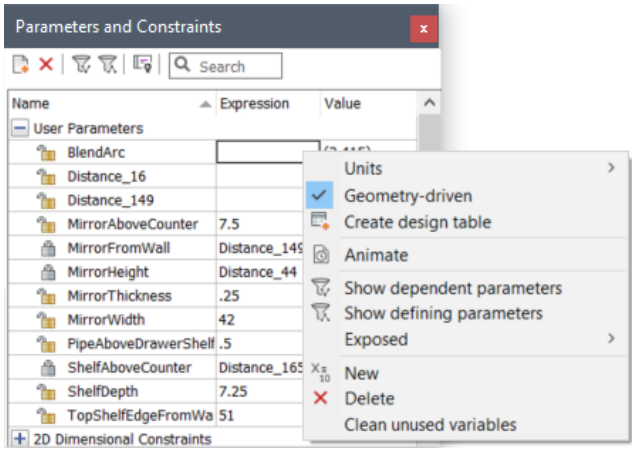
Parameters & constraints panel

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.

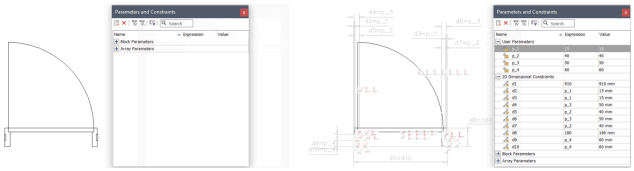


The right-click menu also enables you to control if a parameter is exposed for editing when inserted as a block.

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.

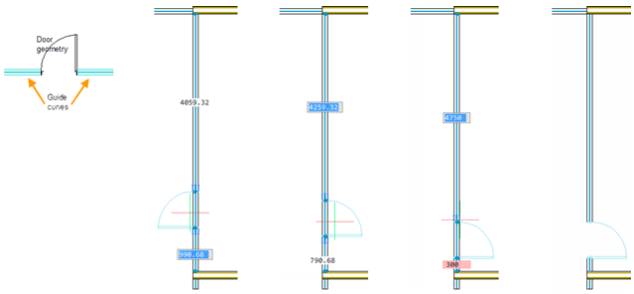


Reference curves

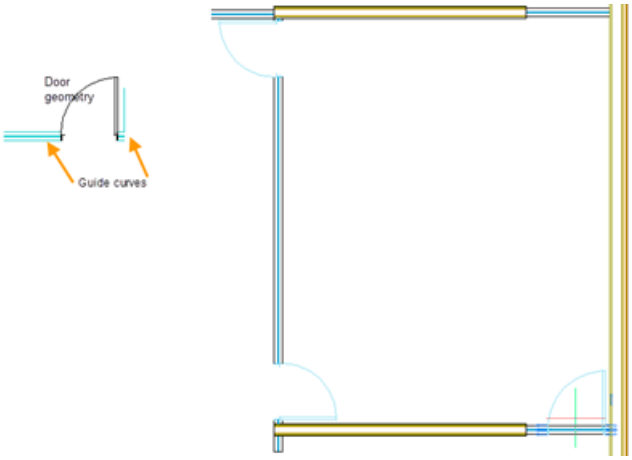
The new REFERENCECURVES command enables you to specify reference geometry for blocks. When you select geometry to use for reference, BricsCAD® creates a REFERENCE_CURVES layer (if it doesn't already exist) and adds the selected entities to that layer.

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.

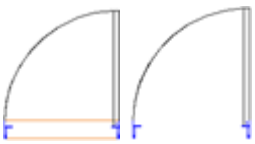


Visibility states

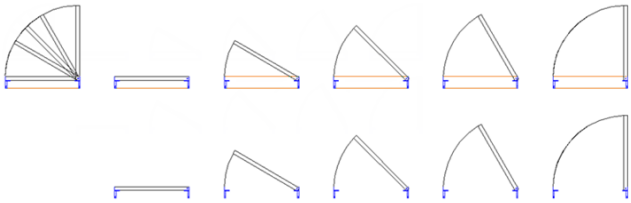
The new VISIBILITYSTATES command enables you to define visibility parameters and states. You can create multiple visibility parameters in a drawing or block definition. And, you can create multiple visibility states in each visibility parameter. When you insert the drawing or block, you can change the 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.

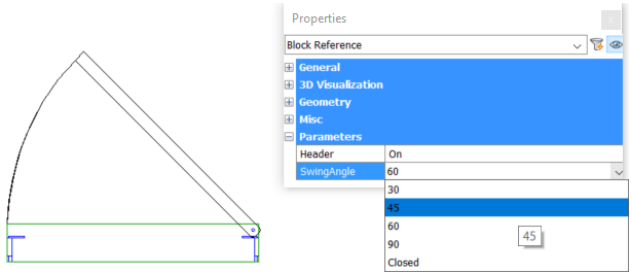
The Header parameter includes two visibility states: **On** and **Off**.



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.

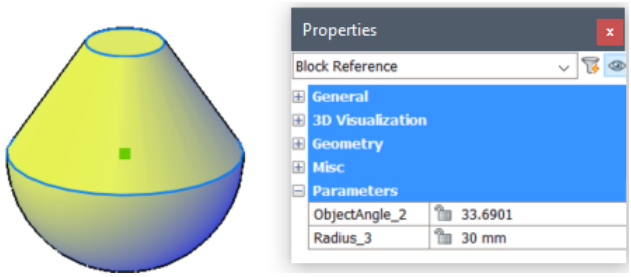


You can easily change the visibility states for a selected block insertion using the Properties panel.



3D Radius constraints

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

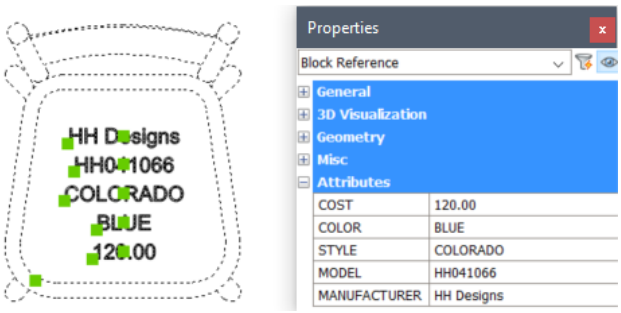


1.3.9 Blocks

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

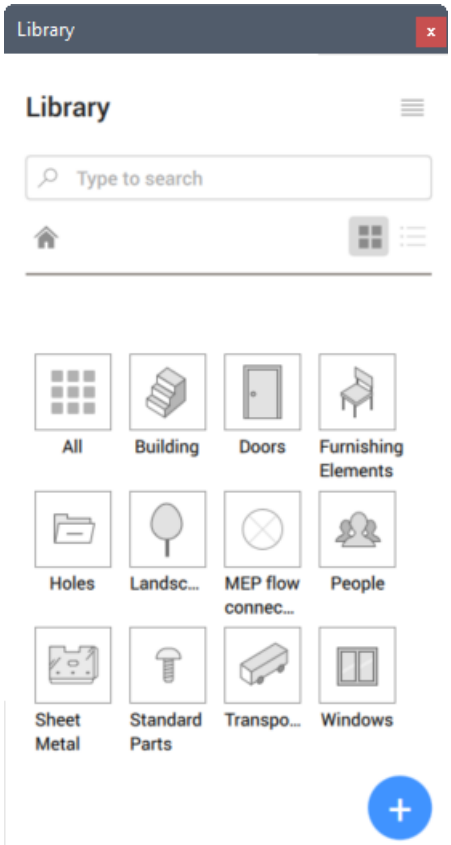
Block attribute properties

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.

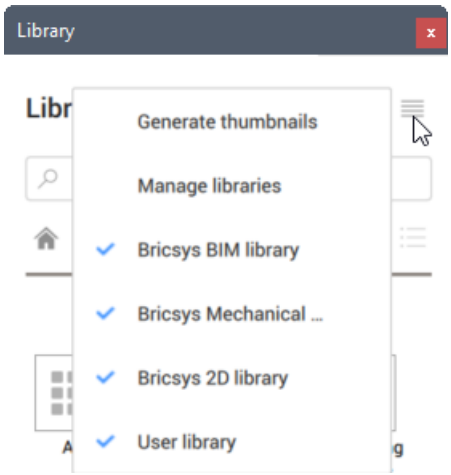


Library panel

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.

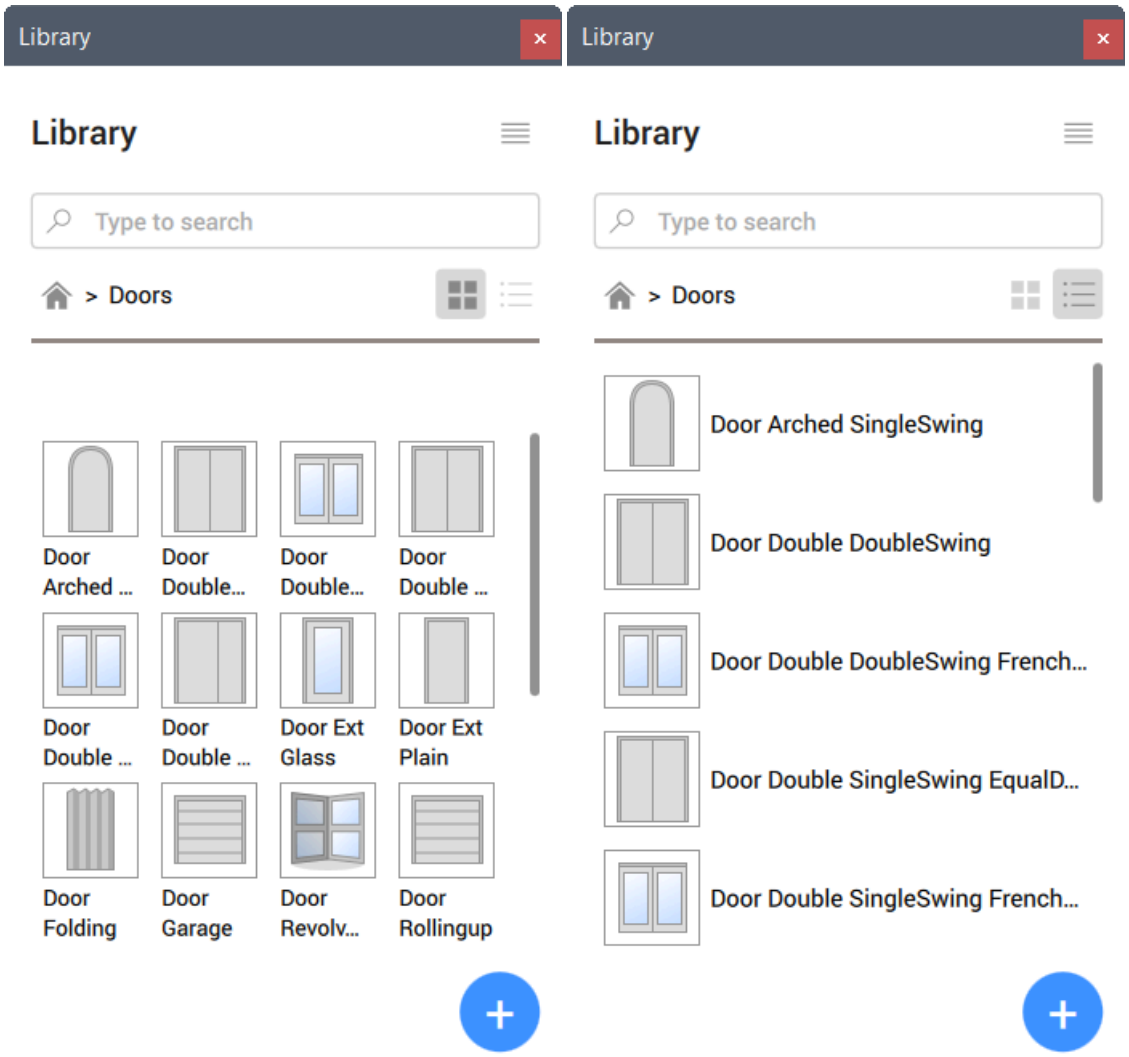


A menu in the upper right corner allows you to control which content is displayed in the Library panel. It may include pre-defined content from Bricsys libraries as well as the User Defined library with your own content.

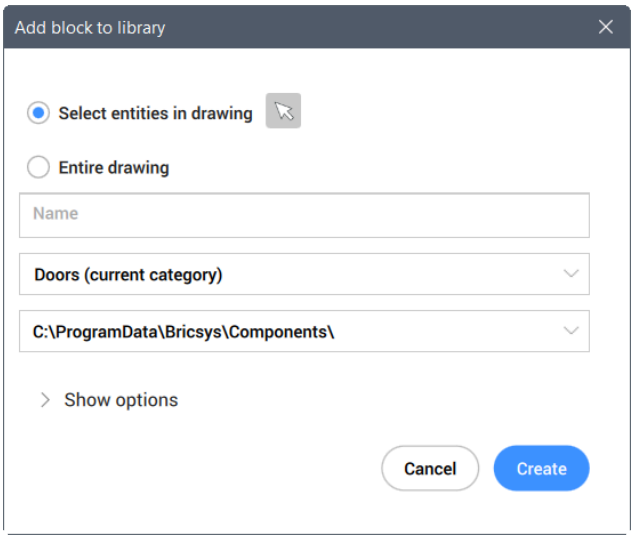


You can specify where BricsCAD® looks for library content by selecting the Manage libraries option. It offers quick access to the COMPONENTSPATH setting so you can view and modify library folder locations. Another menu option automatically generates thumbnail images for any that are missing.

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.

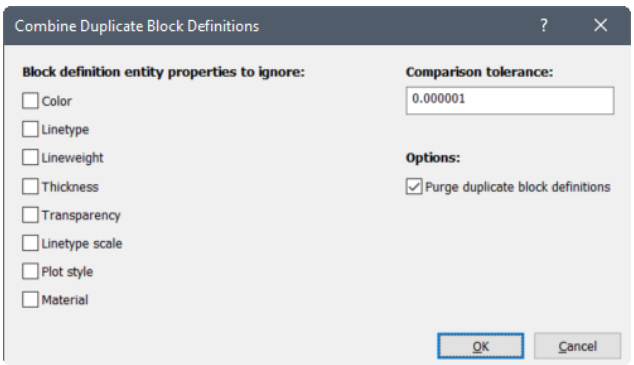
Audit

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

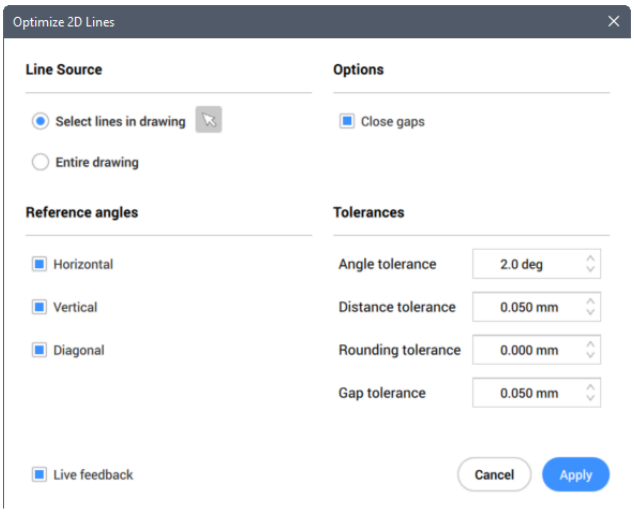
The OVERKILL command enables you to choose if you want to delete duplicate entities or move them to a specified layer. You can specify the layer name using the new OVERKILLLAYER setting or leave the default name, Duplicate Entities.

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.

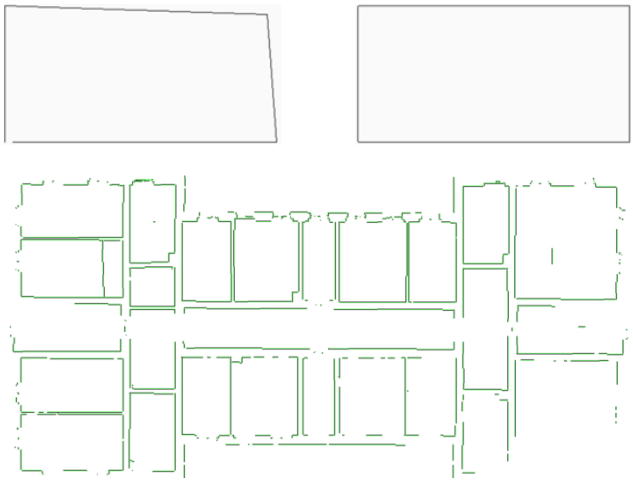


Optimize

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.

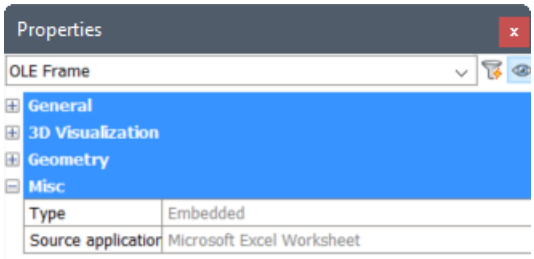


The Optimize tool is useful for cleaning up any 2D lines. For example, you can use it to quickly clean up line geometry extracted from point clouds.



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.

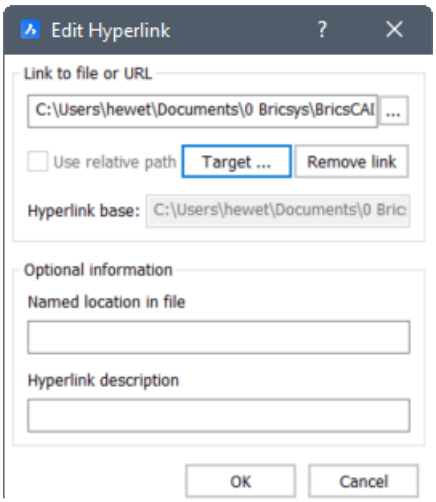


1.3.12 Images

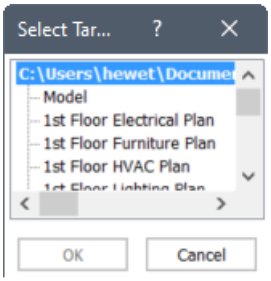
Image manipulation is enhanced in BricsCAD® V21. When you manipulate an attached image using grips and other editing tools the image content displays and updates as you edit the image frame. This enables you to position, align, and scale the image using the image content for reference. The selected image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

1.3.13 Hyperlinks

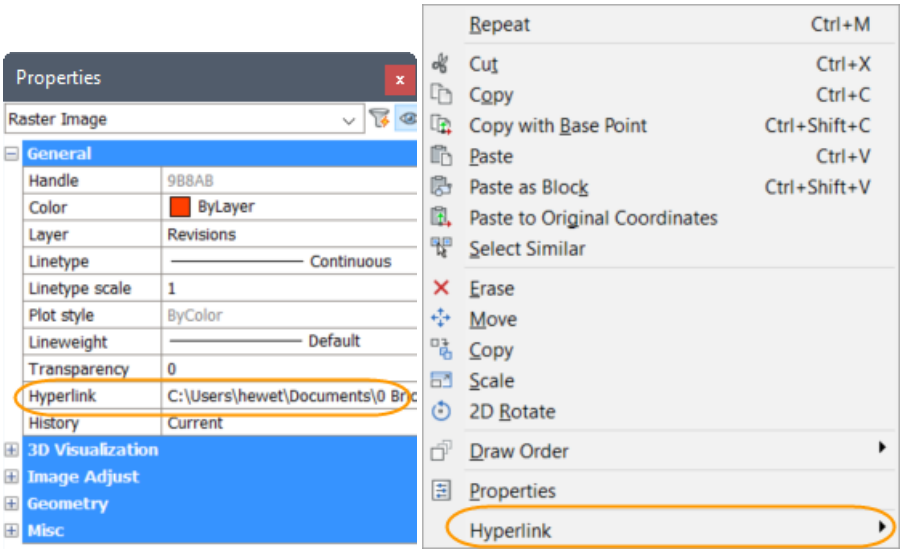
The Edit hyperlink dialog box now enables you to link to layouts and views inside a drawing. After selecting the drawing to which you want to link, choose **Target** to open the Select Target View dialog box.



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.

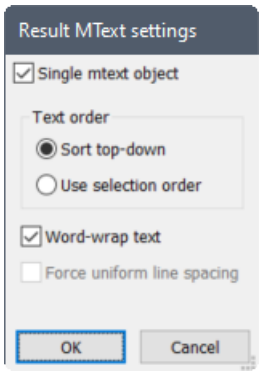


1.4 Drafting

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

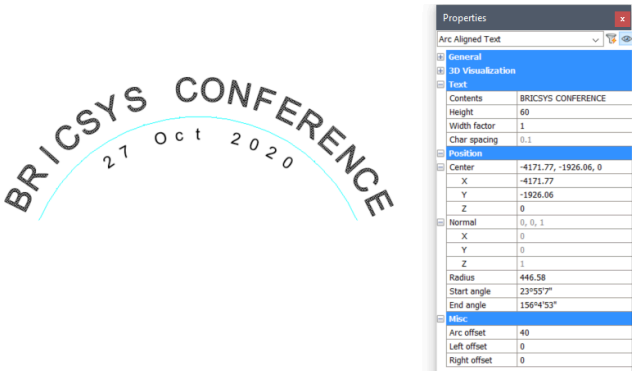
1.4.1 Combine Text to Mtext.

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.



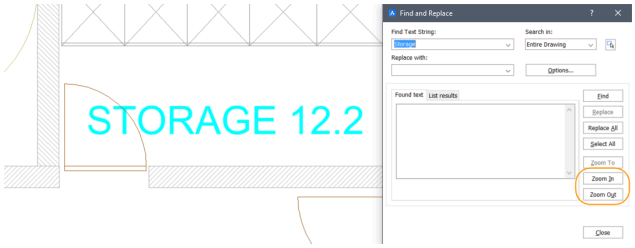
1.4.2 Arc Aligned Text

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



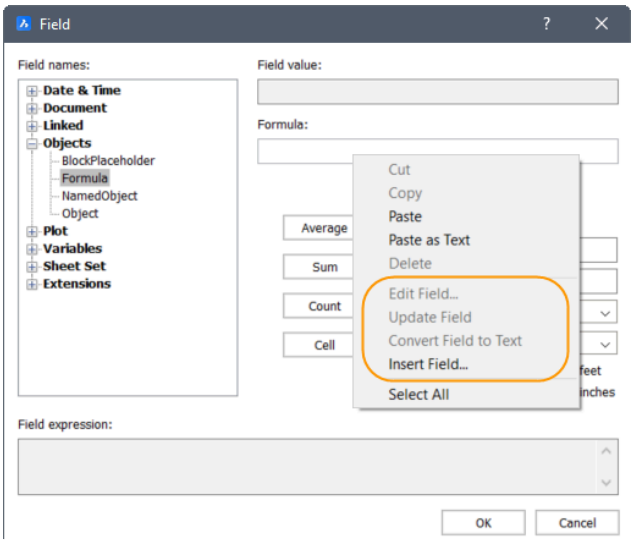
1.4.3 Find and Replace Text

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.

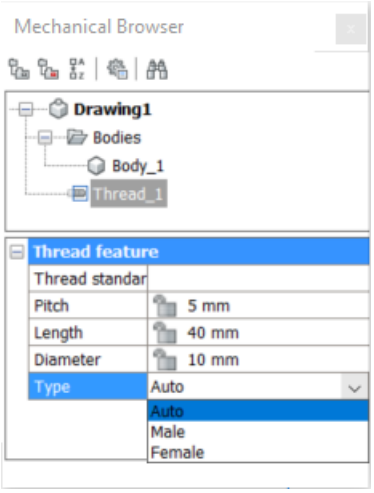
1.5.1 Dynamic UCS

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.

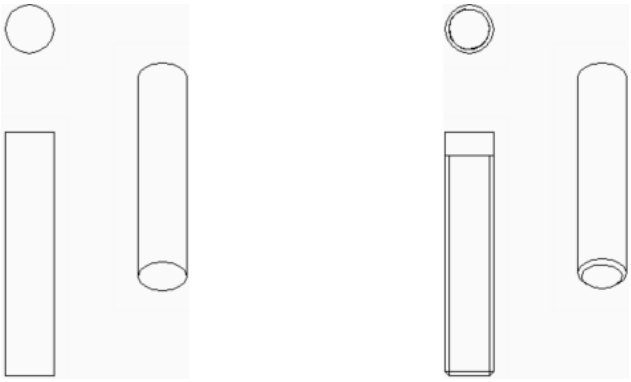
1.5.2 Direct modeling

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.

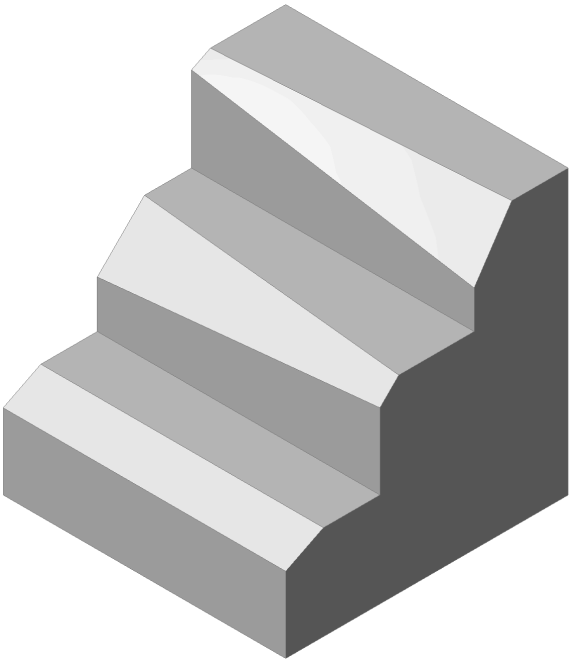


Drawing views created with the VIEWBASE command, the VIEWPROJ command or the VIEWSECTION command represent the threads following drafting standards. The following image show views of a cylinder before and after adding threads.



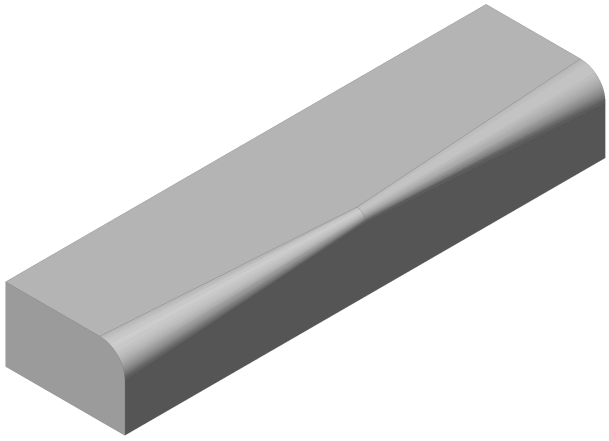
Chamfers

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.



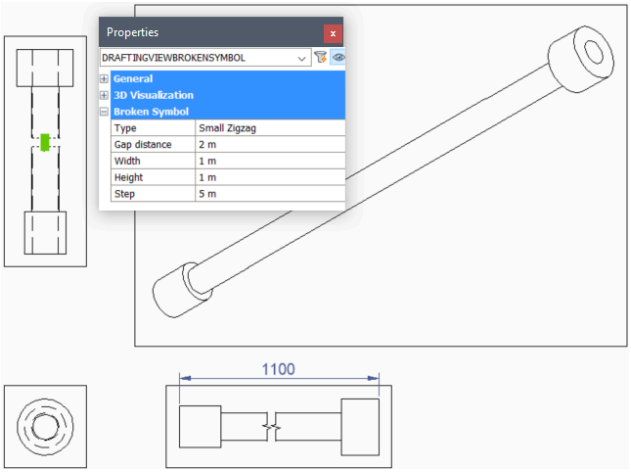
Push Pull

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.

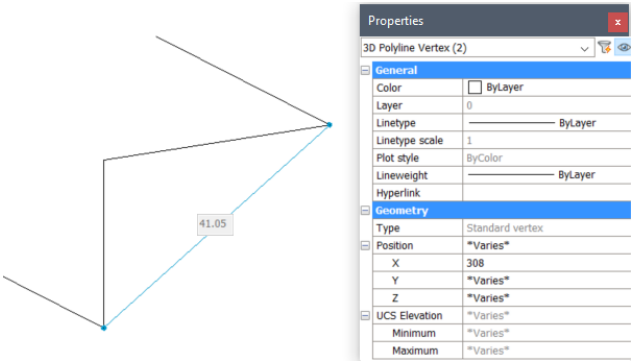


1.5.4 3D Entity snaps

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.

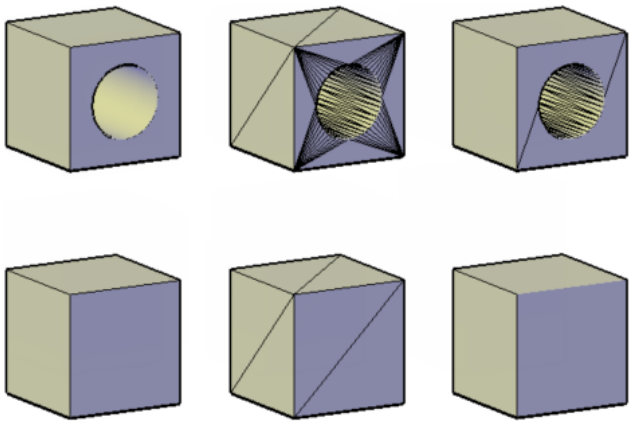
1.5.7 3D display

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

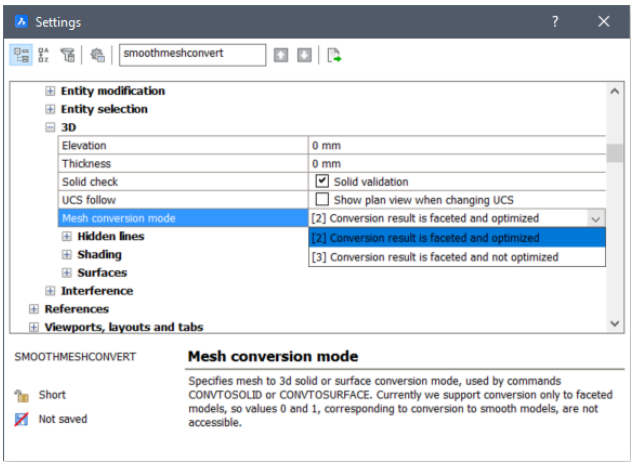
Convert to mesh

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).

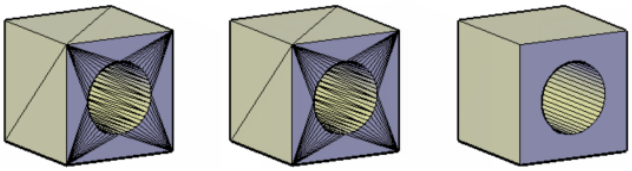


Convert to solid

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

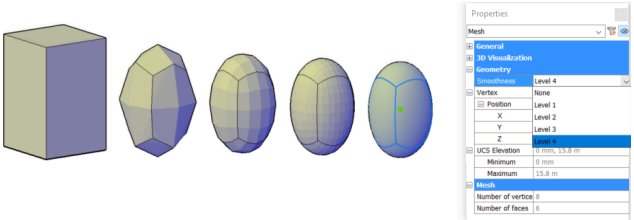


When set to optimized, coplanar and adjacent faces are merged into a single face. As an example, the original mesh on the left is converted to a solid in BricsCAD® V20 (center) and BricsCAD® V21 (right).

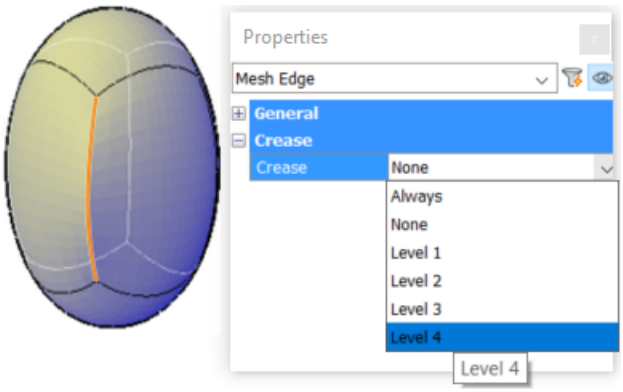


Mesh properties

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

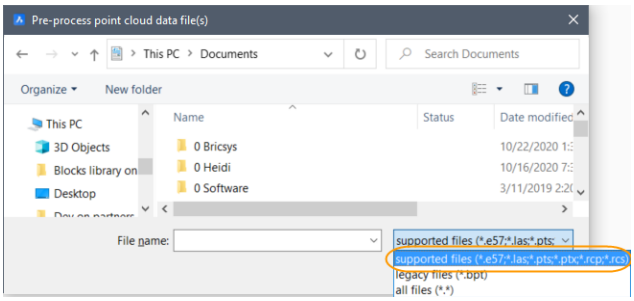
The Animation Editor is enhanced to support the animating of exploded views in model space.

1.5.12 Point clouds

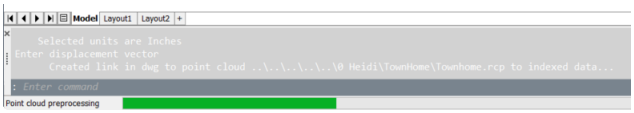
Point cloud functionality is significantly faster and more powerful in BricsCAD® V21.

Pre-processing

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

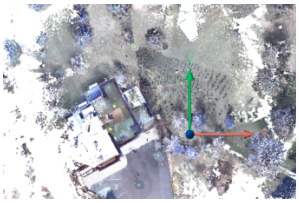


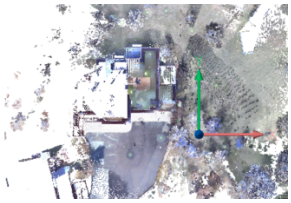
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD® V21 than in V20. A progress meter at the bottom of the display shows the conversion progress.



Alignment

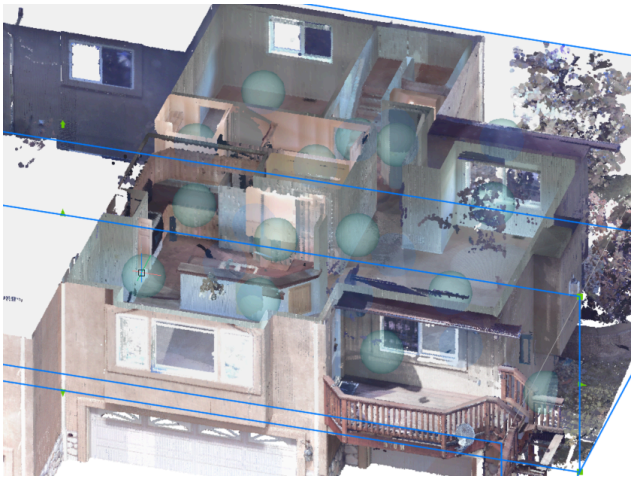
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment it can analyze the entire point cloud or you can specify the most relevant area.





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.

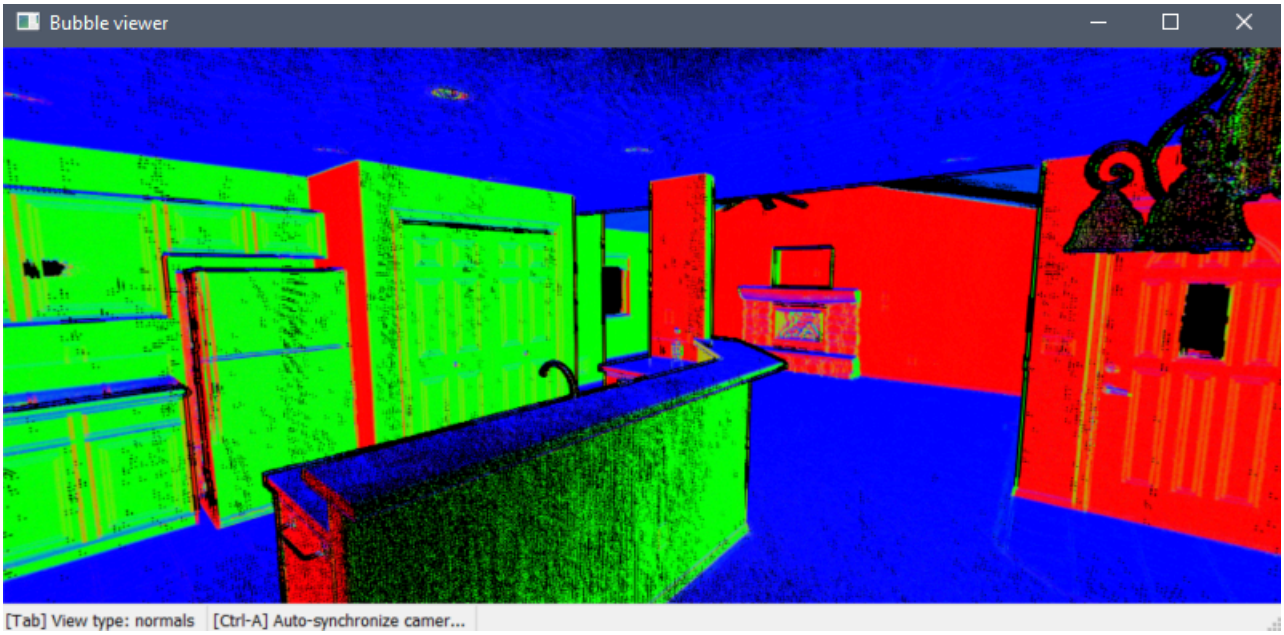


Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location.

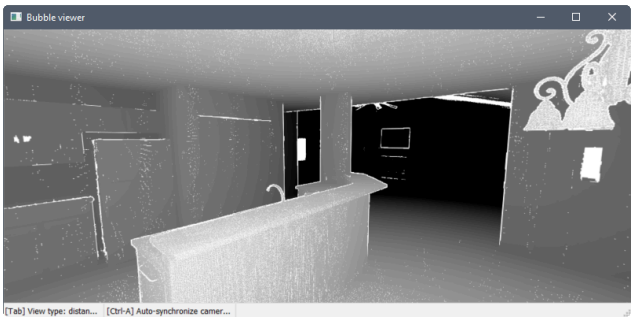
Press the Tab key to cycle between three different visual modes.

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.

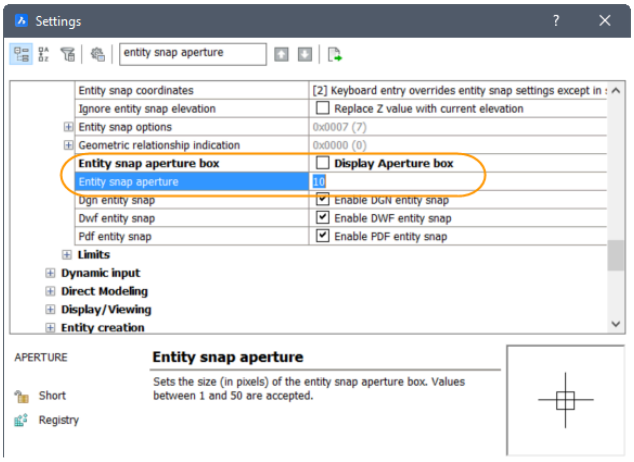


You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.

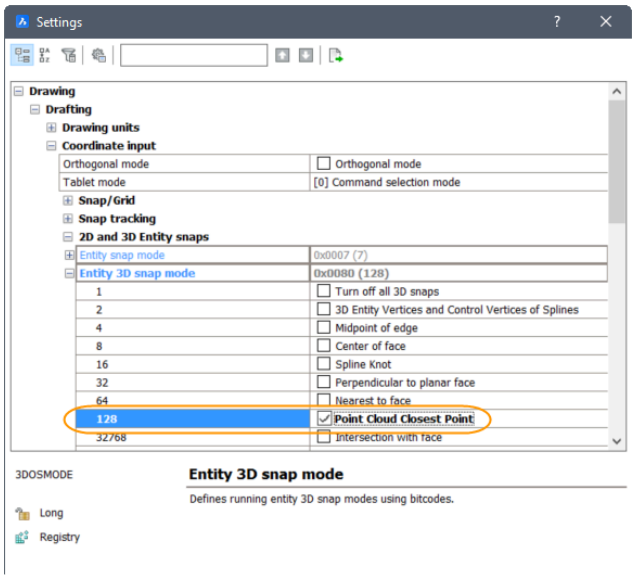
Entity snaps

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

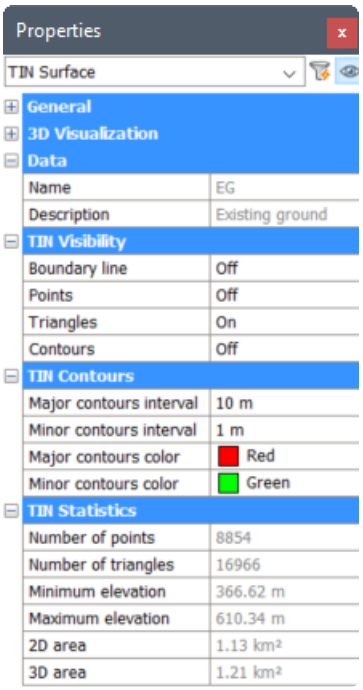
The new CIVIL3DIMPORT command enables you to create multiple civil entities in your BricsCAD® drawing, from an external Autodesk® Civil 3D drawing, in one operation.

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

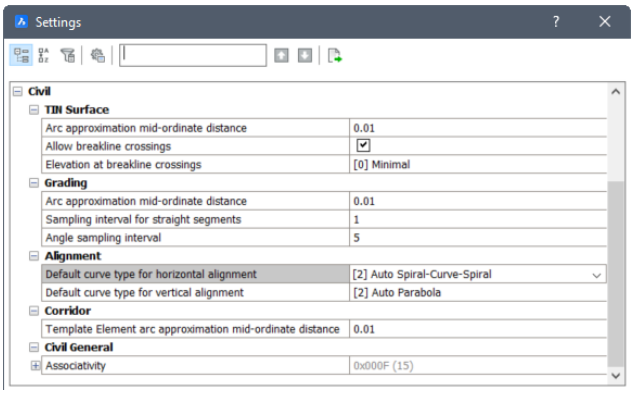
All Civil entities now include Name and Description properties in the Properties panel. Name and description are in the read-only mode in the Properties panel. They can be edited with the API.



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

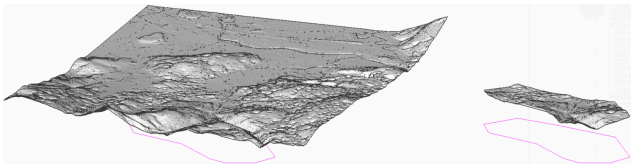
New and enhanced tools for creating and editing TIN surfaces provide more flexibility and a faster workflow.

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.

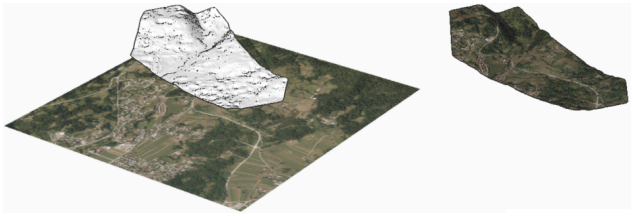
Clipping

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



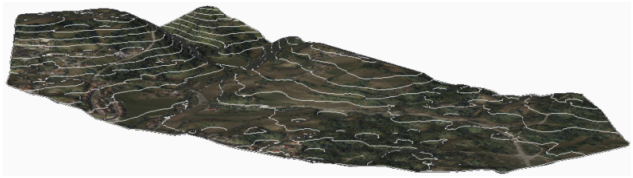
Raster Images

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.

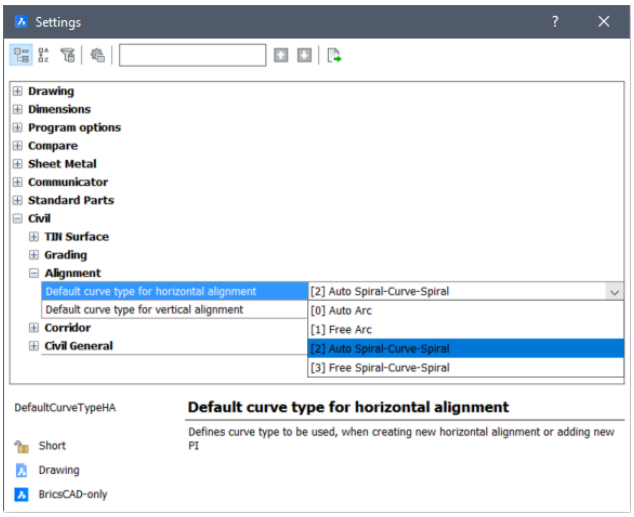
Water Path

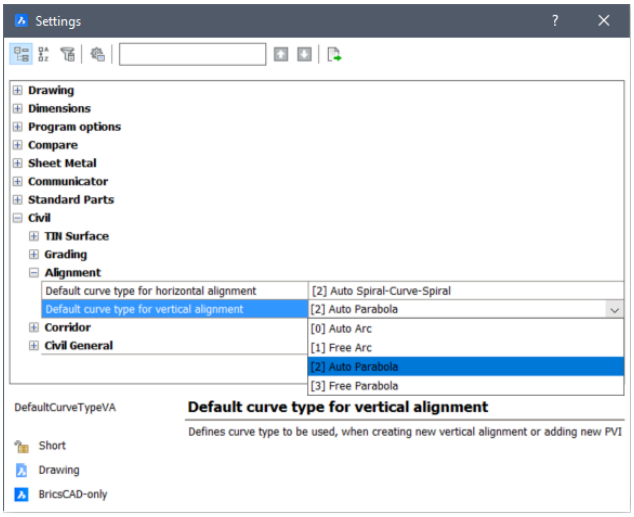
The new TINWATERDROP command enables you to draw a water drop's path from a point on a TIN Surface.

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

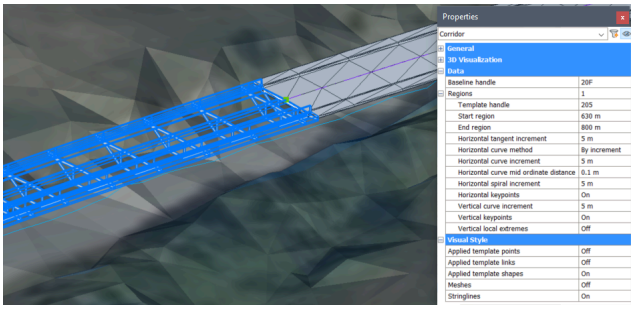
Powerful corridor tools help you create a 3D model of linear infrastructure defined by a template and 3D alignment.

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

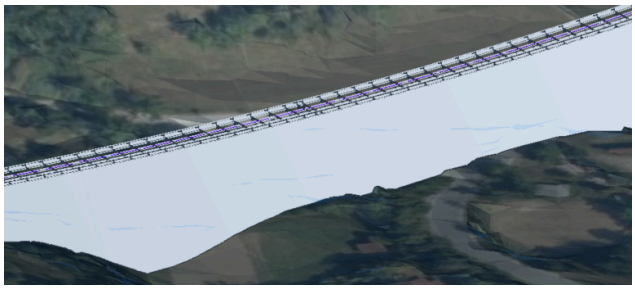
Then use the new CORRIDORTEMPLATE command to create a corridor template from template elements.

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

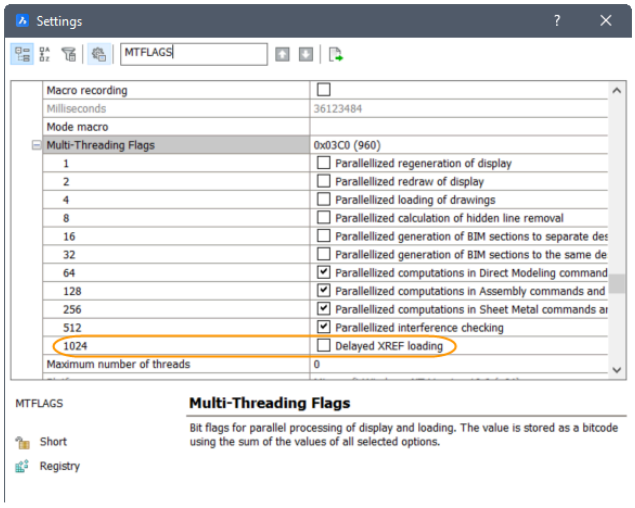
BricsCAD® V21 offers new and enhanced tools to improve performance and compatibility as you work individually or in a collaborative environment.

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.

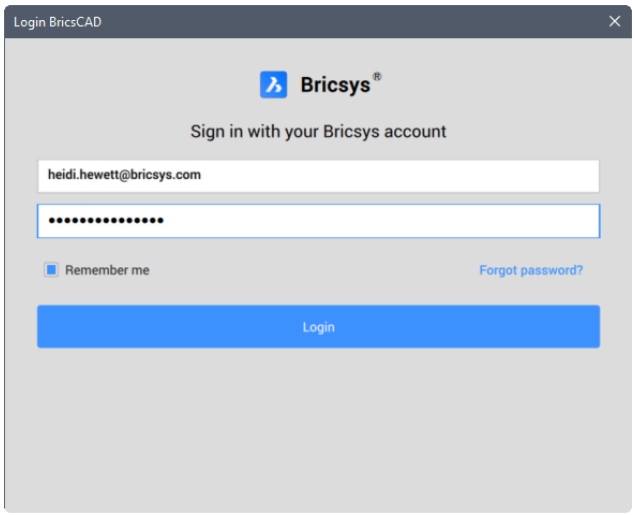
6 Choose Send Support Request.

VERSIONCONTROL Command

The new VERSIONCONTROL command offers the following options:

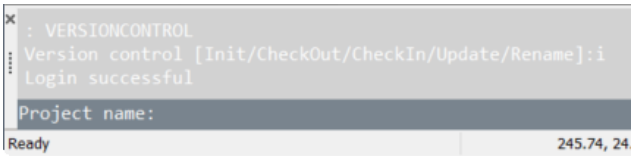
- Init
- Checkout
- Checkin
- Update
- Rename

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

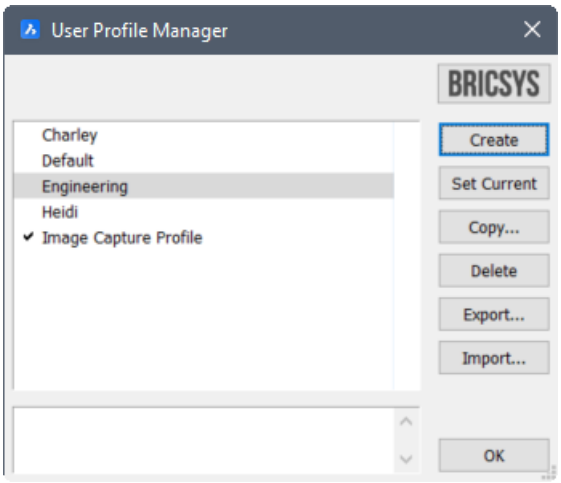
A new DGNEXPORT command enables you to export the current drawing to DGN file format.

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. Installazione e licenza

2.1 Installazione di BricsCAD®

2.1.1 Requisiti Minimi di Sistema e Raccomandazioni

Requisiti di sistema e consigli per PC Microsoft Windows & Linux basati su Intel e AMD, e Apple Mac basati su Intel.

Sistemi Operativi Supportati	• Windows 10 (x64) • Windows 8.1 (x64) • Ubuntu V18.04 o versione successiva • openSUSE builds, successivi ad April 2018 • Fedora builds, successivi ad April 2018 • Mac OSX 10.13 o superiore	***Nota:*** Windows 8.1 raggiungerà la fine del Programma di Supporto Esteso di Microsoft il 10 Gennaio 2023. ***Nota:*** Al momento Mac OSX è supportato solo su hardware basato su Intel.
Sistemi Operativi Deprecati	• Windows Server 2008 R2 • Windows Server 2008 (con SP2 o superiore) • Windows 7 (x64)	***Nota:*** non consigliamo di eseguire software CAD mission-critical su sistemi operativi non supportati, in quanto il PC potrebbe diventare più vulnerabile ai rischi per la sicurezza. • Il 14 Gennaio 2020, Microsoft ha terminato il supporto per Windows Server 2008 e 2008 R2. • Il 14 Gennaio 2020, Microsoft ha terminato l'iniziativa di Supporto Esteso per Windows 7.

CPU minima: • Intel® Core™ i5 • AMD Ryzen™ 5 • 2,5 GHz o superiore	**CPU consigliata:** • Intel® Core™ i7 • Intel® Core™ i9 • AMD Ryzen™ 7 • AMD Ryzen™ 9 • 3,0 GHz o superiore	La CPU con il più alto livello di prestazioni per singolo thread sarà sempre eccellente con **BricsCAD®**. I core aggiuntivi della CPU possono essere un vantaggio, perché **BricsCAD®** supporta il concetto di 'helper threads' per molte attività che fanno un uso intensivo di dati e calcoli. **Nota sulle CPU Intel® Xeon® e AMD Ryzen™ Threadripper™** : queste CPU per workstation di alto livello supportano una grande cache di memoria su chip, più PCIe® I/O lanes and multi-channel, architetture di memoria multicanale ad alta velocità. Aiuteranno **BricsCAD®** a girare più velocemente? Forse. È necessario eseguire test nel proprio ambiente specifico, per determinare se il costo aggiuntivo di questi sistemi è giustificabile.
Memoria minima di sistema: 8 GB	**Memoria di sistema consigliata:** • 16 GB • 32 GB	Per ottenere le massime prestazioni, determinare la configurazione della memoria del PC e assicurarsi che ogni canale di memoria sulla scheda madre sia popolato con almeno un modulo di memoria (SIMMS/DIMM). ***Nota:*** il numero di canali di memoria disponibili potrebbe non essere correlato 1:1 con il numero di socket RAM sulla scheda madre del PC. Domande? Verificare con il fornitore dell'hardware prima di acquistare la memoria di sistema.
Spazio Minimo su Disco: 2 GB per **BricsCAD® Ultimate**, installato	**Spazio Consigliato su Disco:** (In base alle proprie esigenze)	Consigliamo Dischi allo Stato Solido (SSD) come unità di sistema, sia per **BricsCAD®**, sia per Windows. I disegni e i file di supporto possono risiedere su dischi rigidi meccanici più lenti, se necessario. ***Nota:*** I dischi allo stato solido possono avere una lettura fino a 10 volte più veloce e una scrittura di dati fino a 20 volte più veloce rispetto a un'unità disco rigido di fascia media (ad esempio Seagate Barracuda 5.4k RPM, SATA 6.0).

Sistema Grafico Minimo: • Display True Color 1920 x 1080 e scheda grafica con 1 GB di VRAM • Display Apple Mac	**Sistema Grafico Consigliato:** • Display True Color 3840 x 2160 (4k), con un PCIe® scheda grafica con w/GPUs e 4 GB di VRAM • Display Apple Mac Retina (4K e 5K)	Nei sistemi Microsoft Windows, la maggior parte delle schede grafiche e GPU di NVIDIA, AMD e Intel® sono supportate. Nei sistemi Linux, la maggior parte delle schede grafiche e GPU di NVIDIA e AMD sono supportate. Nei sistemi Mac, è supportato l'hardware Apple nativo. Le modalità di visualizzazione non wireframe di **BricsCAD®** sono generate dalla tecnologia RedSDK di Redway3D®. Per ottenere le massime prestazioni RedSDK in queste modalità, è necessaria una GPU. **Nota:** si consiglia di installare gli ultimi driver disponibili per l'hardware della scheda video, come raccomandato dal produttore. **Nota per gli utenti Linux:** l'accelerazione hardware della grafica 3D sui sistemi Linux NON è supportata su: • Intel® schede grafiche/chipset • Notebook con due schede grafiche. Tuttavia, i driver sviluppati recentemente da parte di NVIDIA mostrano una promessa in quest'area. Per ulteriori informazioni sull'hardware grafico supportato per **BricsCAD®**, vedere la pagina di riferimento del chipset GPU Redway3D.

2.1.2 Scaricare BricsCAD®

2.1.2.1 Scaricamento della versione più recente di BricsCAD®

1 Se non l'hai già fatto, sfoglia il sito Bricsys, registrati ed effettua l'accesso.

2 Clicca su **Scarica BricsCAD**.

3 Seleziona la piattaforma, la lingua e la versione (32-bit o 64-bit).

4 Seleziona la casella di controllo **Sono d'accordo con i termino di utilizzo**.

5 Fai clic sul pulsante **Scarica**.

2.1.2.2 Download di una versione precedente di BricsCAD®

1 Se non l'hai già fatto, sfoglia il sito Bricsys, registrati ed effettua l'accesso.

2 Clicca su **Scarica BricsCAD**.

3 Fai clic sul link **Vecchie versioni**.

4 Seleziona la piattaforma e la lingua.

5 Seleziona la versione di **BricsCAD®**.

6 Seleziona la versione (32-bit o 64-bit).

7 Seleziona la casella di controllo **Sono d'accordo con i termino di utilizzo**.

8 Fai clic sul pulsante **Scarica**.

2.1.3 Panoramica delle opzioni di installazione

- **BricsCAD®** viene fornito come un unico file di installazione per tutti i livelli di licenza, tra cui anche **BricsCAD® Shape**.
- Quando si installa una nuova versione di **BricsCAD®** per la prima volta, è possibile eseguirlo in modalità prova per 30 giorni.
- È possibile installare ogni nuova versione principale accanto a una versione principale precedente.

2.1.3.1 Installazione interattiva

In caso di installazione normale (non silenziosa), l'utente fornisce l'input necessario per l'installazione tramite finestre di dialogo.

L'interfaccia utente di Windows Installer interroga il sistema di destinazione e visualizza Installazione guidata. L'installazione guidata consente all'utente di modificare varie opzioni che influiscono sull'installazione.

2.1.3.2 Installazione silenziosa

In caso di installazione invisibile o non presidiata, l'interfaccia utente dell'Installazione guidata è disabilitata e le azioni durante la sequenza dell'interfaccia utente non vengono eseguite.

2.1.3.3 Installazione interattiva

1 Fare doppio clic sul file del programma di installazione.

2 Fare clic su **Avanti**.

3 Selezionare la casella di controllo per accettare il **Contratto di licenza**.

4 Fare clic su **Avanti**.

5 Eseguire una delle seguenti operazioni:
   - Accettare la cartella di installazione predefinita.
   - Fare clic sul pulsante **Cambia...** e selezionare una cartella di installazione.

6 Fare clic su **Avanti**.

7 Fare clic su **Installa** per avviare l'installazione.

8 Scegliere se aggiungere un collegamento sul desktop e/o visualizzare le note di rilascio al termine del processo di installazione.

9 Fare clic su **Avanti**.

10 Fare clic su **Fine** per completare il processo di installazione.

Result: **BricsCAD®** è installato.

2.1.3.4 Installazione Silenziosa (solo per Windows)

Per l'installazione invisibile, sono necessari i diritti di amministratore.

1 Aprire **PowerShell** o il **Prompt dei Comandi** e modificare la directory (cd) del file di installazione .msi di **BricsCAD®**.

Nota: In Windows 10, è possibile accedere al menu contestuale **Apri finestra PowerShell qui** quando si fa clic con il tasto destro del mouse su una cartella nell'**Explorer**. In questo modo è possibile aprire **PowerShell** nella finestra corrente senza dover modificare la directory.

2 Digitare **msiexec /i** ed effettuare una delle seguenti operazioni:

- Digitare il nome del file del programma di installazione di **BricsCAD®** (ad esempio BricsCAD-V20.2.09-1-it_IT(x64).msi.
- Trascinare il file del programma di installazione di **BricsCAD®** dalla finestra di dialogo Esplora File alla finestra dei comandi.

3 Al prompt dei comandi digitare **/qn**.

4 (opzione) Aggiungi **ADDDESKTOPSHORTCUT=""** per sopprimere l'aggiunta di un collegamento sul desktop.

5 (opzione) Aggiungi **SHOWRELEASENOTES=""** per eliminare la visualizzazione delle Note di Rilascio al termine dell'installazione.

6 (opzione) Aggiungi **APPLICATIONFOLDER="la_tua_cartella_di_installazione"** per installare **BricsCAD®** in una cartella personalizzata.

7 (opzione) Aggiungere **BRXLICENSEDESTFILE="percorso_per_file_lic"** per specificare una cartella per il file di licenza se una delle proprietà **BRXLICENSEKEY**, **BRXLICENSEFILE** o **BRXLICENSERVER** sono definite.

L'impostazione predefinita è **APPLICATIONFOLDER\BricsCAD.lic**.

8 (opzione) Aggiungi **BRXLICENSEKEY="tuo_file_di_licenza"** per attivare la licenza e memorizzarla nella cartella specificata da **BRXLICENSEDESTFILE**.

9 (opzione) Aggiungere **BRXLICENSEFILE="tuo_file_di_licenza"** per copiare il file di licenza nella cartella specificata da **BRXLICENSEDESTFILE**.

10 (opzione) Aggiungi **BRXLICENSESERVER="host"** o **BRXLICENSESERVER="port@host"** (*) per creare un file di licenza nella cartella specificata da **BRXLICENSEDESTFILE**, contenente le specifiche dell'host del server.

(*) Se il server delle licenze utilizza una porta diversa dalla porta predefinita 5053.

11 (opzione disponibile solo su 0 BIT) Aggiungi **INSTALLVBA="0"** o **INSTALLVBA=""** per non installare VBA.

La voce completa nel prompt dei comandi può essere: **msiexec /i "BricsCAD-Vxx.x.xx-x-en_US(x64).msi" /qn ADDDESKTOPSHORTCUT="" SHOWRELEASENOTES=""**.

12 Premere **Invio** per avviare l'installazione.

Nota:

- Per ulteriori opzioni, digitare: **msiexec /?** nel prompt dei comandi.
- Sul sito Web di Microsoft, è possibile trovare ulteriori informazioni sul comando **msiexec**.

2.1.4 Modalità Trial

BricsCAD® può essere eseguito in modalità di prova per 30 giorni. Dopo questo periodo, è necessario attivare una valida licenza commerciale.

2.1.4.1 Esecuzione di BricsCAD in modalità trial

Ogni volta che avvi **BricsCAD®** in modalità prova, si apre la finestra di dialogo **BricsCAD - Trial Gratuita** che indica i giorni rimanenti.

Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Continua** per aprire **BricsCAD®** in modalità di prova.
- Cliccare sul pulsante **Inserisci Licenza...** per attivare una licenza Singola o Volume oppure per utilizzare una licenza di Rete.

2.1.4.2 Impostazione del livello di licenza

Quando si esegue **BricsCAD®** in modalità prova, sono disponibili tutte le funzionalità. Viene eseguito come licenza **BricsCAD® Ultimate**.

Il comando SETLICENSELEVEL consente di selezionare un livello di licenza inferiore.

2.2 Attivazione e licenza BricsCAD®

2.2.1 Panoramica delle opzioni di licenza

BricsCAD® offre diverse opzioni di licenza.

2.2.1.1 Livelli di licenza

Esistono 5 livelli di licenza per **BricsCAD®**:

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

1 BricsCAD® Lite

Se si creano solo progetti 2D generici, **BricsCAD® Lite** potrebbe essere la scelta giusta per te. Rispetto a **AutoCAD LT®**, consente anche di visualizzare, spostare, copiare o specchiare modelli 3D. Tuttavia, non è possibile modificare il modello o creare nuove entità 3D. **BricsCAD® Lite** esegue anche routine Lisp. Il

Lisp consente agli utenti **BricsCAD®** di caricare routine personalizzate o piccole applicazioni che verranno eseguite in **BricsCAD®**.

2 BricsCAD® Pro

BricsCAD® Pro include la modellazione 3D diretta con un sistema di vincoli 3D completo, il rendering con materiali e illuminazioni e l'accesso alle applicazioni di terze parti.

3 BricsCAD® BIM

BricsCAD® BIM include tutte le funzionalità di **BricsCAD® Pro**. È possibile utilizzare le attuali competenze CAD per passare senza problemi alla creazione, in tempo record, di Modelli Informativi degli Edifici (Building Information Models). Un prodotto, un flusso di lavoro, tutto in DWG standard del settore.

4 BricsCAD® Mechanical

Oltre a tutte le funzionalità di **BricsCAD® Pro**, **BricsCAD® Mechanical** include un completo kit di strumenti per la progettazione meccanica, tutto incluso nello standard di settore DWG, che include le funzionalità per la progettazione della lamiera metallica.

5 BricsCAD® Ultimate

BricsCAD® Ultimate combina le funzionalità di **BricsCAD® BIM** e **BricsCAD® Mechanical**.

2.2.1.2 Tipi di licenza

Esistono 2 tipi di licenza per **BricsCAD®**:

- Licenza perenne.
- Licenza a noleggio.

1 Licenza perenne

Quando si acquista una licenza perenne, si è proprietari del software. Le edizioni di **BricsCAD®** vengono eseguite su Microsoft Windows, macOS e Linux.
Sono disponibili 2 opzioni di manutenzione:

- L'opzione manutenzione include gli aggiornamenti automatici e il supporto tecnico.
- È possibile acquistare gli upgrade a propria discrezione.

2 Licenza a noleggio

Quando scegli l'opzione del noleggio, hai accesso all'ultima versione di **BricsCAD®** per 1 anno. Include il supporto prioritario e l'accesso a tutte le nuove versioni rilasciate durante il periodo in cui il noleggio è attivo.

2.2.1.3 Ulteriori opzioni di licenza

Ulteriori opzioni di licenza includono licenze accademiche, singole, di rete e volume.

1 Licenza Accademica

Studenti, facoltà, docenti e istituti scolastici hanno accesso gratuito a tutti i software **BricsCAD®** per 12 mesi. E' possibile rinnovarlo annualmente con un ID studente valido. Registrati come studente o scuola/docente con le tue informazioni accademiche.

2 Licenza per utente-singolo

Il contratto di licenza per un singolo-utente è destinato a un singolo utente. Ogni licenza per utente-singolo consente 2 attivazioni. È possibile installarlo su 2 macchine, ad esempio su una workstation in ufficio e su un laptop in cantiere. Tuttavia, è consentito l'utilizzo di una 1 licenza alla volta. Le licenze per singolo-utente sono disponibili in licenza perenne o a noleggio. Le licenze accademiche sono sempre licenze per utente-singolo.

3 Licenza di rete

L'accordo di licenza di rete consente a più utenti di accedere a **BricsCAD®** all'interno di una LAN (Local Area Network). Il numero di licenze di rete disponibili definisce il numero di utenti che possono utilizzare contemporaneamente il software. L'opzione di licenza di rete è disponibile per licenze perenni o a noleggio e include il contratto di manutenzione per 1 anno. Le licenze di rete sono disponibili anche per gli istituti che utilizzano una licenza accademica.

4 Licenza volume

Il contratto di licenza volume consente di utilizzare **BricsCAD®** all'interno di un numero di utenti definito. È possibile installare il software utilizzando un unico codice di licenza valido per ogni utente. L'opzione di licenza volume è disponibile per licenze perenni o a noleggio e include un contratto di manutenzione per 1 anno. È l'impostazione predefinita per le licenze accademiche istituzionali.

2.2.2 BricsCAD® e AutoCAD® comparazione funzionalità

BricsCAD® vs **AutoCAD®** comparazione funzionalità.

Disegni							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Formato file Nativo DWG (AutoCAD 2021)	√	√	√	√	√	√	√

Disegni							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Protezione Password	√	√	√	√	√	o	o
eTransmit	√	√	√	√	√	√	√
Gestione Gruppo di Fogli	√	√	√	√	√	√	√
Posizione Geografica	√	√	√	√	√	√	√
Confronto Disegno	√	√	√	√	√	√	√
Collaborazione Cloud	√	√	√	√	√	√	√

Interfaccia							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Tema a Colore Scuro e Chiaro	√	√	√	√	√	√	√

Interfaccia							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Comandi standard del settore, alias e variabili	√	√	√	√	√	√	√
Barra dei Comandi (Completamento Automatico, Interno Stringa, Opzioni cliccabili)	√	√	√	√	√	√	√
Scheda Inizio	√	√	√	√	√	√	√
Schede File (Disegno)	√	√	√	√	√	√	√
Schede Modello e Layout	√	√	√	√	√	√	√
Gestore Layout	√	√	√	√	√	o	o
Barra dei menu	√	√	√	√	√	√	√

Interfaccia							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Barre degli Strumenti	√	√	√	√	√	√	√
Barra Multifunzione	√	√	√	√	√	√	√
Tavolozze degli Strumenti	√	√	√	√	√	√	√
Menu Contestuali con il Tasto Destro	√	√	√	√	√	√	√
Menu Quad contestuale	√	√	√	√	√	o	o
Barra di Stato	√	√	√	√	√	√	√
Aree di Lavoro	√	√	√	√	√	√	√
Finestra di dialogo Impostazioni (Variabili) ricercabili	√	√	√	√	√	o	o

Interfaccia							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Interfaccia Bloccabile	√	√	√	√	√	√	√
Modalità Schermo Intero	√	√	√	√	√	√	√
Esplora Disegno	√	√	√	√	√	o	o
Interfaccia Utente Personalizzabile (CUI)	√	√	√	√	√	√	√
Navigatore Struttura	√	√	√	√	√	o	o

Selezione, snap & puntamento							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Filtra Selezione (SELERAPID)	√	√	√	√	√	√	√
Aggiungi Selezionati	√	√	√	√	√	√	√

Selezione, snap & puntamento							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Selezione a Simile	√	√	√	√	√	√	√
Selezione Ciclica	√	√	√	√	√	√	√
Snap Entità (Osnaps)	√	√	√	√	√	√	√
Puntamento Polare	√	√	√	√	√	√	√
Snap 3D Zintersezione	√	√	√	√	√	o	o
Distanza Più Vicina	√	√	√	√	√	o	o
Snap alla Griglia Adattiva	√	√	√	√	√	o	o
Individuazione Real time di Contorni	√	√	√	√	√	o	o
UCS Dinamico 2D	√	√	√	√	√	o	o
UCS Dinamico 3D	√	√	√	√	√	o	√

Strumenti di Disegno							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Linea, Polilinea, Polilinea 3D, Spline, Raggio, Xlinea	√	√	√	√	√	√	√
Multilinee e Stili multilinea	√	√	√	√	√	o	√
Cerchio, Arco, Ellisse, Arco Ellittico	√	√	√	√	√	√	√
Poligono, Rettangolo, Regione, Anello, Punto	√	√	√	√	√	√	√
Elica	√	√	√	√	√	o	√
Parametri 2D, Vincoli	√	√	√	√	√	o	√
Autovincolo 2D	√	√	√	√	√	2	2

Strumenti Annotazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Quote, Stili di Quota	√	√	√	√	√	√	√
Multidirettrici, Stili Multidirettrici	√	√	√	√	√	√	√
Testo e Testo Multilinea, Stili di Testo, Editor di Testo	√	√	√	√	√	√	√
Tratteggi, Gradienti, Editor Tratteggio/ Gradienti	√	√	√	√	√	√	√
Entità Coprente	√	√	√	√	√	√	√
Fumetto di Revisione	√	√	√	√	√	√	√
Attributi, Editor Attributi	√	√	√	√	√	√	√
Tabelle, Stili Tabella	√	√	√	√	√	√	√

Strumenti Annotazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Campi	√	√	√	√	√	√	√
Estrazione Dati	√	√	√	√	√	o	√
Collegamento Dati	√	√	√	√	√	√	√
Scala Annotazione	√	√	√	√	√	√	√
Controllo Ortografia	√	√	√	√	√	√	√

Strumenti di Modifica							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Sposta, Copia, Ruota, Scala, Allunga, Specchia, Allinea	√	√	√	√	√	√	√
Taglia, Estendi, Allunga, Spezza, Unisci	√	√	√	√	√	√	√

Strumenti di Modifica							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Raccordo, Cima	√	√	√	√	√	√	√
CopiaGuidata	√	√	√	√	√	o	o
SpostaGuidata	√	√	√	√	√	o	o
Modifica mediante Grip	√	√	√	√	√	√	√
Pannello Proprietà	√	√	√	√	√	√	√
Manipolatore (Gizmo 3D)	√	√	√	√	√	o	√
Ordinedis	√	√	√	√	√	√	√
Annulla per Entità (Cronologia Proprietà)	√	√	√	√	√	o	o
Ottimizza	√	√	√	√	√	o	o
Semplifica	√	√	√	√	√	o	o

Strumenti Riferimento							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Blocchi, Editor Blocchi	√	√	√	√	√	√	√
Blocchi Dinamici	1	1	1	1	1	√	√
Blocco Fotocamera	√	√	√	√	√	o	o
Rileva Blocchi	√	√	√	√	√	o	o
Serie Associative	√	√	√	√	√	√	√
Attacca, Ritaglia, Collega, Modifica Dwg (Xrif)	√	√	√	√	√	√	√
Attacca, Ritaglia PDF	√	√	√	√	√	√	√
Attacca, Ritaglia, Regola, Trasparenza Immagine	√	√	√	√	√	√	√

Strumenti Riferimento							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Mappa Colori Nuvola di Punti	√	√	√	√	√	o	√

Strumenti di Visualizzazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Layers, Stati Layer	√	√	√	√	√	√	√
Trasparenza	√	√	√	√	√	√	√
Stili di Visualizzazione	√	√	√	√	√	o	√
LookFrom (ViewCube™)	√	√	√	√	√	o	√
Corrispondenza Prospettiva	√	√	√	√	√	o	o
Luci	o	√	√	√	√	o	√
Proprietà del Sole	o	√	√	√	√	o	√

Strumenti di Visualizzazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Materiali	o	√	√	√	√	o	√
Rendering	o	√	√	√	√	o	√
Editor Animazione	o	√	√	√	√	o	o
Soluzioni di navigazione	o	√	√	√	√	o	√
Comando ANIMAPERC	o	√	√	√	√	o	√

Strumenti di modellazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Modellazione ACIS	o	√	√	√	√	o	√
Modellazione di Solidi e Superfici	o	√	√	√	√	o	√
Primitive Solidi 3D	o	√	√	√	√	o	√

Strumenti di modellazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Primitive Mesh	o	√	√	√	√	o	√
Polisolido	o	√	√	√	√	o	√
Unione, Sottrai, Interseca, Interferisci	o	√	√	√	√	o	√
Estrusione, Rivoluzione, Loft, Sweep, Ispessimento	o	√	√	√	√	o	√
Torsione	o	√	√	√	√	o	o
Raccordo, Cima	o	√	√	√	√	o	√
Trancia	o	√	√	√	√	o	√
Multi-taglia	o	√	√	√	√	o	o
Connetti a T, Connetti a L, Connetti con il più Vicino	o	√	√	√	√	o	o

Strumenti di modellazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Estrai Spigoli	o	√	√	√	√	o	√
Modellazione Diretta	o	√	√	√	√	o	√
Modellazione Deformabile (a forma libera)	o	√	√	√	√	o	√
Audit, Semplificazione	√	√	√	√	√	o	√
Cuci	o	√	√	√	√	o	√
Parametri e Vincoli 3D	o	√	√	√	√	o	√
Autoparametrizzazione 3D	o	√	√	√	√	o	o
Blocco 3D e BlochificaParametrico	o	√	√	√	√	o	o

Strumenti di modellazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Tabelle di Progettazione Parametrica 3D	o	√	√	√	√	o	o
Piani di Sezione	o	√	√	√	√	o	√
Viste di Disegno	o	√	√	√	√	o	√
Filetto 3D	o	√	√	√	√	o	o
Allineamento Nuvola di Punti	o	√	√	√	√	o	o
Bubble Viewer Nuvola di Punti	o	√	√	√	√	o	o

Strumenti Civili							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Modellazione Superfici TIN	o	√	√	√	√	o	o

Strumenti Civili							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Superficie Volume TIN	o	√	√	√	√	o	o
Terrazzamento	o	√	√	√	√	o	o
Allineamenti	o	√	√	√	√	o	o
Corridoi	o	√	√	√	√	o	o
Civil 3D Object Enabler	√	√	√	√	√	o	√
Architectural Desktop Object Enabler	√	√	√	√	√	o	√
CADWorx Object Enabler	√	√	√	√	√	o	√

Strumenti Architettonici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Caratteristiche BIM	o	o	√	o	√	o	√

Strumenti Architettonici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Strumenti di Modellazione Strutturale	o	o	√	o	√	o	√
Database di Progetto	o	o	√	o	√	o	√
Database Libreria	o	o	√	o	√	o	√
Sito, Edifici e Piani	o	o	√	o	√	o	√
Finestre e Porte	o	o	√	o	√	o	√
Creazione di Componenti BIM Completamente Parametrici	o	o	√	o	√	o	o
Spazi 3D	o	o	√	o	√	o	√
Quickdraw	o	o	√	o	√	o	o
Quickbuilding	o	o	√	o	√	o	o

Strumenti Architettonici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Rilevamento Piani su Nuvole di Punti	o	o	√	o	√	o	o
Proiezione Nuvola di Punti	o	o	√	o	√	o	o
Rilevamento Geometria Nuvola di Punti	o	o	√	o	√	o	o
AdattaPlanare Nuvola di Punti (rilevamento della geometria)	o	o	√	o	√	o	o
Propaga	o	o	√	o	√	o	o
Proprietà BIM Personalizzate	o	o	√	o	√	o	o
Personalizzazioni Disegno	o	o	√	o	√	o	o
Modello Analitico	o	o	√	o	√	o	o

Strumenti Architettonici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Script Python (beta)	o	o	√	o	√	o	o

Strumenti Meccanici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Navigatore Meccanico	o	√	√	√	√	o	o
Libreria di Parti	√	√	√	√	√	o	o
Tubazioni	o	√	√	√	√	o	o
Modellazione Assiemi	o	o	o	√	√	o	o
Viste Esplose	o	o	o	√	√	o	o
Analisi Cinematica	o	o	o	√	√	o	o
Distinta dei Materiali	o	o	o	√	√	o	o

Strumenti Meccanici							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Progettazione Lamiera	o	o	o	√	√	o	o
Conversione da Solidi 3D in Lamiera	o	o	o	√	√	o	o
Scarico d'angolo per Spigoli di Flange Adiacenti	o	o	o	√	√	o	o
Piega con Loft	o	o	o	√	√	o	o
Colorazione	o	o	o	√	√	o	o
Diagnostica per Caratteristiche di Lamiera non Valide	o	o	o	√	√	o	o
Processi Batch	o	o	o	√	√	o	o

Importazione, esportazione, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Stampa/ Plotta	√	√	√	√	√	√	√
Pubblica /Batch Stampa	√	√	√	√	√	√	√
Files Tabelle Stili di Stampa CTB & STB	√	√	√	√	√	√	√
File Configurazione Stampante PC3	√	√	√	√	√	√	√
Impostazioni di Pagina con Nome	√	√	√	√	√	√	√
Importazione, Esportazione PDF	√	√	√	√	√	√	√
Importazione, Esportazione DAE	√	√	√	√	√	o	o
Importazione, Esportazione IFC	o	o	√	o	√	o	√

Importazione, esportazione, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Importazione, Esportazione 3DM	o	√	√	√	√	o	o
Importazione, Esportazione LandXML	o	√	√	√	√	o	o
Importazione RFA	o	o	√	o	√	o	o
Importazione SKP	o	√	√	√	√	o	o
Importazione DGN	√	√	√	√	√	√	√
Importazione RVT	o	o	√	o	√	o	o
Esportazione DWF	√	√	√	√	√	√	√
Esportazione SVG	√	√	√	√	√	o	o
Esportazione FBX	√	√	√	√	√	o	o
Esportazione STL	√	√	√	√	√	o	√

Importazione, esportazione, output							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Esportazione Nuvola di Punti	o	√	√	√	√	o	o
Ulteriori formati disponibili con BricsCAD Communicator	o	√	√	√	√	o	o

Programmazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
ActiveX	√	√	√	√	√	o	√
Editor dati entità	√	√	√	√	√	o	o
Registrazione Script	√	√	√	√	√	o	√
Pieno Supporto LISP (vl, vlr, vla e vlax)	√	√	√	√	√	o	√

Programmazione							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Soluzioni per il Sistema di Sviluppo (SDS/ADS)	√	√	√	√	√	o	√
Motore DCL	√	√	√	√	√	o	√
COM API	o	√	√	√	√	o	√
Visual Basic for Applications (VBA)	o	√	√	√	√	o	o
Supporto progetti VBA e DVB	o	√	√	√	√	o	o
.NET	o	√	√	√	√	o	√
BRX (ARX)	o	√	√	√	√	o	√
TX (Teigha eXtensions)	√	√	√	√	√	o	o
Supporto file menu CUI, con espressioni Diesel	√	√	√	√	√	√	√

Licenza							
	BricsCAD®					AutoCAD®	
	Lite	Pro	BIM	Mechanical	Ultimate	LT	Completo
Licenze Multipiattaforma	√	√	√	√	√	o	o
Perenne	√	√	√	√	√	o	o
Perenne + Manutenzione	√	√	√	√	√	o	o
Sottoscrizione	√	√	√	√	√	√	√
Multiutente Network	√	√	√	√	√	o	o
Volume (Perenne)	√	√	√	√	√	o	o
Accademica	√	√	√	√	√	√	√

(1) È possibile manipolare gli inserimenti di blocchi dinamici ma non è possibile creare definizioni di blocchi dinamici.

(2) È possibile applicare automaticamente vincoli geometrici, ma non è possibile applicare automaticamente una combinazione di vincoli geometrici e dimensionali.

2.2.3 Trial: attivazione online

Prima di iniziare: L'attivazione della licenza di prova di **BricsCAD®** richiede una connessione internet.

1 Lancia **BricsCAD®**.

2 Fare clic su **Avvia Trial** nella finestra di dialogo **Avvio di BricsCAD**.

Result: La licenza trial viene attivata su internet.

2.2.4 Trial: attivazione offline (manuale)

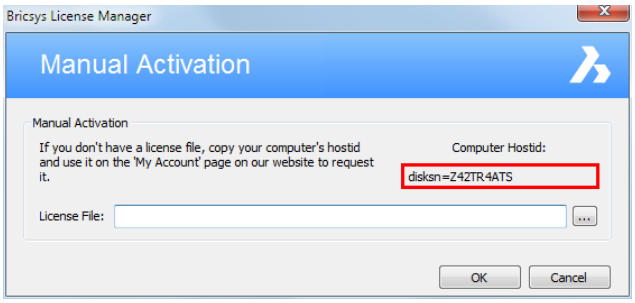
Se il computer non è connesso a internet, è possibile attivare la licenza manualmente tramite un altro computer connesso a internet.

1 Lancia **BricsCAD®**.

2 Cliccare su **Attiva Licenza** nella finestra di dialogo Avvio di BricsCAD.

3 Fare clic su **Attiva Manualmente...** nella finestra di dialogo **Gestore licenze Bricsys**.

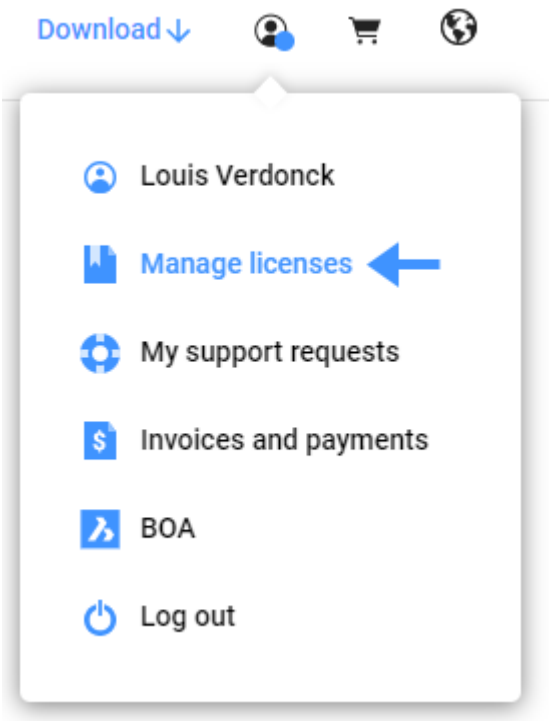
Result: Si aprirà la finestra di dialogo **Attivazione Manuale**.



4 Copiare l'intero contenuto del campo **Hostid Computer**.

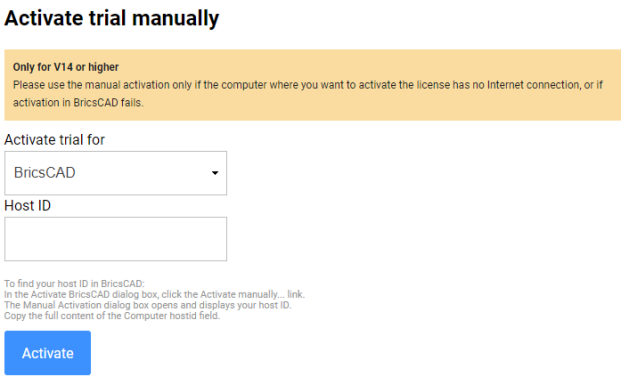
5 Accedere al sito web di Bricsys con il computer connesso a internet.

6 Fare clic sull'icona **Il mio Account** ( ) e selezionare **Gestore Licenze** dal menu.



7 Scegliere **Attiva BricsCAD manualmente**.

8 Nel campo **Host ID**, incollare l'Host ID del computer sul quale si desidera installare **BricsCAD®** e cliccare sul pulsante **Attiva**.



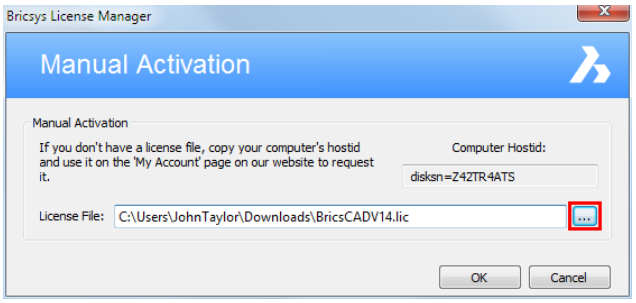
Result: Viene generato un file di licenza e reso disponibile per il download.

9 Cliccare sul pulsante **Scarica file di licenza**.

Result: Il file di licenza viene scaricato nella cartella download predefinita.

10 Nella finestra di dialogo **Attivazione Manuale**, cliccare sul pulsante **Sfoglia** (...).

11 Aprire la cartella in cui è stato copiato il file di licenza e selezionare il file.



12 Cliccare sul pulsante **OK**.

Result: **BricsCAD®** è stato lanciato.

2.2.5 Licenza singola o volume: attivazione online

Prima di iniziare: L'attivazione della licenza di prova di **BricsCAD®** richiede una connessione internet.

1 Lancia **BricsCAD®**.

Nota: Una volta scaduto il periodo di prova, fare clic su **Attiva Licenza** nella finestra di dialogo **Avvio di BricsCAD** e procedere con il passaggio 5.

2 Avviare il comando LICENSENSMANAGER.

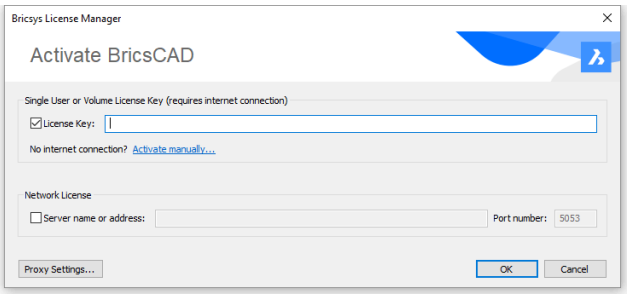
Result: Viene visualizzata la finestra di dialogo Gestione Licenze Bricsys.

3 Fare clic su **Gestisci licenza**.

Result: Viene visualizzata la finestra di dialogo **Informazioni sulla licenza**.

4 Fare clic sul pulsante **Modifica**.

Result: Viene visualizzata la finestra di dialogo **Gestore Licenze BricsCADr** > **Attiva BricsCAD**:



5 Incollare la tua chiave di licenza nel campo **Chiave di Licenza**.

6 Cliccare sul pulsante **OK**.

Result:

- La chiave di licenza è registrata su internet.
- Viene visualizzata una finestra di dialogo quando il codice di licenza viene attivato.

7 Leggere il testo nella finestra di dialogo e fare clic sul pulsante **Ho capito**.

2.2.6 Licenza singola o volume: attivazione offline (manuale)

Se il computer non è connesso a internet, è possibile attivare la licenza manualmente tramite un altro computer connesso a internet.

Nota: Non è possibile disattivare le licenze attivate manualmente.

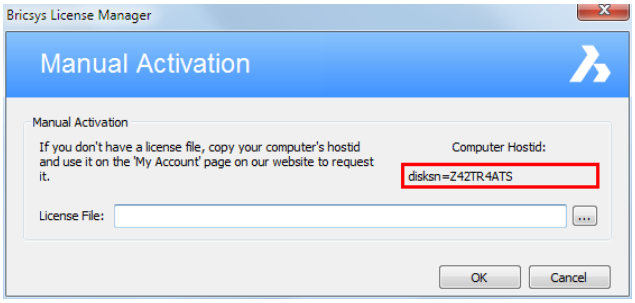
1 Lancia **BricsCAD®**.

2 Eseguire una delle seguenti operazioni:

- Se stati avviando **BricsCAD®** per la prima volta, vai al passaggio successivo.
- Se si avvia **BricsCAD®** in modalità di prova, fare clic su **Inserisci licenza...** nella finestra di dialogo **Trial Gratuita**.
- Se si sostituisce o si aggiorna una licenza esistente, avviare il comando LICENSEMANAGER, quindi fare clic su **Gestisci licenza** e su **Modifica...**.

3 Fare clic su **Attiva Manualmente...** nella finestra di dialogo **Attiva BricsCAD**.

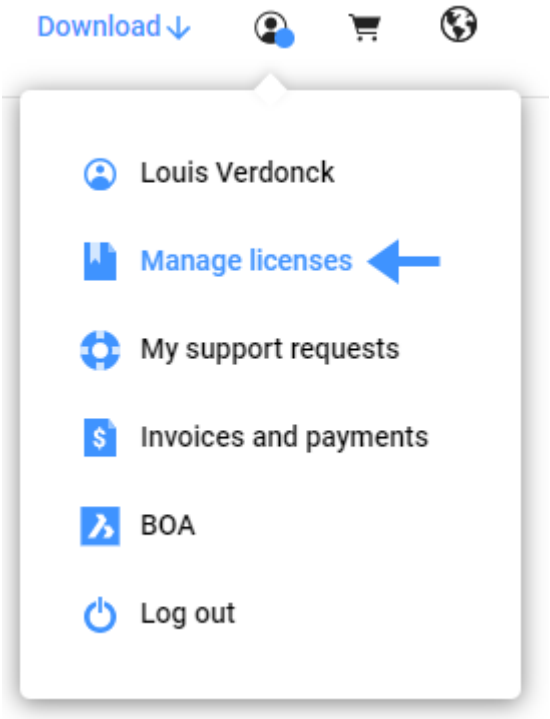
Result: Si aprirà la finestra di dialogo **Attivazione Manuale**.



4 Copiare l'intero contenuto del campo **Hostid Computer**.

5 Accedere al sito web di Bricsys con il computer connesso a internet.
Assicurati di accedere con l'account Bricsys che contiene le tue licenze.

6 Fare clic sull'icona **Il mio Account** ( ) e selezionare **Gestore Licenze** dal menu.



7 Passare il mouse sulla chiave di licenza che si desidera attivare e fare clic sull'icona della matita ( ).

Result: Vengono visualizzati i **Dettagli della Licenza**.

8 Cliccare su **Attivazione Manuale**.

9 Nel campo **ID host** incollare l'ID host del computer sul quale si desidera installare **BricsCAD®**.

10 Fare clic sul pulsante **Attiva**.

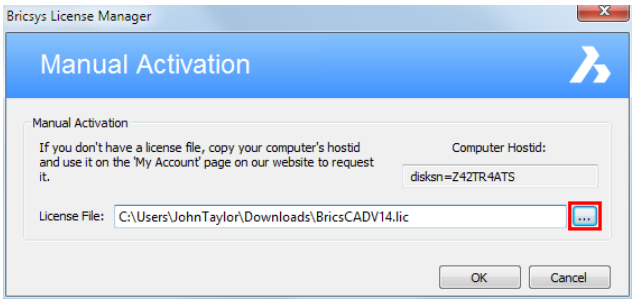
Result: Viene generato un file di licenza e reso disponibile per il download.

11 Fare clic sul pulsante **Scarica file di licenza**.

Result: Il file di licenza **BricsCADVxx.lic** viene scaricato nella cartella download predefinita.

12 Copiare il file di licenza sul computer su cui si desidera installare **BricsCAD®**.

13 Nella finestra di dialogo **Attivazione Manuale**, fare clic sul pulsante Sfoglia ( ). Aprire la cartella in cui è stato copiato il file di licenza **BricsCADVxx.lic** e selezionare il file.



14 Cliccare sul pulsante **OK**.

Result: **BricsCAD®** è stato lanciato.

2.2.7 Utilizzo di una licenza di rete su un computer client

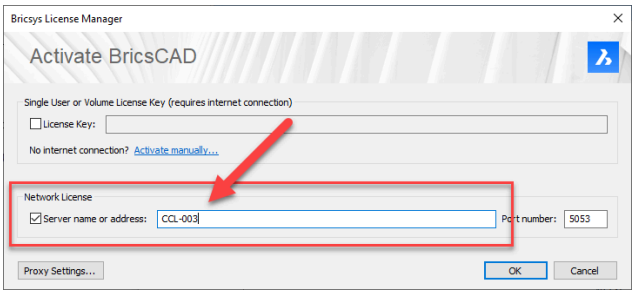
Una volta attivata la licenza di rete sul server delle licenze, può essere utilizzata sui computer client.

1 Eseguire una delle seguenti operazioni:

- Se si avvia **BricsCAD®** per la prima volta, fare clic sul pulsante **Attiva licenza** nella finestra di dialogo Avvio di BricsCAD.
- Se si sostituisce o si aggiorna una licenza esistente, avviare il comando LICENSEMANAGER, quindi fare clic su **Gestire Licenze** e su **Modifica...**.

2 Selezionare l'opzione **Licenza di Rete** nella finestra di dialogo **Attiva BricsCAD**.

3 Digitare il nome host o l'indirizzo IP del server delle licenze nel campo **Indirizzo o nome del server**.



4 (Opzionale) Digitare il numero di porta nel campo **Numero porta**.

5 Se il computer è connesso al server licenze tramite un server proxy, fare clic sul pulsante **Impostazioni proxy** per configurare le impostazioni proxy per una licenza di rete.

6 Cliccare sul pulsante **OK**.

2.3 Configurazione di un server licenze di rete

2.3.1 Server delle licenze di rete

Per rendere disponibile una licenza di rete in **BricsCAD®** su un computer client, l'amministratore di rete deve prima installare un server di licenza di rete da **Reprise Software**.

Bricsys Network License Manager è un programma separato che installa il server. Dopo aver avviato il server delle licenze di rete, è possibile configurarlo tramite un'interfaccia web.

Proprio come **BricsCAD**® ha bisogno di un file di licenza per l'esecuzione, il server licenze di rete ha bisogno di un file di licenza di rete che è possibile attivare online o manualmente.

È possibile attivare una licenza di rete una sola volta ed è bloccata sul server.

Dopo l'attivazione della licenza di rete e l'avvio del server, è possibile utilizzare la licenza di rete in **BricsCAD**® su un computer client.

Nota: Reprise Software non offre un server licenze di rete per computer Mac. Tuttavia, è possibile installare un server licenze di rete Windows o Linux per servire le licenze sui computer client con Mac.

2.3.2 Guida rapida all'utilizzo di una licenza di rete

Innanzitutto, assicurarsi che l'amministratore di rete/IT sia coinvolto nel processo.

2.3.2.1 1. Azioni da eseguire sul server

1 Scaricare il Bricsys Network License Manager e installarlo con le impostazioni predefinite:

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

Informazioni relative ai criteri di sicurezza del server:

dopo l'installazione, il processo RLM.exe deve poter utilizzare 3 porte: **5053** per la licenza, **5054** per l'interfaccia utente web, una **porta allocata dinamicamente** da Windows per il server ISV che può essere impostata fissa dopo l'attivazione come descritto nell'articolo Configurazioni Avanzate.

2 Quando appare la finestra di dialogo dell'attivazione, attivarlo utilizzando la chiave di licenza (un codice) presente nel tuo account Bricsys.

Informazioni relative ai criteri di sicurezza del server:

l'applicazione che gestisce l'attivazione è *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager\actNetworkLicense.exe* e deve essere in grado di raggiungere il link **license.bricsys.com/actpro** sulla porta **80**

3 La pagina di amministrazione del RLM License Manager è raggiungibile su **localhost:5054**

2.3.2.2 2. Azioni da eseguire sulla macchina client

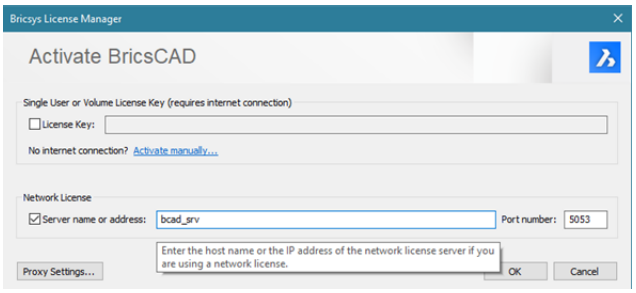
Il computer client deve:

- Trovarsi nella stessa **LAN** con la macchina RLM, fisicamente o tramite connessione VPN.
- Essrere in grado di comunicare con il computer RLM sulla porta **5053** tramite il protocollo **TCP**.

1 Installare **BricsCAD**® con le impostazioni predefinite.

2 Avviare **BricsCAD**® e attivare la licenza di rete: <nomeserver>/<numero ip> e <porta>

(potrebbe non essere necessario in quanto durante l'avvio, **BricsCAD**® verificherà la rete alla ricerca di un server delle licenze)



2.3.3 Configurazione di un server delle licenze di rete Windows

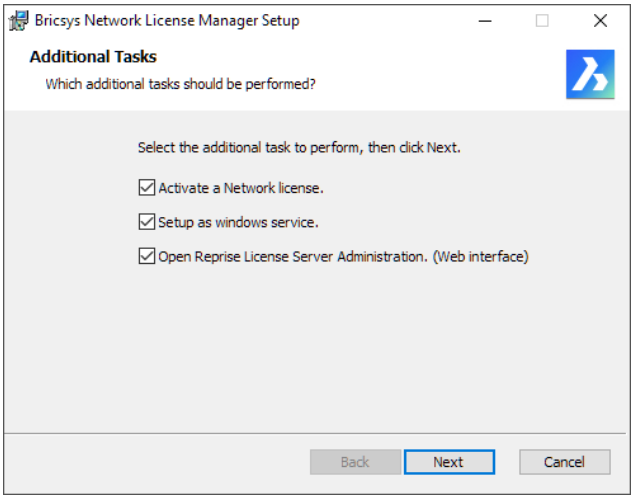
2.3.3.1 Download del Bricsys Network License Manager

Scaricare il Bricsys Network License Manager (Windows) dal seguente collegamento:

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

2.3.3.2 Nuova installazione del Bricsys Network License Manager

Nella finestra di dialogo **Additional Tasks**, accertarsi di selezionare tutte le caselle di controllo (vedere immagine del Bricsys Network License Manager).



- **Attivare una licenza di rete**

  Dopo l'installazione, viene aperto il **Bricsys Network License Manager** e viene chiesto di attivare una licenza di rete. È possibile aprire il **Bricsys Network License Manager** in un secondo momento facendo clic sul collegamento License activation nel menu Start.

- **Installazione come Servizio di Windows**

  Ogni volta che si avvia Windows, il server delle licenze verrà automaticamente avviato come servizio. È possibile farlo in seguito con il seguente comando nel prompt dei comandi con diritti di amministratore: rlm.exe-dlog rlm.log-install_service

- **Aprire il Reprise License Server Administration (interfaccia web)**

Dopo l'installazione, viene aperta l'interfaccia web del Reprise License Server Administration. Consente di gestire la licenza di rete. È possibile aprire il Reprise License Server Administration in un secondo momento facendo clic sul collegamento License Server Administration nel menu Start.

2.3.3.3 Aggiornamento del Bricsys Network License Manager

1 Scaricare e installare il Bricsys Network License Manager sul computer che verrà utilizzato come server delle licenze di rete.

2 Assicurarsi di deselezionare tutte le caselle di controllo accanto a **Additional Tasks**.

Nota: L'installazione riavvia il server delle licenze.

2.3.3.4 Installazione dalla riga di comando

È inoltre possibile avviare il Network License Manager da una finestra di comando.

Per impostazione predefinita, le opzioni **Activate a Network license**, **Setup as Windows service** e **Open Reprise License Server Administration** (interfaccia Web) sono disabilitate durante l'installazione tramite la riga di comando. Per attivare queste opzioni, utilizzare i rispettivi parametri ACTIVATELICENSE, INSTALLSERVICE e OPENWEBINTERFACE.

Ad esempio, **msiexec /qb /i Bricsys-NetworkLicenseManager.msi ACTIVATELICENSE=1 INSTALLSERVICE=1** mostrerà solo la finestra di dialogo dell'attivazione ed installerà come servizio senza visualizzare l'interfaccia web.

Nota:

- Per ulteriori opzioni, digitare: **msiexec /?** nel prompt dei comandi.
- Sul sito Web di Microsoft, è possibile trovare ulteriori informazioni sul comando **msiexec**.

2.3.4 Configurazione di un server delle licenze Linux

Prima di iniziare: Scaricare e installare Bricsys Network License Manager sul computer che verrà utilizzato come server delle licenze di rete.

1 Scaricare Bricsys Network License Manager (Linux)

2 Estrarre il file **Bricsys-NetworkLicenseManager.tgz** in qualsiasi cartella (linea di comando: tar xvzf Bricsys-NetworkLicenseManager.tgz).

3 Avviare l'applicazione RLM in quella cartella (riga di comando in questa cartella: ./rlm).

4 Andare alla pagina web **http://localhost:5054**.

5 Fare clic sul pulsante **System Info** sulla sinistra della pagina Web. Copiare un IEthernet host ID e utilizzarlo per attivare la licenza nella pagina **Il mio Account** del sito Web Bricsys.

6 Quando si scarica un file di licenza dal sito Bricsys, memorizzarlo nella cartella dove avete estratto Bricsys-NetworkLicenseManager.tgz.

7 Cliccare il pulsante **Reread/Restart Serves**.

8 Fare clic sul pulsante **Stato** per vedere lo stato del server, nella pagina successiva fare clic sul pulsante **bricsys** nella tabella per visualizzare le licenze Bricsys.

9 Fare clic sul pulsante **RLM Manual...** per informazioni dettagliate su come configurare il server licenze. Tra le altre cose, la pagina spiega come avviare il RLM automaticamente all'avvio del server.

Result: Una volta attivata la licenza di rete e avviato il server, il server è pronto per servire le licenze. Ora è possibile utilizzare la licenza di rete di **BricsCAD®** su un computer client.

2.3.5 Licenza di rete: attivazione online

L'attivazione della licenza di prova di **BricsCAD®** richiede una connessione internet.

1 Fare clic sul collegamento **License activation** nel menu **Start** del sistema operativo.

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

Viene visualizzata la finestra di dialogo **Bricsys License Manager - Activate Network License**.

2 Immettere la nuova licenza di rete nel campo **License Key** e fare clic su **OK**.

3 Fare clic sul collegamento **License server administration** nel menu **Start** del sistema operativo per aprire **Reprise License Server Administration**.

4 Fare clic sull'opzione **Reread/Restart Servers** nel menu a sinistra.

2.3.6 Licenza di rete: attivazione offline (manuale)

Se il computer non è connesso a internet, è possibile attivare la licenza manualmente tramite un altro computer connesso a internet.

Nota: Non è possibile disattivare le licenze attivate manualmente.

1 Fare clic sul collegamento **License activation** nel menu **Start** del sistema operativo.

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

Viene visualizzata la finestra di dialogo **Bricsys License Manager - Activate Network License**.

2 Copiare l'intero contenuto del campo **Hostid Computer**.

3 Accedere al sito web di Bricsys con il computer connesso a internet.

4 Fare clic sull'icona **Il mio Account** ( ) e selezionare **Gestore Licenze** dal menu.

Assicurarsi di accedere con l'account Bricsys dell'azienda che contiene la licenza di rete. Se necessario, contattare il proprio account manager Bricsys.

5 Passare il mouse sulla chiave di licenza che si desidera attivare e fare clic sull'icona della matita ( ).

Vengono visualizzati i **Dettagli della Licenza**.

6 Cliccare su **Attivazione Manuale**.

7 Nel campo **ID host** incollare l'ID host del computer sul quale si desidera installare **BricsCAD®**.

8 Fare clic sul pulsante **Attiva**.

Viene generato un file di licenza e reso disponibile per il download.

9 Fare clic sul pulsante **Scarica file di licenza**.

Il file di licenza **BricsCAD.lic** viene scaricato nella cartella download predefinita.

10 Copiare il file di licenza nel server, nella cartella di installazione del **Bricsys Network License Manager** (per impostazione predefinita: *C:\Program Files (x86)\Bricsys\Bricsys Network License Manager*).).

11 Eseguire **reread/restart server** dall'interfaccia utente web del RLM: localhost:5054, affinché Reprise

License Manager utilizzi il file .lic.

2.3.7 Amministrazione del server delle licenze di rete

Quando è in esecuzione il server delle licenze di rete (denominato anche server licenze RLM), è possibile configurarlo nell'interfaccia Web di Reprise License Server Administration.

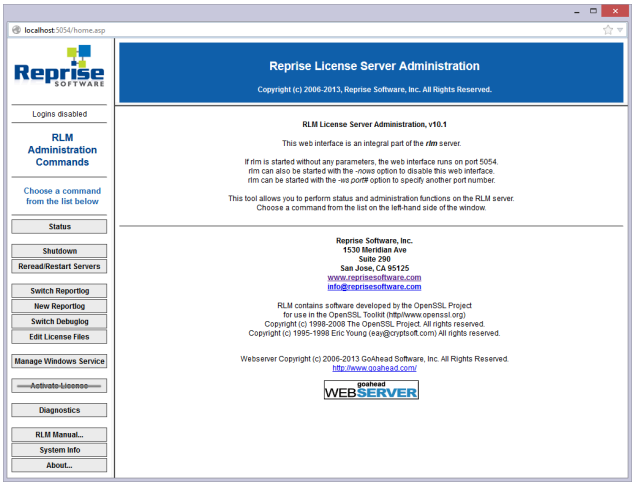
2.3.7.1 Aprire l'interfaccia web

1 Se il server delle licenze di rete non è in esecuzione, scegliere il collegamento **License Server** nel menu Start (in Windows Vista o versione successiva: cliccare il pulsante destro del mouse su > Esegui come amministratore).

Result: Si apre la finestra della console.

2 Andare su **http://localhost:5054**.

Result: Viene aperta l'interfaccia web Reprise License Server Administration.



Nota:

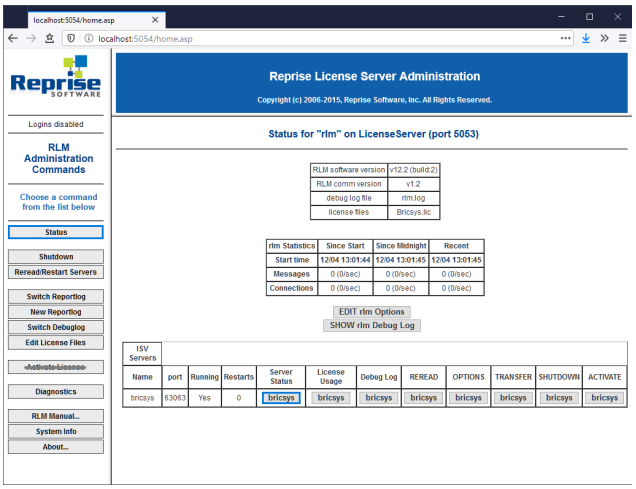
- Non utilizzare il pulsante **Activate License**!
- Per ottenere una panoramica di tutte le opzioni, cliccare sul pulsante R**LM Manual...** a sinistra o andare su Manuale RLM License Administration.

2.3.7.2 Controllo dello stato della licenza e del server

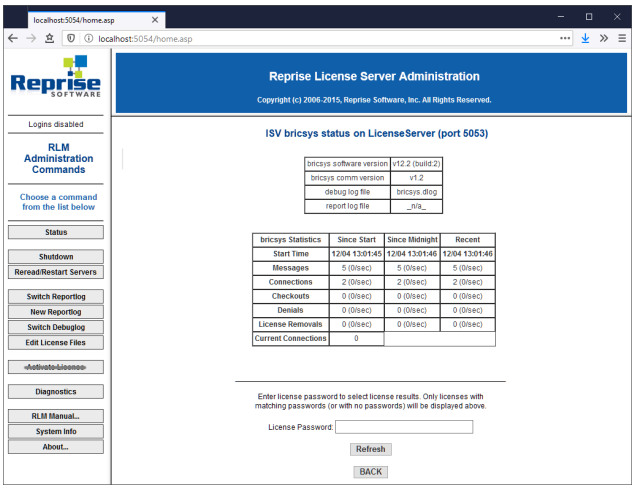
È possibile controllare lo stato di tutte le licenze di rete sul server.

1 Fare clic sul pulsante **Stato** a sinistra.

Result: Viene visualizzata la pagina **Status for "rlm" on LicenseServer (port 5053**.



2 Fare clic sul pulsante **bricsys** nella colonna **Server Status** della tabella ISV Servers per visualizzare lo stato della licenza di rete Bricsys.

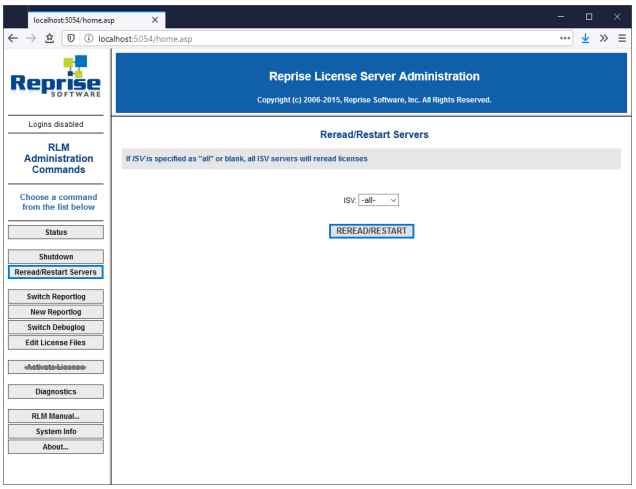


2.3.7.3 Rileggere la licenza / Riavvio del server

Dopo l'attivazione di una licenza di rete mentre il server era in esecuzione, si deve rileggere la licenza / riavviare il server.

1 Fare clic sul pulsante **Reread/Restart Servers** sulla sinistra.

Result: Viene visualizzata la pagina **Reread/Restart Servers**.



2 Cliccare il pulsante **Reread/Restart Serves**.

Result: Il server si riavvia e rilegge la licenza di rete.

2.3.8 Configurazioni avanzate

Una volta configurato il server licenze di rete, **BricsCAD®** potrebbe segnalare un errore -17 durante il tentativo di connessione al server.

I motivi possono essere dovuti alla presenza di software antivirus e/o firewall e/o di un server proxy.

2.3.8.1 Anti-virus

Se un software antivirus è in esecuzione sul server e blocca il programma del server delle licenze (**rlm.exe** su Windows, **rlm** su Linux/Mac), è necessario sbloccare il servizio RLM nell'antivirus.

Vedere la configurazione del tuo antivirus su come sbloccare i servizi.

2.3.8.2 Firewall

Se è installato un firewall tra il server e **BricsCAD®**, devono essere aperte nel firewall 2 numeri di porta: la porta numero 5053 ed il numero di porta assegnato al server delle licenze Bricsys.

Questo numero di porta viene assegnato dinamicamente all'avvio del server licenze. Per aprire questa porta nel firewall, è necessario prima impostarla nel file Bricsys.lic sul server:

1 Aprire l'interfaccia Web del License Server Administration su **http://localhost:5054**.

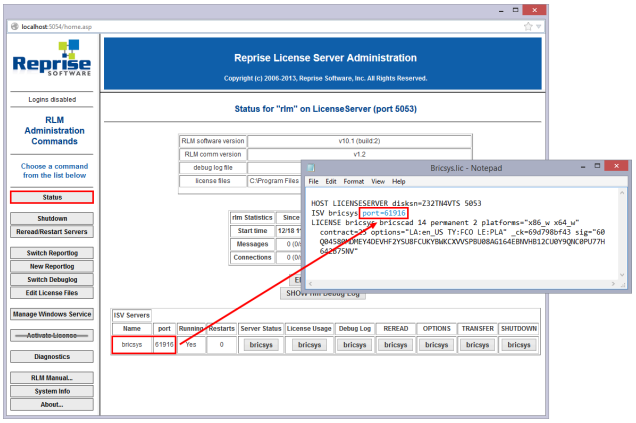
2 Fare clic sul pulsante **Status** a sinistra. Viene visualizzata la pagina **Status for "rlm" on [LicenseServer] (port 5053)**.

3 Cercare il numero di porta nella tabella Server ISV. Il numero di porta viene visualizzato nella riga bricsys nella seconda colonna.

4 Copia questo numero di porta.

5 Aprire il file Bricsys.lic in un editor di testo (ad esempio Blocco note, Gedit, ...). Il percorso predefinito del file è C:\Program Files (x86)\Bricsys\Bricsys Network License Manager.

È possibile correggere il numero di porta modificando la riga ISV bricsys con: ISV bricsys port=[numero porta], dove [numero porta] deve essere sostituito dal numero di porta, copiato dalla tabella ISV Servers.



6 Nel firewall, aprire la porta numeri 5053 ed il numero di porta che è stato aggiunto nel file Bricsys.lic.

7 Arrestare e riavviare il programma del server delle licenze.

Nota: Se questo non risolve il problema, aggiungere il programma del server delle licenze (rlm.exe su Windows, rlm su Linux/Mac) come eccezione all'elenco eccezioni del firewall e riavviare il server licenze.

2.3.8.3 Server proxy

Se esiste un server proxy tra **BricsCAD®** e il server licenze, è necessario configurare le impostazioni proxy in **BricsCAD®**.

Vedere le impostazioni del Proxy su come configurare le impostazioni del proxy in modo permanente per l'utilizzo della licenza di rete.

2.4 Gestione delle licenze

2.4.1 Modifica di una licenza BricsCAD®

Utilizzare questa procedura per aggiornare o sostituire una licenza.

1 Avviare il comando LICENSENSMANAGER.

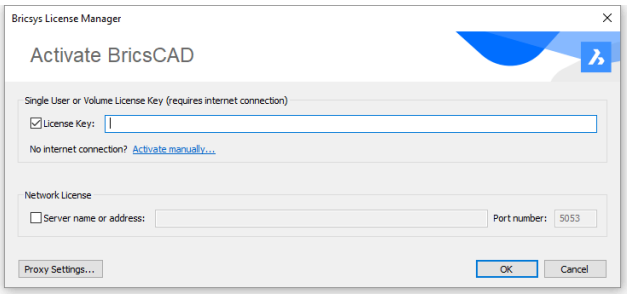
Result: Viene visualizzata la finestra di dialogo Gestione Licenze Bricsys.

2 Fare clic su **Gestisci licenza**.

Result: Viene visualizzata la finestra di dialogo **Informazioni sulla Licenza**.

3 Cliccare sul pulsante **Modifica**.

Result: Viene visualizzata la finestra di dialogo **Gestore Licenze Bricsys** - **Attiva BricsCAD**:



4 Incollare la chiave di licenza nel campo **Chiave di Licenza**.

5 Cliccare sul pulsante **OK**.

Result:

- La chiave di licenza è registrata su internet.
- Viene visualizzata una finestra di dialogo quando il codice di licenza viene attivato.

6 Leggere il testo nella finestra di dialogo e fare clic sul pulsante **Ho capito**.

2.4.2 Disattivazione o revoca di una licenza BricsCAD®

Utilizzare questa procedura per:

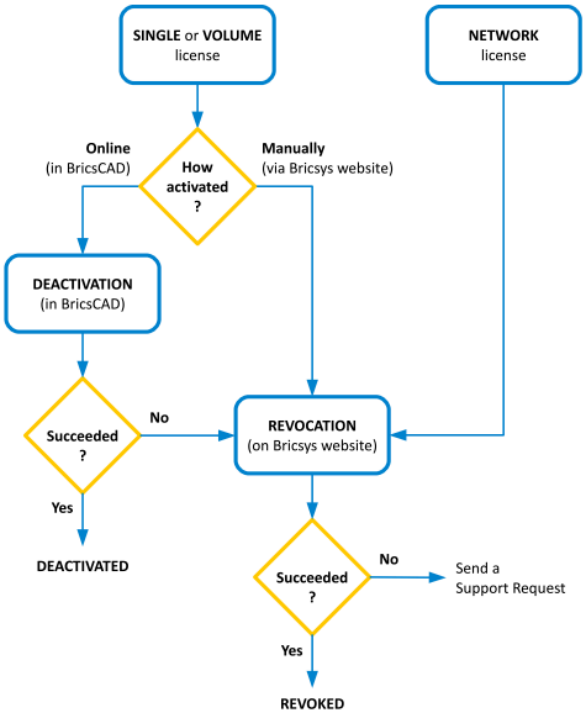
- Migrare una licenza su un altro computer.
- Rimuovere definitivamente **BricsCAD®** da un computer.
- Riattivare **BricsCAD®** dopo aver installato un nuovo disco rigido, modificare le partizioni o aggiornare il sistema operativo.

Per disattivare una licenza **BricsCAD®**, è necessaria una connessione a internet. Non è possibile disattivare manualmente una licenza tramite un altro computer connesso a internet.

Non è possibile disattivare:

- Licenze singole e volume attivate manualmente.
- Licenze di rete.
- Licenze di prova.

Il grafico spiega come disattivare una licenza **BricsCAD®**. Se non è possibile disattivare una licenza **BricsCAD®**, è possibile revocare la licenza.



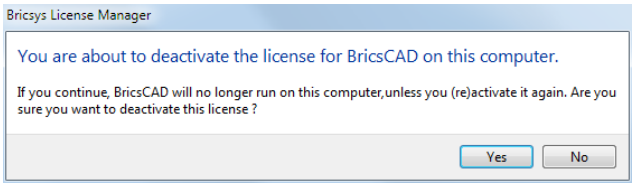
2.4.2.1 Disattivazione di una licenza

1 Avviare il comando LICENSENSMANAGER. Fare clic sull'opzione **Gestore Licenze** nella finestra di dialogo **Gestore Licenze Bricsys**.

Result: Viene visualizzata la finestra di dialogo **Informazioni sulla Licenza**.

2 Se il computer si connette al server delle licenze tramite un server proxy, fare clic su **Impostazioni proxy** per configurare le impostazioni proxy.

3 Fare clic su **Disattiva...**



4 Fare clic sul pulsante **Sì** nella finestra di dialogo di conferma per disattivare la licenza.

Result: La licenza è disattivata e può essere attivata su un altro computer.

2.4.2.2 Revocare una licenza

Revocare una licenza solo se la disattivazione non è possibile.

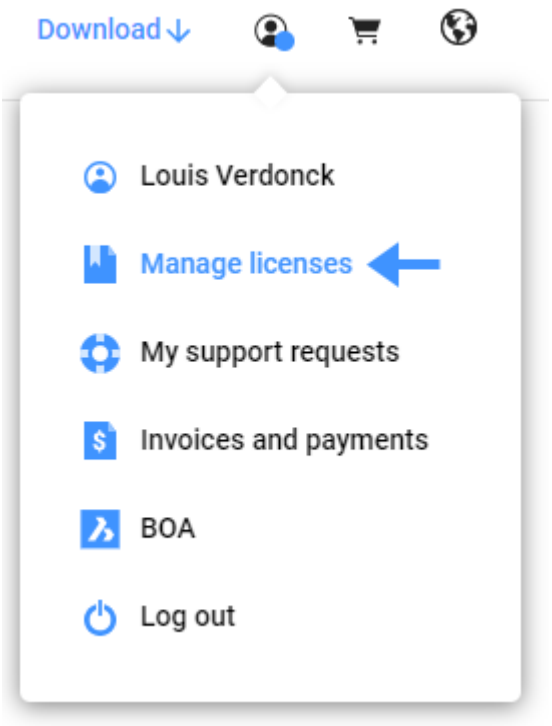
Utilizzare questa procedura per:

- Disattivare manualmente una licenza network o una licenza singola o volume.
- Riattivare **BricsCAD®** dopo un guasto al disco rigido.

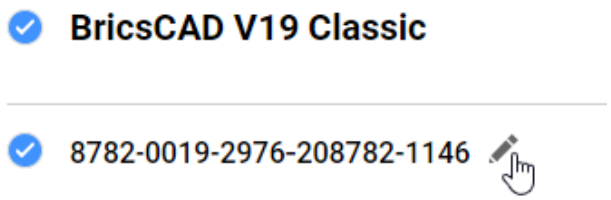
- Riattivare **BricsCAD®** dopo aver installato un nuovo disco rigido, modificare le partizioni o aggiornare il sistema operativo.
- Migrare una licenza su un altro computer.
- Rimuovere definitivamente **BricsCAD®** da un computer.

1 Accedi al sito web di Bricsys.
Assicurati di accedere con l'account Bricsys contenente le tue licenze.

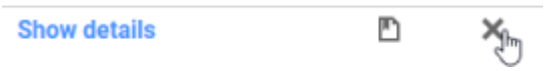
2 Fare clic sull'icona **Il mio Account** ( ) e selezionare **Gestore Licenze** dal menu.



3 Passa il mouse sopra il codice di licenza della licenza da revocare e fare clic sull'icona a forma di matita ( ).



4 Scorrere verso il basso e fare clic sull'icona della **X** per revocare la licenza.



2.4.3 Migrazione di una licenza singola o volume su un altro computer

Tutte le licenze valide sono disponibili sul sito web di Bricsys (accesso richiesto):

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

1 Disattivare la licenza sul vecchio computer.

2 Avvia **BricsCAD®** sul nuovo computer.

3 Attiva la licenza sul nuovo computer.

2.4.4 Migrazione di una licenza di rete su un altro server

Eseguire le seguenti operazioni:

1 Revocare la licenza.

2 Attivare la licenza sul nuovo server.

3 Chiudere il gestore licenze sul vecchio server ed eliminare il file di licenza sul vecchio server.

Nota: Il numero di revoche è limitato.

2.4.5 Pooling della licenza

2.4.5.1 Modifica delle opzioni

E' possibile modificare le opzioni in 2 modi:

- Nel file bricsys.opt, che si trova nella cartella di installazione del Network License Manager.
- Sull'interfaccia web del Network License Manager.
    - Andare su *http://localhost:5054* per accedere al server delle licenze.
    - Cliccare su **Status** nel menu sul lato sinistro.
    - Fare clic su **bricsys** nella colonna OPTIONS per la riga bricsys ISV.

Una volta modificato, cliccare su Reread/Restart per riavviare il server delle licenze.

2.4.5.2 Opzioni utilizzate di frequente

I gruppi possono essere utilizzati per definire un elenco di nomi utente, nomi host o indirizzi IP.

Il nome può essere utilizzato in seguito per applicare le regole per quel gruppo.

		Sintassi	Esempi
GRUPPO	Definisce un gruppo di nomi utente.	GROUP nome lista-di-nomiutente	Ingegneri del GRUPPO userA userB userC
HOST_GROUP	Definisce un gruppo di nomi host.	HOST_GROUP nome lista-di-nomihost	HOST_GROUP Pianoprimo macchinaA macchinaB macchinaC

		Sintassi	Esempi
INTERNET_GROUP	Definisce un gruppo di indirizzi IP.	INTERNET_GROUP nome lista-di-indirizzi-ip	INTERNET_GROUP Mianetwork 192.168.1.* 192.168.2.* Il carattere jolly (*) può essere utilizzato negli indirizzi IP.
RESERVE	Riserva un numero di licenze a un specifico utente o gruppo di utenti	RESERVE num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	RESERVE 20 bricscad GROUP ingegneri
MAS	Limita il numero di licenze disponibili ad un specifico utente o gruppo di utenti	MAX num product user|host|group|host_group|internet|internet_group|project who [id=nnn]	MAX 5 bricscad HOST_GROUP Pianoprimo
INCLUDE	Consente solo licenze per un prodotto a un specifico utente o gruppo di utenti Chiunque non sia specificato dalla riga INCLUDE non è autorizzato a utilizzare il prodotto.	INCLUDE product userhost|group|host_group|internet|internet_group|project who [id=nnn]	INCLUDE bricscad GROUP utentibricscad
EXCLUDE	Esclude licenze per un prodotto a uno specifico utente o gruppo di utenti	EXCLUDE product user|host|group|host_group|internet|internet_group|project who [id=nnn]	EXCLUDE bricscad INTERNET_GROUP 1.1.*.*

Ulteriori opzioni sono disponibili nel manuale dell'utente finale di RLM sotto il paragrafo **The ISV Options File**

2.4.5.3 Identificazione di un file di licenza

Dalla versione V15, una nuova funzionalità nel gestore licenze Reprise consente di aggiungere un campo '_id' nel tuo file di licenza.

Con questo "_id" è possibile identificare ciascun file di licenza come un pool di licenze separato.

Quando specificando un parametro [id=nnn] per un OPTION aggiunto, tale OPTION avrà effetto solo sul file di licenza corrispondente.

Esempio di utilizzo: Hai 2 licenze, una con 10 postazioni per **BricsCAD® Lite**, e una con 10 postazioni per **BricsCAD® Pro**, e vuoi specificare quali utenti possono utilizzare quale livello.

FILE DI LICENZA

Licenza Lite

HOST host_name ethernet_mac 5053

ISV bricsys

LICENSE bricsys bricscad 15 permanent 10 share=uh

ccontract=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=""

Licenza Pro

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=""

Ora la licenza Lite è definita da id=1, e la licenza Pro da id=2.

LE OPZIONI

Se, ad esempio, desideri consentire solo agli ingegneri della tua azienda di utilizzare la versione BIM, puoi farlo aggiungendo le seguenti opzioni.

Ingegneri del GRUPPO userA userB userC

INCLUDE bricscad GROUP engineers id=2

Sintomi

Voglio regolare il modo in cui le mie licenze sono raggruppate.

Maggiori informazioni http://www.reprisesoftware.com/RLM_Enduser.html

2.4.5.4 Pooling delle licenze by RLM_PROJECT

Quanto segue si applica ai casi in cui più livelli di licenze di rete sono attivati sullo stesso server RLM. Di conseguenza, gli utenti sono in grado di decidere quale livello di licenza acquisire al momento dell'attivazione **BricsCAD®**.

L'utente può ottenere un specifico livello di licenza **BricsCAD®**, basato sulla variabile di ambiente RLM_PROJECT che deve corrispondere alle opzioni del server ISV. Le postazioni in licenza saranno riservate sul server ISV per specifici utenti PROJECT da parte di un IDentifer.

Nota: Notare che le seguenti azioni devono essere eseguite da un responsabile IT.

1. Azioni server RLM

1 Aggiungere _id= identificatore per i file di licenza, appena prima della stringa _ck=xxxxxxx:

……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 Aggiungere l'opzione RESERVE per PROGETTI e ID licenze al server ISV, per tutte le postazioni delle licenze:

RESERVE 5 bricscad PROJECT bc_pro id=1

RESERVE 5 bricscad PROJECT bc_lite id=2

2. Azioni dell'utente BricsCAD®

1 Creare uno script che IMPOSTA una variabile di ambiente temporanea RLM_PROJECT sul computer utente, in modo da consentire a BricsCAD di negoziare con il server RLM il pool di licenze PROJECT.

2 L'esecuzione di uno script specifico consentirà a BricsCAD di estrarre il livello di licenza desiderato

Può trattarsi di file *.CMD (prompt dei comandi):

- @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"

Poiché tutte le postazioni di licenza sono RISERVATE nelle opzioni RLM, **BricsCAD**® non sarà in grado di acquisire una licenza quando la variabile RLM_PROJECT non esiste o quando ha un valore che non corrisponde alle opzioni del server ISV.

Sollevare i posti disponibili al di sopra di 5 o abbassare il numero di RESERVE al di sotto di 5 permetterebbe che alcune licenze siano liberamente disponibili.

Il RLM_PROJECT funziona anche con le altre opzioni (EXCLUDE/INCLUDE/...) in modo che sia possibile configurare molte configurazioni.

2.4.6 Report utilizzo licenze di rete

Esistono diverse possibilità per controllare l'utilizzo della licenza di rete:

- Dall'interfaccia utente web di RLM

  vai a localhost:5054/home.asp > Status > License Usage > bricsys
- Dal file "bricsys.dlog" che contiene le informazioni IN/OUT. Questo file si trova nella cartella RLM *C:\Programmi (x86)\Bricsys\Bricsys Network License Manager*
- Opzione REPORTLOG. Questa opzione è descritta nelle pagine 77 e 103 nel manuale RLM:

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

  Deve essere aggiunto al file **bricsys.opt** come questo:

  REPORTLOG + D:_il_tuo_percorso\nome_report.log

  Successivamente, eseguire un Reread/Restart Servers.

  Il segno + indica che i nuovi dati vengono aggiunti al file, altrimenti il file verrà sovrascritto.
- Pacchetto Amministrazione Licenze RLM

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

  https://jtbworld.com/jtb-flexreport

2.5 Utilizzo di una licenza di rete

2.5.1 Roaming

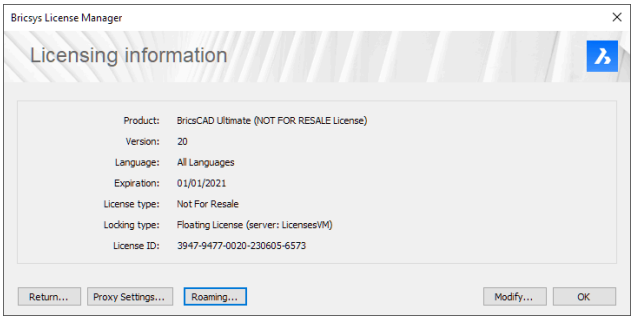
Con una licenza di rete, è possibile impostare una licenza flottante per il roaming a un sistema che verrà successivamente disconnesso dalla rete. La licenza risultante può essere utilizzata per il numero di giorni specificato quando la licenza è stata impostata per il roaming. Dopo questo periodo, la licenza viene riassegnata automaticamente. Se necessario, è possibile restituire la licenza roamed al pool di licenze in anticipo.

I file di licenza sul server delle licenze di rete contengono un parametro *max_roam_count*. Questo valore, per impostazione predefinita, è impostato sul numero totale di postazioni di una licenza di rete.

2.5.1.1 Roaming di una licenza di rete in un computer client

Una licenza di rete può essere concessa in roaming per utilizzare **BricsCAD**® senza connessione al server licenze (lavoro esterno all'ufficio, a casa, ...).

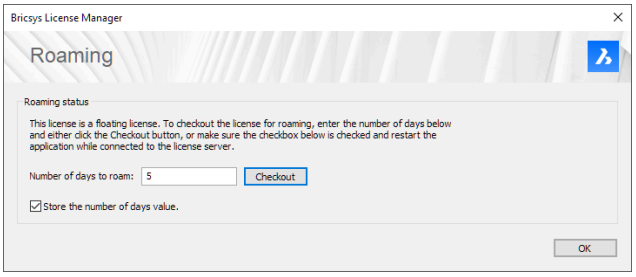
Quando una licenza di rete viene utilizzata in un computer client, il pulsante **Roaming..** viene visualizzato nella finestra di dialogo Gestore Licenze Bricsys.



1 Se il computer è connesso al server licenze tramite un server proxy, fare clic sul pulsante **Impostazioni proxy** per configurare le impostazioni proxy per una licenza di rete.

2 Cliccare sul pulsante **Roaming...**

Result: Viene visualizzata la finestra di dialogo **Roaming**.



3 Inserire il numero di giorni in cui desideri estrarre la licenza per il roaming e fare clic sul pulsante **Checkout**.

4 (Opzionale) Selezionare la casella di controllo se si desidera aggiornare automaticamente il periodo di roaming ogni volta che il computer è connesso al server.

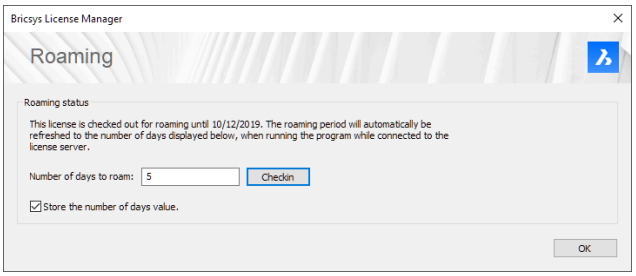
5 Riavviare **BricsCAD®** per applicare le modifiche.

2.5.1.2 Restituzione di una licenza di rete flottante al server

Se si è connessi al server delle licenze, è possibile restituire una licenza in roaming al server.

1 Fare clic sul pulsante **Roaming...** nella finestra di dialogo Gestore Licenze Bricsys.

Result: Viene visualizzata la finestra di dialogo **Roaming**.



2 Cliccare il pulsante **Checkin**.

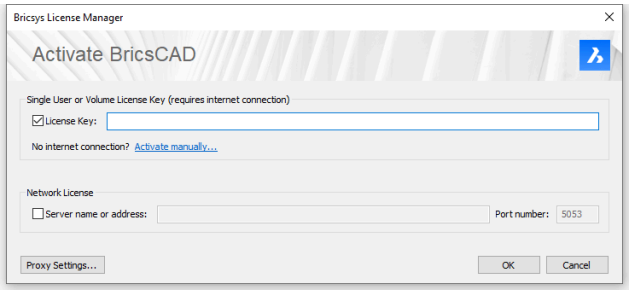
3 Riavviare **BricsCAD®** per applicare le modifiche.

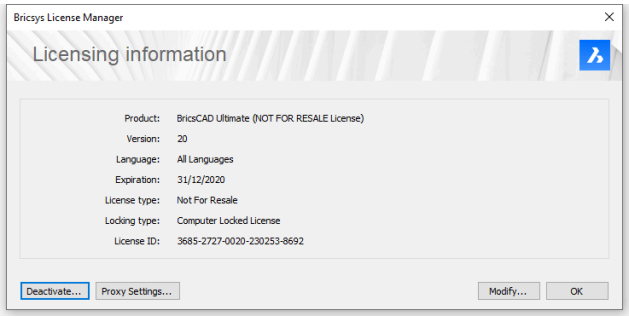
2.5.2 Impostazioni proxy su un computer client

Prima di iniziare: Alcuni computer sono connessi a una rete o a internet tramite un server proxy. Pertanto, le impostazioni proxy devono essere configurate prima di attivare o disattivare una licenza.

2.5.2.1 Configurazione delle impostazioni proxy

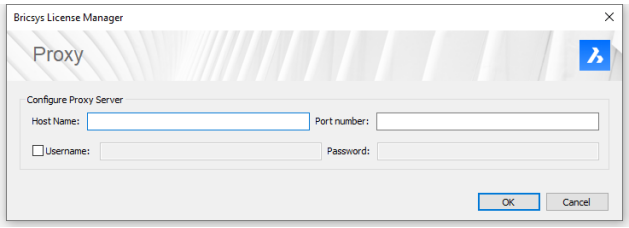
1 Fare clic sul pulsante **Impostazioni proxy...** nella finestra di dialogo **Attiva BricsCAD** o nella finestra di dialogo **Informazioni sulla licenza**.





Result: Viene visualizzata la finestra di dialogo **Proxy**.

2 Completare i campi necessari nella finestra di dialogo **Proxy**.



3 Cliccare sul pulsante **OK** per continuare.

Result:

Nota: La configurazione delle impostazioni proxy rimane attiva solo per la sessione di **BricsCAD®** corrente. Se è necessario riattivare o disattivare una licenza in una sessione futura, sarà necessario riconfigurare le impostazioni proxy.

2.5.2.2 Configurare in modo permanente le impostazioni proxy per l'uso della licenza di rete

Prima di iniziare: Se si utilizza una licenza di rete, **BricsCAD®** necessita di connettersi al server delle licenze ogni volta che viene aperta una sessione. Pertanto, le impostazioni proxy devono essere configurate in modo permanente.

Procedi come segue sul computer in cui è in esecuzione **BricsCAD®**:

1 Creare la variabile **HTTP_PROXY**: HTTP_PROXY=[LicenseServer]:[PortNumber]

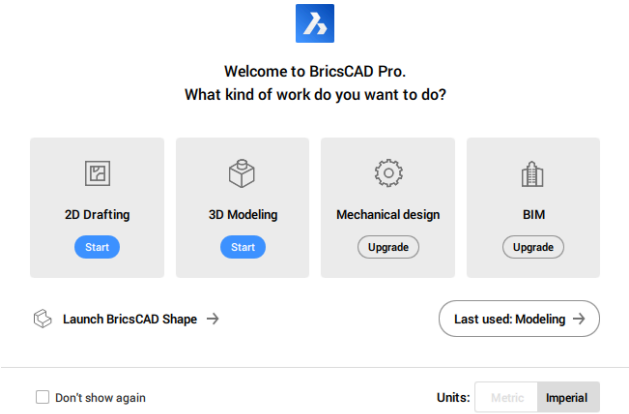
2 (Facoltativo) Se il server proxy supporta l'autenticazione di base, creare la variabile **HTTP_PROXY_CREDENTIALS**: HTTP_PROXY_CREDENTIALS=[UserName]:[Password]

3. Getting started with BricsCAD®

3.1 Avvio di BricsCAD®

3.1.1 Schermata di Avvio di BricsCAD

Se la variabile di sistema GETSTARTED è ON, la finestra di dialogo Avvio BricsCAD viene visualizzata quando si lancia **BricsCAD®**. Qui è possibile scegliere l'area di lavoro per la sessione corrente. Le aree di lavoro disponibili dipendono dal livello di licenza.



- **Disegno 2D**: avvia l'area di lavoro Disegno. E' richiesta una licenza **BricsCAD® Lite** o superiore.
- **Modellazione 3D**: avvia l'area di lavoro Modellazione. E' richiesta una licenza **BricsCAD® Pro**o superiore.
- **Mechanical**: avvia l'area di lavoro Meccanico. E' richiesta una licenza **BricsCAD® Mechanical** o **BricsCAD® Ultimate**.
- **BIM**: avvia l'area di lavoro BIM. E' richiesta una licenza **BricsCAD® BIM** o **BricsCAD® Ultimate**.
- **Avvia BricsCAD Shape**: avvia **BricsCAD® Shape**. Non è richiesta alcuna licenza.
- **Ultimo usato**: avvia l'ultima area di lavoro utilizzata.
- **Non mostrare più**: imposta la variabile di sistema GETSTARTED su 0. Per reimpostare la visualizzazione della finestra di dialogo Avvio di BricsCAD, digitare **getstarted** nella barra dei comandi e scegliere **1 per ON**.
- **Unità**: imposta le unità per il template del disegno predefinito. Definisce la variabile di sistema BASEFILE.

Nota: Lo strumento gratuito di modellazione solida, **BricsCAD® Shape**, è integrato nel singolo pacchetto di installazione di **BricsCAD®**. Al termine del periodo di prova di 30 giorni di **BricsCAD® Ultimate**, la licenza viene ripristinata in una licenza **BricsCAD® Shape** perenne, che può essere convertita in una licenza commerciale completa acquistando una chiave di licenza. Non è necessario disinstallare **BricsCAD®**.

3.2 Esplorare l'interfaccia

3.2.1 La scheda Inizio

La scheda **Inizio** si trova sopra l'area di disegno, prima delle schede del documento.

Nella scheda **Inizio** è possibile:

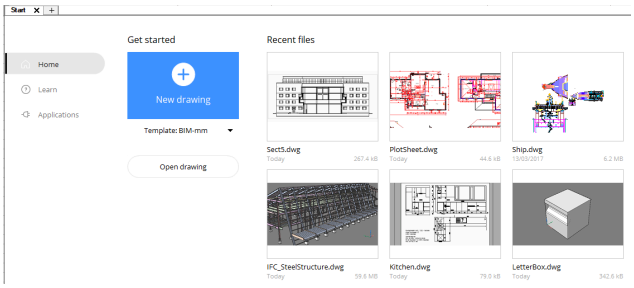
- **Aprire un disegno recente**

  Cliccare sull'immagine in miniatura di un file recente.

  Il numero di file recenti disponibili è controllato dal valore della variabile di sistema RECENTFILES.
- **Creare un nuovo disegno**

  Cliccare sul riquadro **Nuovo disegno** per creare un nuovo disegno utilizzando il modello di disegno corrente. È possibile aprire l'elenco dei **Template** per selezionare un modello di disegno diverso.
- **Aprire un disegno**

  Visualizza la finestra di dialogo **Apri file**.



La scheda **Inizio** è attiva per impostazione predefinita. Se la scheda è stata chiusa accidentalmente, è possibile riaprirla con la scorciatoia da tastiera **Ctrl+Home** o con il comando VAIASTART.

Nota: Quando si utilizza la scorciatoia da tastiera **Ctrl+Home**, assicurarsi di disattivare il blocco Num.

Nota: La variabile di sistema SHOWDOCTABS deve essere **ON** per visualizzare la scheda **Inizio**.

3.2.2 Schede Documento

Per ogni disegno attualmente aperto, una scheda documento si trova nella parte superiore dell'area di disegno.

Le schede dei documenti consentono di:

- Passare facilmente tra tutti i disegni aperti.
- Creare un nuovo disegno: fare clic sul pulsante **+** a destra delle schede.
- Chiudere un disegno: fare clic sulla **X** nella scheda del documento.

3.2.3 Scelta dell'Area di Lavoro

Le aree di lavoro predefinite sono:

- **Disegno**: utilizza la barra multifunzione.
- **Disegno (barre degli strumenti)**: utilizza menu e barre degli strumenti.

- **Modellazione**: richiede un livello di licenza **BricsCAD® Pro** o superiore; utilizza la barra multifunzione.
- **Modellazione (barre degli strumenti)**: richiede un livello di licenza **BricsCAD® Pro** o superiore; utilizza menu e barre degli strumenti.
- **Meccanico**: richiede un livello di licenza**BricsCAD® Mechanical** o **BricsCAD® Ultimate**.
- **BIM**: richiede un livello di licenza**BricsCAD® BIM** o **BricsCAD® Ultimate**.

L'area di lavoro corrente configura il layout complessivo della finestra dell'applicazione **BricsCAD®**.

Un'area di lavoro definisce il contenuto di:

- Menu
- Barre degli Strumenti
- Pannelli
- La barra multifunzione
- Le nozioni di base sul Quad

È possibile impostare l'area di lavoro corrente mediante:

- La schermata di Avvio di BricsCAD
- Il campo Area di Lavoro corrente nella barra di stato.
- Barra degli strumenti Aree di lavoro.
- Il comando AREALAVORO.

3.2.4 L'area di disegno

Quando si avvia una sessione di disegno, l'area di lavoro iniziale viene chiamata **spazio modello**. Lo spazio modello è un'area in cui è possibile creare entità bidimensionali e tridimensionali basate sul sistema WCS (World Coordinate System) o su un sistema di coordinate utente (UCS). **BricsCAD®** fornisce un'area di lavoro aggiuntiva chiamata **spazio su carta**. Ogni layout rappresenta un foglio di carta. Puoi creare fino a 255 layout sullo spazio carta in un unico disegno.

Utilizzare la scheda **Modello** per visualizzare e lavorare nello spazio modello. Fare clic su una delle schede **Layout** nella parte inferiore dell'area di disegno per passare allo spazio carta.

3.2.4.1 Spazio modello

Lo spazio modello è costituito da una singola vista che riempie lo schermo. Se necessario, è possibile creare visualizzazioni aggiuntive chiamate finestre, che possono mostrare diverse visualizzazioni del disegno o del modello 3D. È possibile gestire le finestre di disegno con il comando FINESTRE.

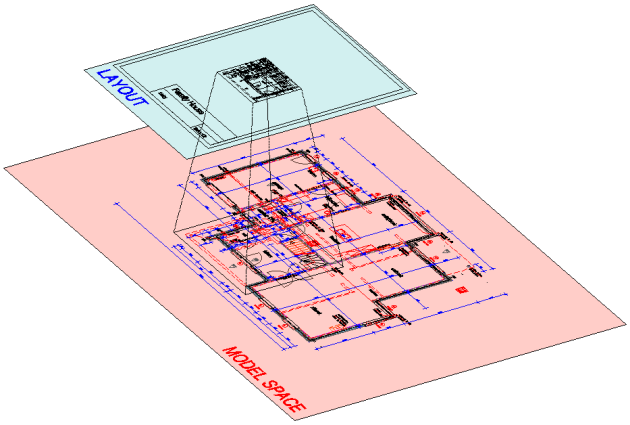
Le finestre vengono visualizzate come riquadri. Puoi lavorare contemporaneamente solo in una di queste finestre, ma tutti le finestre vengono aggiornate contemporaneamente. Fare clic su una finestra per renderla corrente.

3.2.4.2 Spazio carta

Lo spazio carta rappresenta il layout cartaceo del disegno. In quest'area di lavoro è possibile creare e disporre diverse viste del disegno in modo simile al modo in cui si organizzano i disegni su un foglio di

carta. Nello spazio carta puoi anche aggiungere keynote, annotazioni, bordi, cartigli e altre entità correlate alla stampa che non desideri visualizzare nello spazio del modello.

Ogni disegno dispone di almeno un **Layout** in cui è possibile disporre di una o più finestre di layout. È possibile considerare una finestra di layout come una finestra che si apre per visualizzare lo spazio modello:



È possibile posizionare le finestre di layout in qualsiasi punto del layout dello spazio carta. Ogni finestra di layout ha una propria scala e visibilità dei layer. È possibile stampare contemporaneamente tutte le finestre nello stesso layout.

È possibile gestire le finestre di layout con il comando FINMUL.

Lo spazio carta offre molteplici vantaggi durante la stampa del disegno:

- È possibile creare più finestre per stampare più viste. Ciascuna di queste viste può visualizzare una parte del disegno alla stessa o a scale diverse.
- È possibile creare più layout per stampare lo stesso disegno con diverse impostazioni di stampa, ad esempio spessori delle linee, file di configurazione della stampante, scala di disegno...
- È possibile aggiungere entità correlate alla stampa che non sono essenziali per il modello stesso, ad esempio keynote, annotazioni, cartigli...

3.2.5 La barra dei comandi

La barra dei comandi è un pannello agganciabile opzionale che si trova per impostazione predefinita nella parte inferiore dell'area di disegno.

La barra dei comandi è un metodo rapido ed efficiente per avviare i comandi e impostare le variabili di sistema e le preferenze utente.

BricsCAD® richiede l'input dell'utente nella barra dei comandi e riporta qui i risultati dei comandi come DIST, AREA o LISTA.

Quando la barra dei comandi è chiusa, le voci della tastiera e le richieste dei comandi vengono visualizzate nella Barra di Stato.

3.2.6 Menu

Per impostazione predefinita, la barra dei menu è nascosta in tutte le aree di lavoro.

Il contenuto del menu è definito nell'area di lavoro corrente.

3.2.6.1 Controllo della visualizzazione della barra dei menu

1 Spostare il cursore su una barra degli strumenti o un pannello multifunzione.

2 Cliccare il pulsante destro del mouse e scegliere **Barra dei Menu** nel menu contestuale.

3.2.7 Barre degli Strumenti

Le barre degli strumenti vengono utilizzate per impostazione predefinita nelle aree di lavoro Disegno (barre degli strumenti) e Modellazione (barre degli strumenti), ma possono essere utilizzate in qualsiasi area di lavoro.

- È possibile scegliere icone piccole (16 x 16), grandi (32 x 32) o molto grandi (64 x 64) sulle barre degli strumenti.
- Su alcune icone, una piccola freccia nera nell'angolo in basso a destra dell'icona indica che è disponibile un barra a comparsa. Una barra a comparsa contiene un gruppo di strumenti correlati. Tenere premuto il tasto sinistro del mouse per espandere la barra a comparsa e scegliere uno degli strumenti a comparsa. Lo strumento scelto l'ultima volta rimane visibile nella barra degli strumenti compressa.
- E' possibile aprire o chiudere le barre degli strumenti in qualsiasi momento.

3.2.8 La barra multifunzione

La barra multifunzione è un insieme di barre degli strumenti a schede, riempite con pulsanti degli strumenti e altri elementi grafici di controllo, raggruppati per funzionalità.

Il contenuto della barra multifunzione dipende dall'area di lavoro corrente.

Le aree di lavoro Disegno, Modellazione, Meccanico e BIM utilizzano la barra multifunzione per impostazione predefinita.

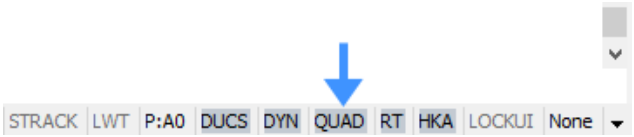
Utilizzare il comando BARRAMULTIF e il comando CHIUDIBARRAMULTIF per aprire o chiudere la barra multifunzione.

3.2.9 Le nozioni di base sul Quad

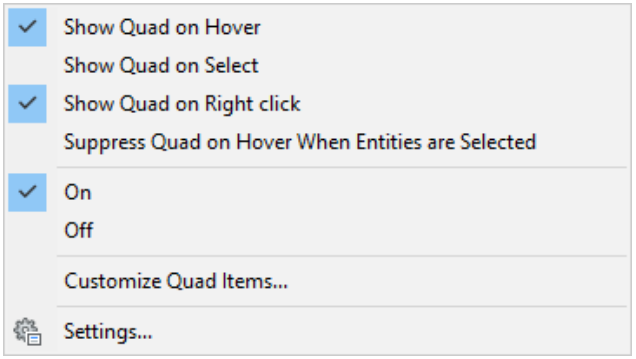
Il **Quad** è un sistema di comandi completo situato direttamente sul cursore del mouse.

Mentre si sposta il cursore sullo schermo o quando esiste un gruppo di selezione, il Quad rileva il tipo di entità e presenta i comandi che funzionano con quell'entità o con il gruppo di entità selezionate. Quando non viene rilevata o selezionata alcuna entità, è possibile utilizzare il Quad per creare entità, inserire blocchi, attaccare i sottoposti o aprire le finestre di dialogo **Impostazioni** e **Esplora Disegno**.

- Per attivare o disattivare la visualizzazione del **Quad**, fare clic sul campo Quad nella Barra di Stato nella parte inferiore della finestra dell'applicazione **BricsCAD®**.



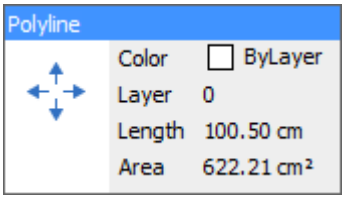
- Cliccare il pulsante destro del mouse sul campo **Quad** per definire le impostazioni del Quad in un menu contestuale.



3.2.9.1 Procedura di base per modificare le entità utilizzando il Quad

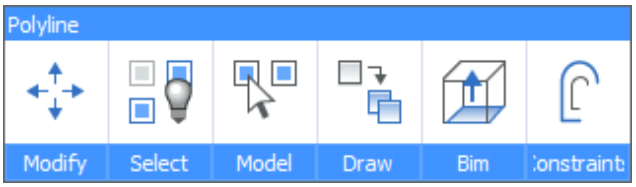
1 Spostare il cursore sull'entità. L'entità si evidenzia e il Quad visualizza il comando utilizzato più di recente con questo tipo di entità.

Se la variabile di sistema ROLLOVERTIPS = ON, le proprietà dell'entità vengono visualizzate nel Quad.



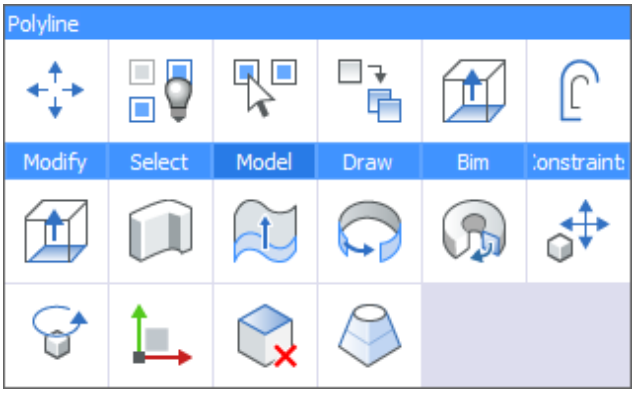
2 Spostare il cursore sul Quad per visualizzare i comandi utilizzati più di recente con questo tipo di entità.

Alcune schede con le categorie dei comandi vengono visualizzate in basso.



3 Spostare il cursore sopra le etichette delle schede dei gruppi di comandi per espandere il relativo

gruppo di comando.



4 Fare clic su un'icona per avviare il comando.

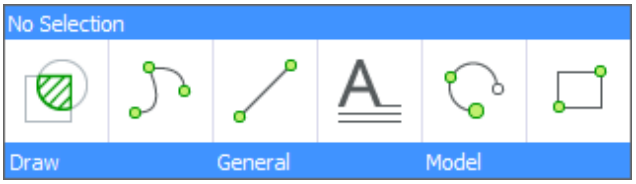
3.2.9.2 Utilizzare il Quad su Nessuna Selezione

Assicurati che l'opzione **Visualizza il Quad su tasto destro** sia attivata.

1 Eseguire una delle seguenti operazioni:

- Spostare il cursore su un'area vuota della finestra di disegno e cliccare il pulsante destro del mouse.
- Quando il cursore si trova su un'entità, tenere premuto più a lungo il pulsante destro del mouse.

Viene visualizzato il messaggio **Nessuna selezione**, che mostra le icone dei comandi utilizzati più di recente.



2 Eseguire una delle seguenti operazioni:

- Fare clic su uno dei comandi utilizzati più di recente.
- Spostare il cursore su una scheda di comando e selezionare un comando quando si espande.

3 Eseguire il comando.

3.2.10 Menu del Tasto Destro

I menu contestuali o i menu di scelta rapida visualizzano un elenco di comandi accanto al cursore quando si clicca il pulsante destro del mouse.

Quando il Quad è attivo, tenere premuto il pulsante destro del mouse un po' più a lungo (= tasto destro del mouse lungo).

Il contenuto di un menu di scelta rapida dipende dall'entità o dal gruppo di selezione corrente.

Quando il Quad non è attivo e non è selezionato nulla, con il pulsante destro del mouse viene ripetuto il comando più recente. Tenere premuto un po' più a lungo il pulsante destro del mouse per visualizzare il menu contestuale.

3.2.11 La barra di stato

La barra di stato si trova nella parte inferiore dell'area di disegno.

- Mostra i suggerimenti degli strumenti quando si passa il mouse sopra un pulsante della barra di stato.
- Permette di immettere i nomi dei comandi e le risposte alle opzioni dei comandi quando la barra dei comandi è chiusa.
- Visualizza lo stato attuale delle impostazioni dell'entità: layer, colore, stile linea, ... e consente di modificare queste impostazioni.

  Nota: Per impostazione predefinita, vengono disattivati in qualsiasi area di lavoro.
- Visualizza lo stile attuale della quote e dei testi e consente di impostarli.
- Visualizza lo stato corrente degli aiuti di disegno come lo snap ad entità e consente di attivarli/disattivarli.
- ...

Fare clic sulla piccola freccia verso il basso nera sul lato destro della barra di stato per gestire la visualizzazione dei campi.

3.2.12 Pannelli

I pannelli sono finestre di dialogo agganciabili.

I pannelli disponibili sono:

- Pannello Editor Animazione
- Allegati
- BIM BCF
- Composizioni BIM
- Profili BIM
- BIM - Navigatore Progetto
- Gestore BOM
- Bricsys 24/7
- Barra dei Comandi
- Navigatore Contenuto
- Dettagli
- Personalizzazioni Disegno
- Layer
- Libreria
- Navigatore Meccanico
- Parametri e Vincoli

- Proprietà
- Materiali di Rendering
- Report
- Barra Multifunzione
- Gruppo di Fogli
- Struttura
- Suggerimenti
- Strumenti Tavolozze

3.2.13 Modalità schermo intero

In modalità schermo intero, è possibile massimizzare l'area di disegno eliminando gli elementi dell'interfaccia utente quali la barra dei menu, la barra multifunzione, le barre degli strumenti, i pannelli, la barra dei comandi, la barra di stato e le schede dei documenti.

È ancora possibile utilizzare il Quad per avviare i comandi.

Tenere premuto il tasto Ctrl e premere il tasto 0 (zero) per attivare/disattivare la modalità schermo intero.

Nota: Non utilizzare il tastierino numerico.

Nota: Nelle tastiere AZERTY: non premere il tasto Maiusc quando si preme il tasto 0.

3.2.14 Widget Look From

Il widget Look From consente di controllare la visualizzazione del disegno. È possibile selezionare un numero di visualizzazioni preimpostate, ad esempio viste ortografiche e viste isometriche.

Per impostazione predefinita, il widget **Look From** viene visualizzato nell'angolo in alto a destra della schermata grafica.

Il comportamento del widget **Look From** può essere controllato da una serie di variabili di sistema e preferenze utente.

3.3 Lavorare in BricsCAD®

3.3.1 Creare un disegno

Esistono diversi modi per creare nuovi disegni in BricsCAD:

- Nella scheda **Inizio**.
- Utilizzando il comando CNUOVO.
- Utilizzando il comando NUOVO.
- Utilizzando il comando NEWWIZ.
- Doppio clic su un file *.dwt in una finestra di Windows Explorer.

Nota: Un nuovo disegno viene sempre creato come copia di un template. Il modello predefinito è salvato tramite la variabile di sistema BASEFILE.

3.3.2 Template di Disegno

Un template di disegno (modello) è un file di disegno con estensione file DWT.

Viene utilizzato per creare un nuovo disegno che è una copia del template. Qualsiasi disegno con estensione DWT è considerato un template. I template di **BricsCAD®** si trovano nella cartella template. Utilizzare il comando TEMPLATEFOLDER per aprire la cartella. Se si salvano i template personalizzati nella cartella template, questi saranno disponibili nella scheda Inizio per creare nuovi disegni. L'utilizzo di modelli adeguatamente preparati è essenziale per un uso produttivo di **BricsCAD®**.

Un template non contiene entità nello spazio del modello, ma contiene impostazioni in base ai requisiti specifici di disegno e di progetto:

- Unità: imperiale o metrica e visualizzazione delle unità.
- Layer.
- Stili di quota e Stili multidirettrice.
- Stili di testo.
- Stili di tabella.
- Più layout, ciascuno con il proprio cartiglio, dimensioni carta, stampante e stile di stampa.
- Blocchi standard per note, tabelle, simboli e altre entità.

3.3.3 Impostazioni creazione entità

Le nuove entità vengono create nel layer corrente, utilizzando le impostazioni di creazione entità correnti.

- Colore
- Tipolinea
- Scala Tipolinea.
- Spessorelinea
- Trasparenza

È possibile definire le impostazioni di creazione delle entità nella sezione **Generale** del Pannello delle Proprietà.

È possibile definire tutte le impostazioni di creazione delle entità **ByLayer**. Ciò significa che viene utilizzata l'impostazione corrispondente del layer corrente.

Potete definire le proprietà dei layer nella sezione Layer della finestra di dialogo Esplora Disegno.

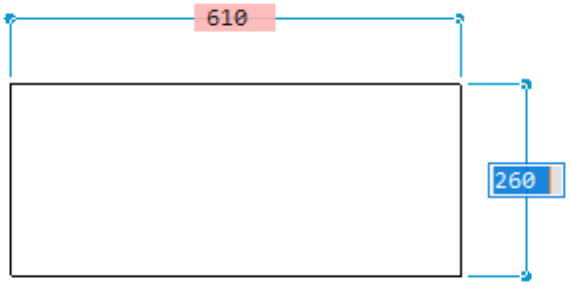
3.3.4 Come disegnare con precisione

Le nuove entità vengono create nel piano XY del sistema di coordinate corrente. Per impostazione predefinita, questo è il Sistema di Coordinate Globale (WCS). Tutte le dimensioni di un file DWG sono espresse in unità di disegno (UD). La variabile di sistema INSUNITS (Unità di Inserimento) imposta il valore di 1 UD.

Per i nuovi disegni, la variabile di sistema INSUNITS viene definita nel disegno del template selezionato per creare il disegno. **BricsCAD®** fornisce una serie di template per unità decimali e imperiali.

Per disegnare con precisione, **BricsCAD**® fornisce diversi aiuti per il disegno:

- Snap ad entità 2D: è possibile utilizzare punti chiave su entità 2D esistenti, come punto finale, punto intermedio o centro e intersezione. È anche possibile disegnare perpendicolarmente, parallelamente o tangenzialmente a un'entità.
- Snap entità 3D: è possibile utilizzare punti chiave su entità 3D, ad esempio solidi.
- Dimensioni dinamiche. Quando le dimensioni dinamiche sono **ON**, è possibile inserire distanze e angoli nei campi delle dimensioni dinamiche. Quando vengono visualizzati più campi di dimensione dinamica, ad esempio quando si disegna un rettangolo, premere il tasto Tab per passare al campo successivo. Quando si immette un valore e si preme il tasto Tab, il campo diventa rosso, il che indica che il valore è fisso.



- Tracciamento polare forza la direzione di disegno ad angoli specificati rispetto agli assi (X, Y, Z) del sistema di coordinate corrente.
- Tracciamento snap ad entità combina gli snap ad entità e il tracciamento polare.
- Input coordinate: consente di immettere le coordinate XYZ per specificare i punti.
- Snap e griglia. Una griglia è un insieme di linee o punti equidistanti che servono come riferimento visivo delle distanze. Il cursore si aggancia ai punti della griglia, consentendo di disegnare velocemente a intervalli fissi.
- Limiti di disegno consente di limitare l'area di disegno, evitando la creazione di entità al di fuori dei limiti del disegno.
- Modalità orto forza il cursore a spostarsi parallelamente all'asse X o Y del sistema di coordinate corrente.
- Sistema coordinate utente È possibile definire sistemi di coordinate utente denominati (UCS) situati in qualsiasi punto dello spazio tridimensionale. È possibile ripristinare un sistema UCS con Nome in qualsiasi momento.
- Inserimento diretto della distanza consente di inserire lunghezze, raggi, diametri e vettori di spostamento.
- La Distanza più Vicina consente di visualizzare e modificare dinamicamente la distanza tra due entità o entità secondarie selezionate.
- Il Manipolatore è uno strumento grip versatile per spostare, ruotare, scalare e specchiare rapidamente le entità selezionate.

3.3.5 Esecuzione dei comandi

In **BricsCAD**® è possibile avviare i comandi nei seguenti modi:

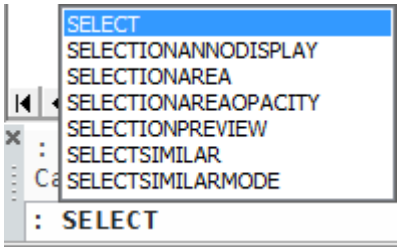
- Digitare il comando nella Barra dei Comandi.
- Scegliere il comando in un menu.
- Fare clic sul pulsante dello strumento di comando in una Barra degli Strumenti.
- Fare clic sull'icona del comando nella Barra Multifunzione.
- Scegliere il comando nel Quad.

3.3.5.1 Digitare un comando nella barra dei comandi

Se si conosce il nome del comando, è possibile inserirlo nella barra dei comandi. Quando la barra dei comandi è chiusa, le voci della tastiera e le richieste dei comandi vengono visualizzate nella Barra di Stato.

Se è impostata l'opzione **Abilita** della variabile di sistema AUTOCOMPLETEMODE, i nomi dei comandi vengono automaticamente completati mentre si digita.

Tutti i comandi e/o le variabili di sistema e/o le preferenze utente che contengono la stringa di caratteri corrente vengono visualizzati nell'elenco dei Suggerimenti.



Eseguire una delle seguenti operazioni:

- Premere Invio se viene visualizzato il comando attualmente evidenziato.
- Fare clic su un comando nell'elenco dei suggerimenti.

E' possibile lanciare alcuni comandi con un alias di comando. Ad esempio, è possibile avviare il comando LINEA immettendo L nella barra dei comandi.

3.3.5.2 Scelta delle opzioni dei comandi

Molti **BricsCAD**® comandi offrono molteplici opzioni. Queste opzioni vengono visualizzate nella barra dei comandi o nella barra di stato (quando la barra dei comandi è chiusa) e anche in un menu contestuale.

Eseguire una delle seguenti operazioni:

- Premere Invio per scegliere l'opzione predefinita. L'opzione predefinita si trova alla fine della richiesta nella barra dei comandi tra i simboli minore e maggiore (<>).
- Fai clic sulle opzioni nella barra dei comandi.
- Fare clic sull'opzione nel menu contestuale.
- Digitare la lettera maiuscola dell'opzione e premere Invio o cliccare il pulsante destro del mouse.

3.3.6 Controllo della Visualizzazione

In **BricsCAD**® è possibile controllare la visualizzazione del disegno in vari modi:

- Usando mouse e tastiera
- Utilizzando i comandi di visualizzazione
- Utilizzando il widget Look From

Le manipolazioni delle viste più frequentemente utilizzate sono:

Azione	Mouse	Comando	Look From
Visualizza Estensioni	Doppio clic sulla rotellina del mouse	**Zoom** > **Estensione**	Cliccare un'area di attivazione, la visualizzazione viene ruotata e si ingrandisce alle estensioni del disegno.
Zoom	Rotellina	**Zoom** > **In Zoom** > **Out**	
Pan	Tenere premuta la rotella del mouse ***Nota:*** Quando la variabile di sistema MBUTTONPAN = ON	Pan	
Ripristina Vista Piana (Vista dall'alto)		**Piana** > **UCS Corrente**	Fare clic sull'area di attivazione centrale

3.3.7 Selezione delle entità

Comandi di manipolazione come SPOSTA, COPIA e SPECCHIA richiedono la selezione di una o più entità. Se la variabile PICKFIRST è ON, è anche possibile comporre il gruppo di selezione prima di avviare il comando.

BricsCAD® fornisce diversi metodi di selezione. È possibile combinare i metodi di selezione per comporre un gruppo di selezione.

3.3.7.1 Selezione di entità una ad una

1 Spostare il cursore sull'entità.

Result: L'entità viene evidenziata.

Compare il Quad e viene visualizzato il comando utilizzato più di recente ed alcune proprietà dell'entità.

2 Eseguire una delle seguenti operazioni:

- Fare clic per selezionare l'entità.

- Premere ripetutamente il tasto Tab per evidenziare le entità oscurate e fare clic quando viene evidenziata l'entità desiderata.

3 Ripetere i passaggi precedenti per selezionare più entità o avviare un comando.

3.3.7.2 Selezionare più entità

BricsCAD® utilizza i metodi di selezione finestra per selezionare più entità:

1 Fare clic per definire l'angolo del rettangolo di selezione.

2 Eseguire una delle seguenti operazioni:

- Spostare il cursore verso destra. Viene visualizzato un rettangolo blu. L'icona **cerchio all'interno del rettangolo** indica che verranno selezionate tutte le entità che si trovano completamente all'interno del rettangolo di selezione.
- Spostare il cursore a sinistra. Viene visualizzato un rettangolo verde. L'icona **cerchio che si sovrappone al rettangolo** indica che verranno selezionate tutte le entità che si trovano all'interno o che si sovrappongono al rettangolo di selezione.

3 Fare clic per definire l'angolo opposto del rettangolo di selezione.

3.3.7.3 Rimozione di entità da un gruppo di selezione

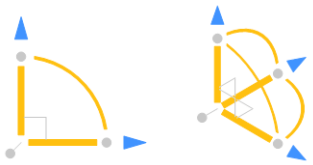
Tenere premuto il tasto Maiusc durante l'esecuzione di un metodo di selezione per rimuovere le entità dal gruppo di selezione corrente.

Premere il tasto Esc per cancellare il gruppo di selezione corrente.

3.3.8 Utilizzare il Manipolatore

Il Manipolatore è uno strumento grip versatile per spostare, ruotare, scalare e specchiare rapidamente le entità selezionate.

Tenendo premuto il tasto Ctrl durante queste azioni, le entità selezionate vengono copiate.

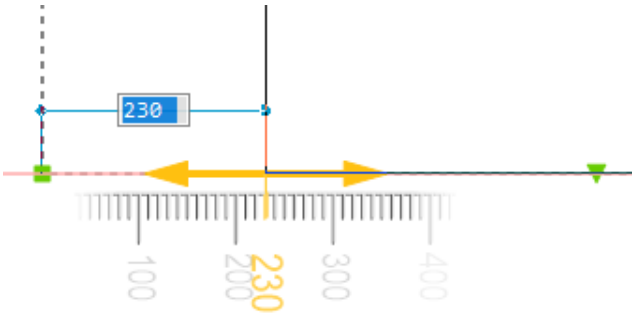


Per impostazione predefinita, il Manipolatore viene visualizzato quando si seleziona un'entità con un lungo clic (tenendo premuto il pulsante sinistro del mouse un po' più a lungo).

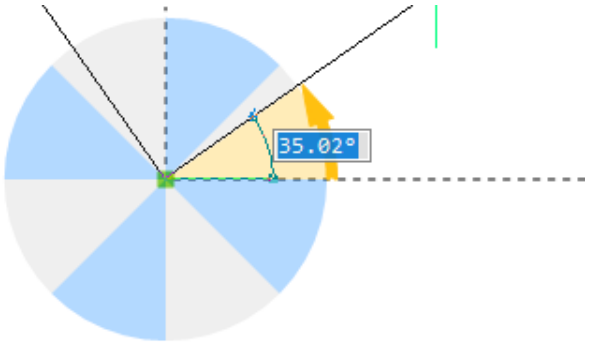
Il punto di ancoraggio del Manipolatore si trova nel punto di grip più vicino, rispetto al punto di selezione.

Posizionare il cursore su una delle zone di attivazione, per avviare un'azione:.

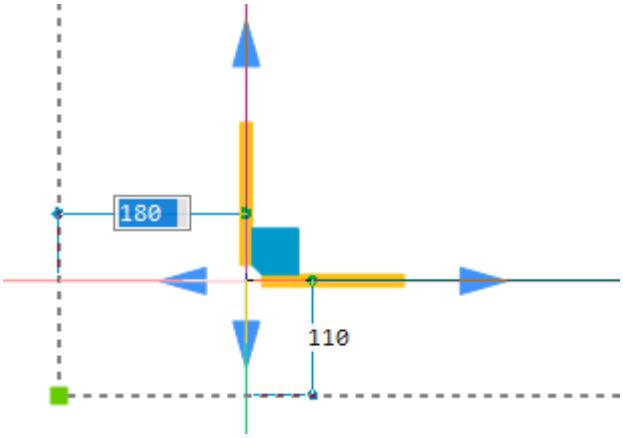
- Assi, per spostare lungo l'asse:



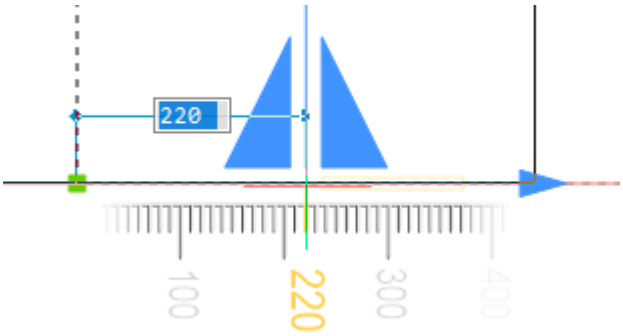
- Archi, per ruotare:



- Riquadri piano (= quadrati nel punto di ancoraggio), per spostarsi liberamente nel piano selezionato (XY, XZ o YZ):



- Teste delle frecce, per specchiare:



Nota: Nota: la dimensione del passo del righello dipende dal fattore di zoom. Ingrandire per ridurre le dimensioni del passo, eseguire lo zoom indietro per aumentare.

3.3.9 Operazioni con il Pannello Struttura

Il Pannello Struttura visualizza una vista ad albero strutturata del contenuto del disegno.

BricsCAD® fornisce 3 definizioni di configurazione struttura ad albero (CST):

- **Default** visualizza una struttura basata sui layer.
- **BIM** visualizza una struttura basata sulle entità BIM.
- **Mechanical** visualizza una struttura basata su entità.

Fare clic sul menu ad hamburger (☰) per selezionare una definizione.

Nel Panello Struttura è possibile:

- Fare clic sull'icona Espandi (**+**) per espandere un nodo.
- Fare clic sull'icona Comprimi (**-**) per comprimere un nodo.
- Selezionare un nodo, cliccare il pulsante destro del mouse e scegliere **Espandi Tutto** per espandere tutti i sottonodi del nodo selezionato.

  Selezionare il nome del disegno e utilizzate questa opzione per espandere tutti i nodi nell'intero albero..
- Selezionare un nodo, cliccare il pulsante destro del mouse e scegliere **Comprimi Tutto** per comprimere tutti i sottonodi del nodo selezionato.

  Selezionare il nome del disegno e utilizzare questa opzione per comprimere l'intero albero.
- Fare clic su un nodo o un'entità nel Pannello Struttura per selezionarlo nel disegno.

  Nota: Questo dipende dalle impostazioni della **Struttura ad Albero**. Per configurare la Struttura ad Albero, cliccare il pulsante destro del mouse su **Pannello Struttura** > **Configura** > **Opzioni**.

  Tenere premuto il tasto Ctrl per comporre un gruppo di selezione.
- Cliccare il pulsante destro del mouse per visualizzare un menu contestuale, con le seguenti opzioni:
  - **Mostra**: visualizza la selezione se precedentemente nascosta.
  - **Nascondi**: nasconde la selezione.
  - **Isola**: isola la selezione, nasconde tutto il resto.
  - **Zoom**: ingrandisce la selezione.

Nota: Selezionare il nome del disegno. Quindi, cliccare il pulsante destro del mouse e scegliere **Mostra** nel menu contestuale per visualizzare l'intero disegno oppure scegliere **Mostra Entità** nella scheda **Generale** del Quad su Nessuna Selezione).

4. Interfaccia utente

4.1 The BricsCAD application window

The components of the BricsCAD application window are:

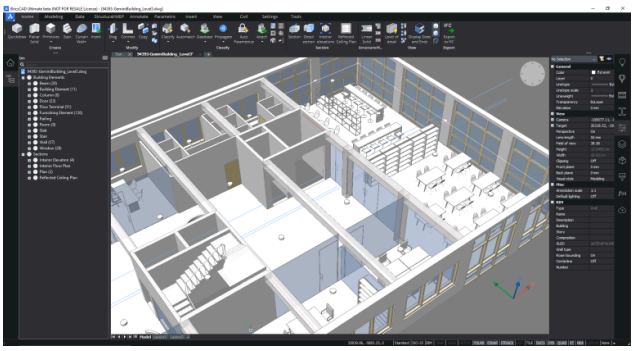
- Menu bar
- Toolbars
- Ribbon Document tabs
- Model space viewports
- Paper space viewports
- Model and Layout tabs
- Dockable panels
- Command line
- Status Bar

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

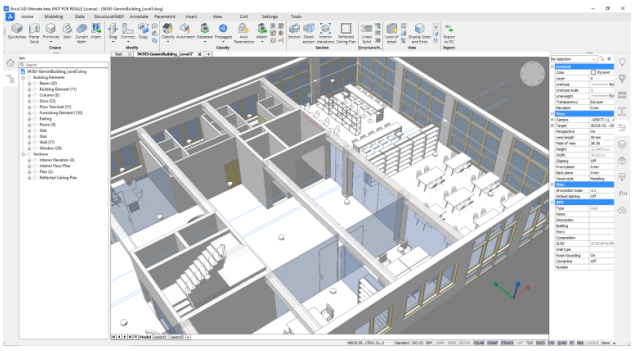
4.1.1 The color theme

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 choose 1 for light or 0 for 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 the RIBBON command or the RIBBONHIDE command.
- Right click a toolbar, a panel or an empty space in the Layout or Drawing tab area, then choose **Panels** and click **Ribbon** in the context menu.

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

4.1.4 Document tabs

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

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

4.1.5 Model and layout tabs

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

4.1.6 Scroll bars

Scrollbars are off by default.

To toggle scroll bars on/off:

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

4.1.7 The Command line

Execute the COMMANDLINE command to display the Command line.

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

4.1.8 The status bar

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

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

4.1.9 Hiding a selection of user interface elements

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

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

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

On AZERTY-keyboards:

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

4.2 The Command line

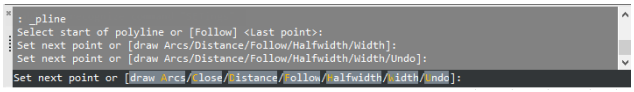
To display the Command line, do one of the following:

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

4.2.1 Setting the Command line state

The Command line can either be docked or floating.

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



To modify the height of the Command line:

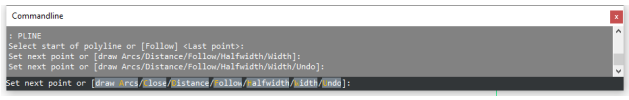
1 Place the cursor over the top edge of the Command line.

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

3 Move the mouse.

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

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



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

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

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

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

Nota:

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

4.2.2 Launching a command in the Command line

If the **Enable** flag of theAUTOCOMPLETEMODEsystem variable is set, command entries are completed automatically as you type.

All commands and/or system variables and/or user preferences that contain the current character string are displayed in the**Suggestion**list.

Do one of the following:

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

4.2.3 Setting the Command line properties

1 Open the**Settings**dialog box.

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

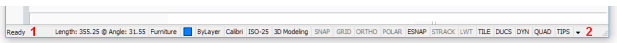
4.3 The Status bar

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

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

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

4.3.1 The Status bar fields



4.3.2 1. Status field

- Displays the status of the software.

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

4.3.3 2. Field List button

Click the arrow button (▼) to display a list of available **Status** bar fields. Click an item in the list to toggle the display of the corresponding field in the **Status** bar:

4.3.4 Cursor coordinate values

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

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

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

4.3.5 Current Layer

Displays the name of the current layer.

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

4.3.6 Current Color

Displays the current color.

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

4.3.7 Current Linetype

Displays the name of the current linetype.

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

4.3.8 Current Text Style

Displays the name of the current text style.

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

4.3.9 Current Dimension Style

Displays the name of the current dimension style.

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

4.3.10 Current Work Space

Displays the name of the current workspace.

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

4.3.11 Snap

Indicates the status of the SNAPMODE system variable.

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

4.3.12 Grid

Indicates the status of the GRIDMODE system variable.

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

4.3.13 Ortho

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

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

4.3.14 Polar

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

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

4.3.15 Esnap

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

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

4.3.16 Strack

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

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

4.3.17 Lineweight (LWT)

Indicates the status of the LWDISPLAY system variable.

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

4.3.18 Paper/Model

Allows you to choose between model space and paper space.

- When working in **Model Space**, the field reads:

  Tile (= Model space with tiled viewports).

  Click to switch to the most recently opened paper space layout.

  Right-click to choose a paper space layout.
- When working in a layout, the field reads:
- **M:Layout**: Model space with floating viewports

  or

  P:Layout: Paper space

  Click to toggle between paper space and model space with floating viewports.
- Either in Model Space or Paper Space, right-click and choose:
  - **Paper Space**: Switch to paper space.
  - **Model Space**: Switch to model space with floating viewports.
  - **Model**: Switch to model space with tiled viewports.
  - **Layout**: Switch to the selected paper space layout.

4.3.19 Annotation Scale

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

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

4.3.20 Autoscale

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

4.3.21 Tablet

Initializes the use of a drawing tablet.

4.3.22 Viewport Lock (VPLOCK)

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

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

4.3.23 Dynamic UCS (DUCS)

Indicates the status of the UCSDETECT system variable.

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

4.3.24 Dynamic Input (DYN)

Indicates the status of the DYNMODE system variable.

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

4.3.25 Quad

Indicates the status of the QUADDISPLAY system variable.

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

4.3.26 Rollover tips (RT)

Indicates the value of the ROLLOVERTIPS system variable.

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

4.3.27 Hotkey Assistant (HKA)

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

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

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

4.3.28 Lock User Interface (LOCKUI)

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

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

4.3.29 GIS Coordinate System

Displays the name of the current GIS coordinate system.

4.3.30 Tray Icon Items

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

4.4 Document tabs

4.5 Toolbars

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

4.5.1 To set the icon size

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

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

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

4.5.2 Toolbar Flyouts

On some tool buttons, a small black arrow at the bottom right corner of the icon indicates that a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar.

Each of the flyouts can be opened as a separate toolbar.

To tear off a flyout toolbar:

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

4.5.3 Opening a toolbar

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

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

   A context menu appears.
2. In the context menu, choose **Toolbars**, then choose the Main CUI file (e.g. BRICSCAD) or one of the

Partial CUI files (if any).

A list of available toolbars in the selected CUI file is displayed.

Toolbars that are already open, are ticked.

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

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

3 Click a toolbar on the context menu.

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

Nota:

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

4.6 The Quad cursor menu

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

4.6.1 Activating the Quad

Activation of the Quad is controlled by the QUADDISPLAY system variable.

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

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

4.6.2 Setting the Quad display options

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

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

4.6.3 Editing the Quad settings

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

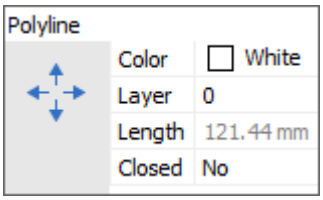
The **Settings** dialog box opens, showing the **Quad settings** screen.

4.6.4 Using the Quad

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

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

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

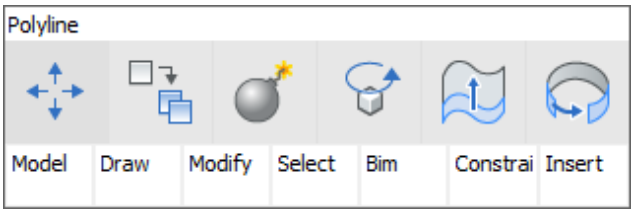


Launching the default command

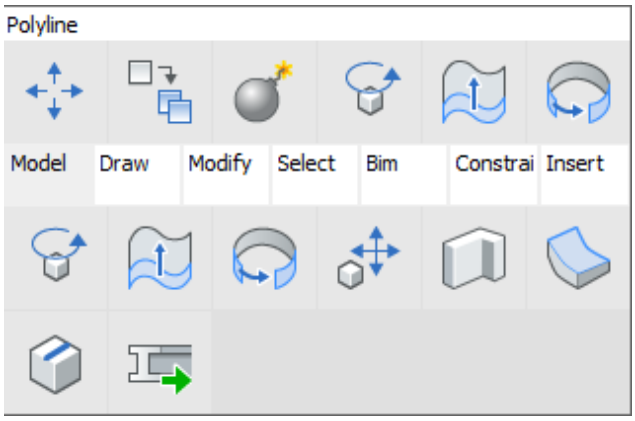
- If QUADCOMMANDLAUNCH = 1, right-click without entering the Quad to launch the command.
- If QUADCOMMANDLAUNCH = 0, click the command button.

Launching a different command

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



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



Click an icon to launch the command.

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

4.6.5 Selecting multiple entities

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

1 The entity is selected and remains highlighted.

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

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

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

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

6 The entity is removed from the selection set.

Nota:

- When the Quad is active, the value of theSELECTIONPREVIEW system variableis neglected unless a command prompts for entity selection.
- When working with solids, theSELECTIONMODESsystem variable controls which subentities are detected: edges, faces and/or boundaries.
- Holding down the Ctrl key inverts the current selection mode.
- Enable detection of 3D solid edges
- Enable detection of 3D solid faces
- Enable boundary detection
- Repeatedly press the Tab key to highlight all (sub)entities under the cursor one by one.

4.6.6 Using the Quad to create entities

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

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

The available entity creation commands in the Quad are:

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

4.7 Rollover tips

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

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

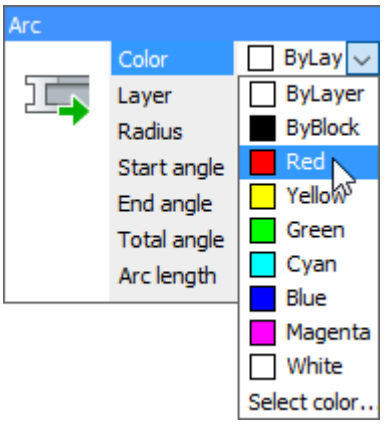
The availability of Rollover Tips is controlled through the ROLLOVERTIPS system variable.

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

Rollover tips are displayed when:

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

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



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

4.8 Prompt menus

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

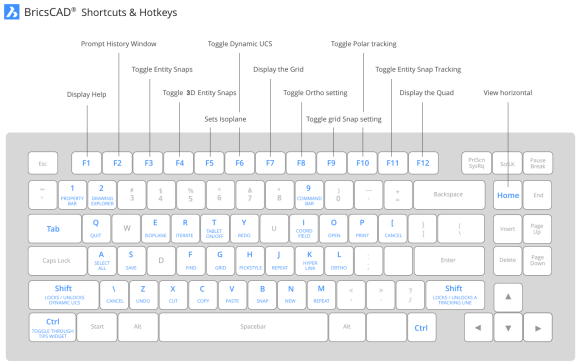
4.8.1 Controlling the display of prompt menus

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

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

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

4.9 Keyboard shortcuts



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

The following keyboard shortcuts are available:

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

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

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

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

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

4.10 Digitizing tablet

A digitizing tablet, also called a graphics tablet, a graphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

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

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows \System32.

4.10.1 Using a digitizing tablet in BricsCAD

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

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

4.10.2 Downloading the tablet overlay and the CUI files

Click here to download the file Tablet.zip.

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

The ZIP file contains the following files:

- **overlay.png**: an image of the tablet overlay
- **overlay(A3).pdf**: to print the tablet overlay on a A3 paper size.
- **overlay(cm).dwg**: a DWG in which the **overlay.png** is attached as an image.

The drawing contains two layouts to print the overlay using decimal units.

- **overlay(inch).dwg**: a DWG in which the **overlay.png** is attached as an image.

  The drawing contains two layouts to print the overlay using imperial units.
- **tablet.cui**: a partial cui file to be used with a tablet.
- **tablet(acadLike).cui**: an AutoCAD-like partial cui file to be used with a tablet.

4.10.3 Loading the overlay CUI file

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

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

Nota:

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

4.10.4 Switching the tablet ON / OFF

Do one of the following:

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

4.10.5 Configuring the tablet

1 Launch the TABLET command.

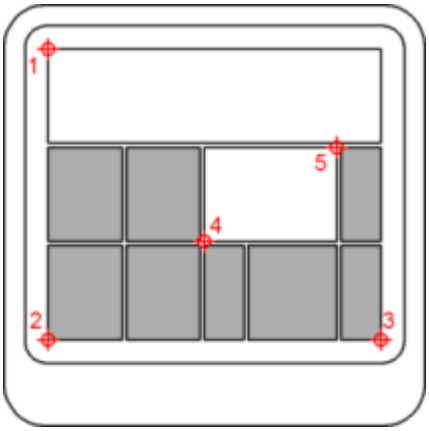
2 Choose the **Configure** option.

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

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

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

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



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

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

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

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

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

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

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

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

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

4.10.6 Calibrating the tablet

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

1 Launch the TABLET command.

2 Choose the **Calibrate** option.

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

Click a point on the digitizing tablet.

You are prompted: Enter coordinates for point #1:

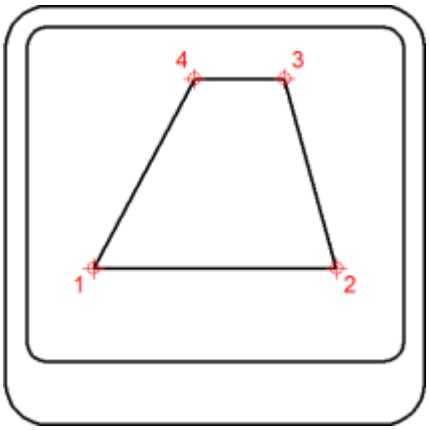
4 Do one of the following:
- Click a point in the BricsCAD drawing window.
- Enter the coordinates (X,Y) in the Command line.

5 Repeat step 3 to specify the following calibration point.

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

7 Do one of the following:
- Repeat step 4 to specify an additional calibration point. (You can specify up to 10 calibration points.)
- Press Enter to conclude the calibration process.

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



9 Select the transformation type.

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

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

4.11 Panels

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

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

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

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

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

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

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

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

The possible stack types are:

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

4.11.1 Opening or closing a panel

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

A context menu displays.

2 Place the cursor over **Panels**.

A list of available panels displays.

Panels that are currently open are checked.

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

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

4.11.2 Available panels

Icon	Title	Command
	Animation editor panel alla pagina 169	
	Attachments panel alla pagina 171	
	BIM BCF panel alla pagina 173	
	BIM compositions panel alla pagina 174	

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

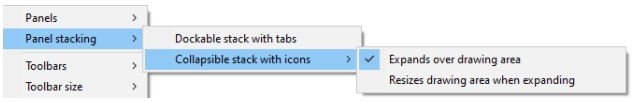
Icon	Title	Command
	Sheet sets panel alla pagina 228	
	Structure panel alla pagina 231	
	Tips panel alla pagina 233	
	Tool palettes panel alla pagina 234	Tool palettes alla pagina 860

4.11.3 Setting the stack type

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

A context menu displays.

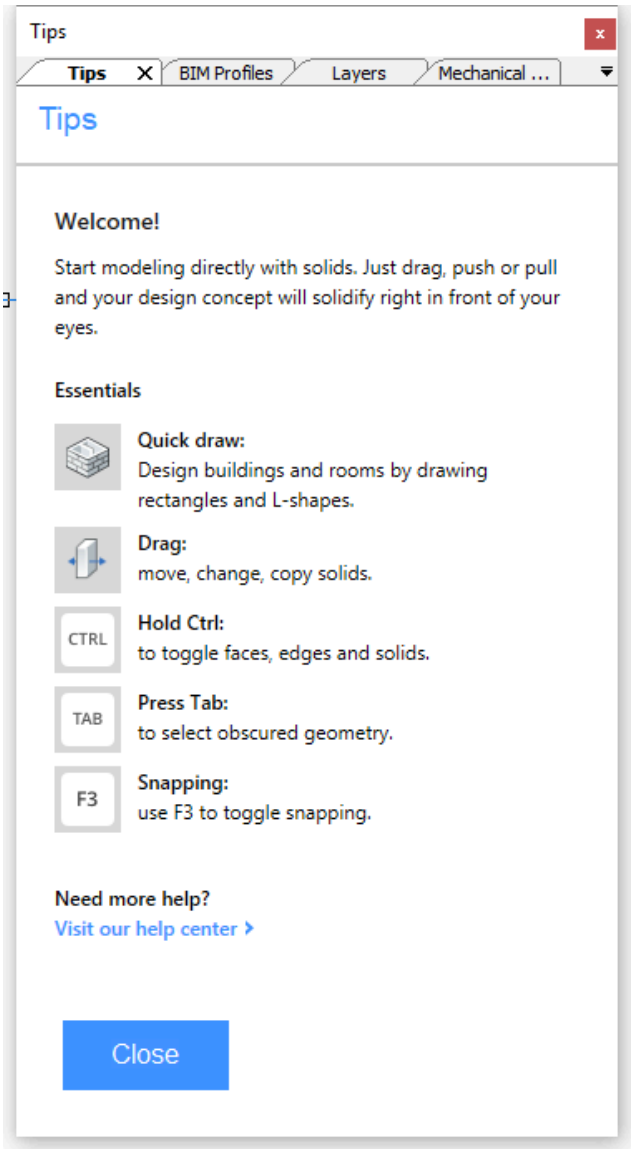
2 Choose **Panel stacking**.



3 Select the desired stack type.

Dockable stack with tabs

Panels are docked in a tabbed panel.

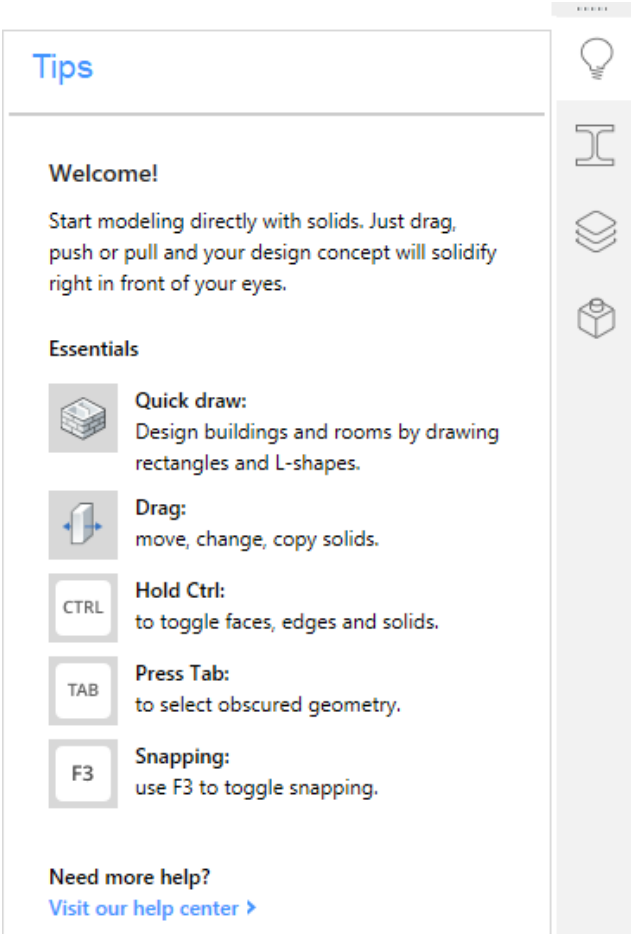


- Click a tab to display the panel.
- Press and hold the left mouse button to drag a tab to re-arrange the tab order.
- Double click a tab to make the panel floating.
- Double click the title bar of floating panel to dock it at its previous position.

Collapsible stack with flyouts

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

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

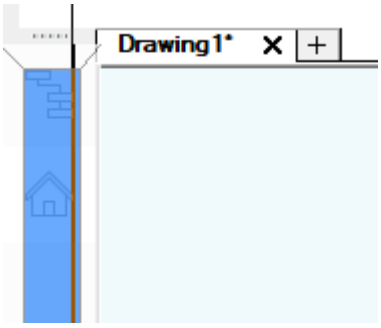


- Click an icon to expand a panel.
- To collapse an expanded panel click its icon again.
- Press and hold the left mouse button to drag an icon to re-arrange the icon order.
- Double click the icon to make the panel floating.
- Double click the title bar of a floating panel to dock it at its previous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

4.11.4 Moving a panel to the other icon tab strip

1 Drag the icon to the other icon strip.

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

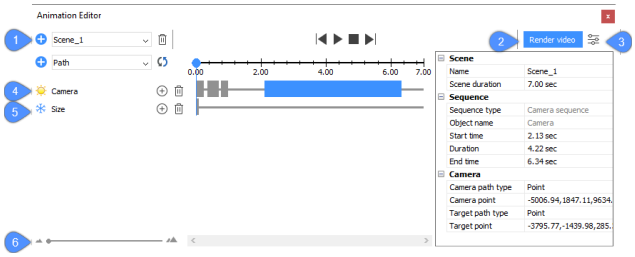


2 Move the cursor over the center docking location then release the left mouse button.

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

4.11.5 Animation editor panel

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



1 Add scene

2 Render video

3 Configurator

4 Disable timeline

5 Enable timeline

6 Slider

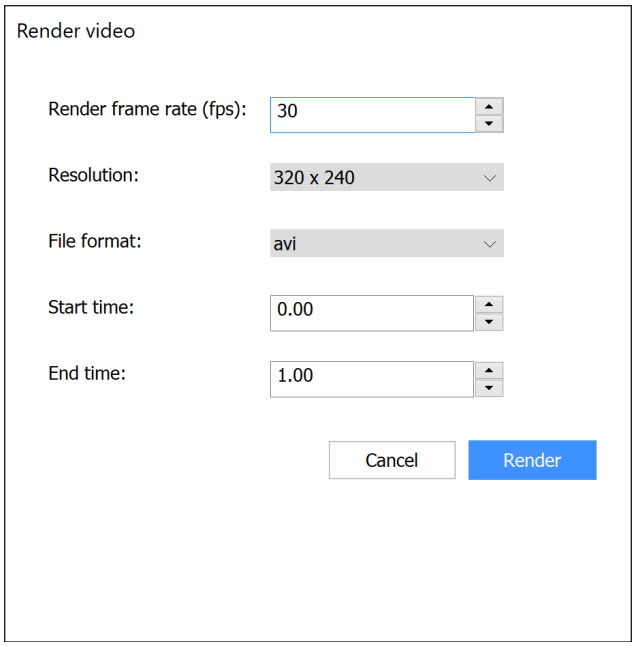
4.11.5.1 Add scene

Scene name.

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

4.11.5.2 Render video

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



4.11.5.3 Render frame rate (fps)

Sets the framerate of the video.

4.11.5.4 Resolution

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

4.11.5.5 File format

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

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

4.11.5.6 Start time

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

4.11.5.7 End time

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

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

4.11.5.8 Configurator

Toggles the configurator panel.

4.11.5.9 Disable timeline

Disables timeline effects.

4.11.5.10 Enable timeline

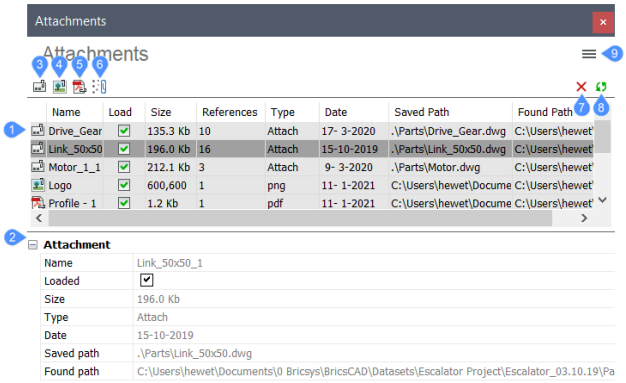
Enable timeline effects.

4.11.5.11 Slider

Move the slider to change the scale of the timeline.

4.11.6 Attachments panel

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



1 Attachment list

2 Properties

3 Attach DWG

4 Attach Image

5 Attach PDF

6 Attach Point Cloud

7 Detach

8 Refresh

9 Menu

4.11.6.1 Attachment list

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

4.11.6.2 Name

Specifies the attachment name. By default, the attachment name is the same as the file name. To rename an attachment, double-click the name and enter the new name. Renaming an attachment does not change the file name.

4.11.6.3 Load

Specifies the load status of the attachments. Attachments with the Load property enabled are loaded and displayed in the drawing editor. Attachments with the Load property disabled maintain their attachment information, such as scale and location, but are not loaded or displayed in the drawing editor.

4.11.6.4 Size

Specifies the file size of the attachments.

4.11.6.5 References

Specifies how many times the attachments are referenced in the current drawing.

4.11.6.6 Type

Specifies the attachment type. Click the attachment type to toggle between Attach and Overlay.

4.11.6.7 Date

Specifies the date the attached files were last saved.

4.11.6.8 Saved Path

Specifies the path, including location and file name, of the attachments. To change the saved path, click it and enter the new path or select the Browse button to display the Select reference file dialog box. The Browse button is only displayed after you click on the saved path.

4.11.6.9 Found Path

Specifies the path, including location and file name, of the attachments. The Found path may or may not be the same as the Saved path.

4.11.6.10 Properties

Displays the properties of the selected attachment in a row format. These are the same properties displayed in the Attachments list with a column format.

4.11.6.11 Attach DWG

Displays the Open file to attach dialog box like the XREF command.

4.11.6.12 Attach Image

Displays the Select image file dialog box like the IMAGEATTACH command.

4.11.6.13 Attach PDF

Displays the Select PDF underlay file dialog box like the PDFATTACH command.

4.11.6.14 Attach Point Cloud

Displays the Preprocessed Point Cloud Data File dialog box like the POINTCLOUDATTACH command.

4.11.6.15 Detach

Detaches the selected attachment from the drawing.

4.11.6.16 Refresh

Updates the display.

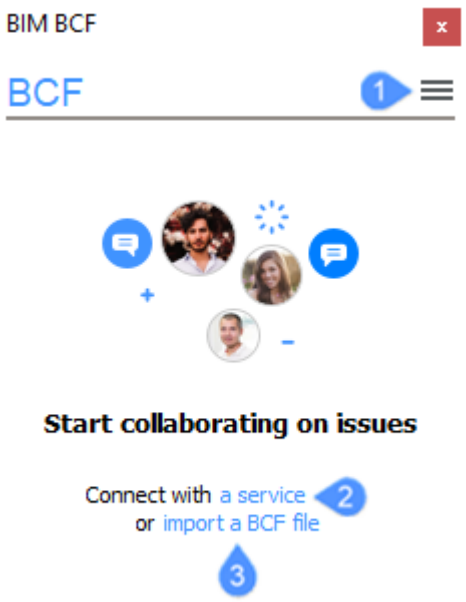
4.11.6.17 Menu

Grid View
Displays the attachments in a detailed grid view.

Tree View
Displays the attachments in a structured tree view.

4.11.7 BIM BCF panel

The BIM BCF panel allows you to link information such as a comment, a screenshot, a list of involved objects and a camera viewport to an issue.



1 Menu

2 Connect with a service

3 Import a BCF file

4.11.7.1 Menu

Opens the main menu of BIM BCF panel.

Import BCF file
Allows to import BIM Collaboration Format Files (*.bcf,*.bcfzip).

Log in with custom service
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

Log in with custom service
Allows you to connect to a cloud service to start collaborating on issues in the drawings.

Refresh
Makes sure you see the latest updates in case you are working on the cloud.

Clear sources
Disconnects you with the cloud service, in case you are online, or closes the .bcf file, in case you are working on an imported .bcf file.

4.11.7.2 Connect with a service

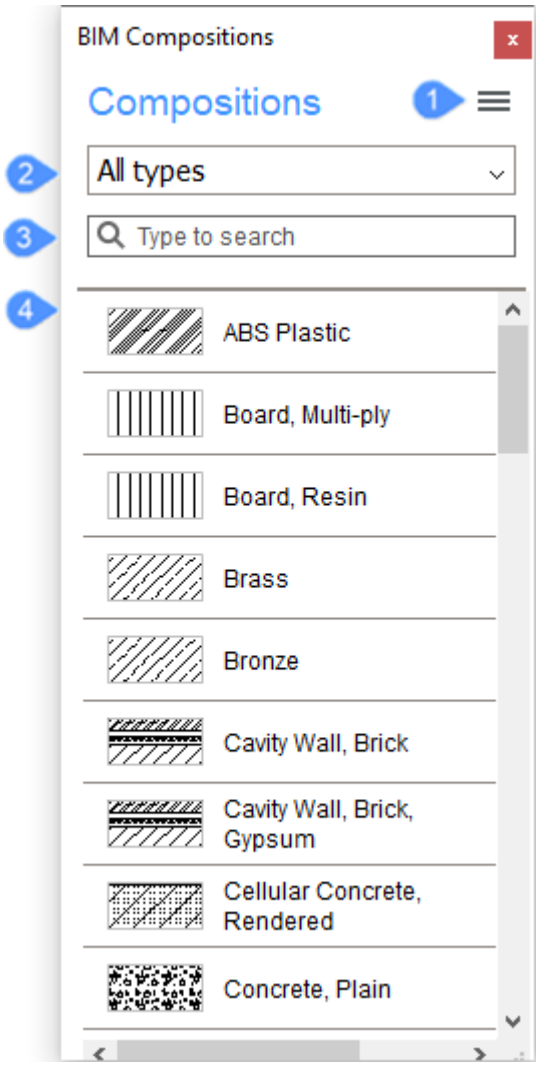
Opens the Custom log in dialog box.

4.11.7.3 Import a BCF file

Displays the Select a BIM Collaboration Format file to import dialog box.

4.11.8 BIM compositions panel

The BIM composition panel allows you to assign compositions to any solid in a BIM model.



1 Menu

2 Compositions types list

3 Search

4 Available compositions

4.11.8.1 Menu

Open material dialog...
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

Open composition dialog...
Opens the Compositions dialog box that allows you to create, modify and delete compositions.

Open project dialog...
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

Show only compositions in project
Displays only the compositions that are used in the project.

4.11.8.2 Compositions types list

Displays the compositions of the selected category (all types, generic, roof, slab, wall) in the Library database.

4.11.8.3 Search

Filters the list of compositions. Only compositions that contain the current search term are displayed.

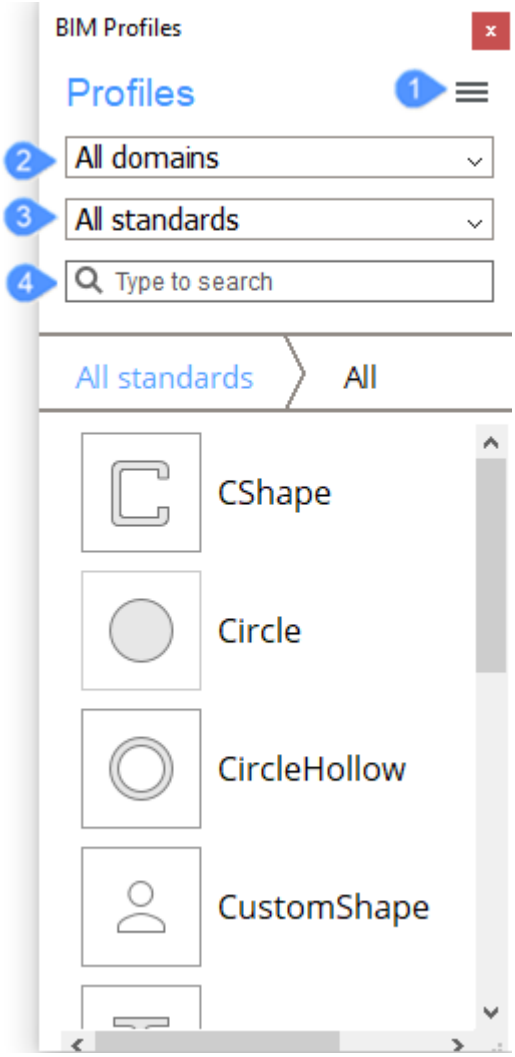
4.11.8.4 Available compositions

Displays all the available compositions.

When you double-click on a composition the Compositions dialog box will open.

4.11.9 BIM profiles panel

The BIM Profiles panel offers a central location to access standard and custom profiles.



1 Menu

2 Domains

3 Standards

4 Search bar

4.11.9.1 Menu

Open material dialog...
Opens the Physical Material dialog box that allows you to create, modify and delete physical materials.

Open profile dialog...
Opens the Profiles dialog box that allows you to create, modify and delete profiles

Open project dialog...
Opens the BIM Project Info dialog box that allows you to control the location, properties, and content of the Project and Library databases.

Show only profiles in project
Displays only the profiles that are used in the project.

4.11.9.2 Domains

Filter profiles based on the domain.

4.11.9.3 Standards

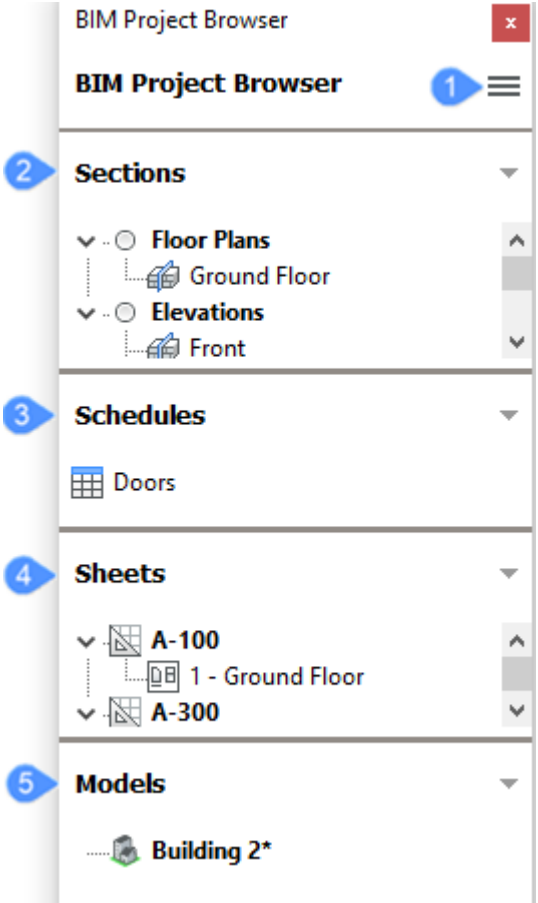
Filter profiles based on the country standard.

4.11.9.4 Search bar

Search for specific profiles using the search bar.

4.11.10 BIM project browser panel

The BIM project browser panel offers a central location to manage your BIM projects.



1 Menu

2 Sections

3 Schedules

4 Sheets

5 Models

4.11.10.1 Menu

Refresh
Refreshes the BIM Project Browser panel to match changes in the model.

Project Setup
Opens the Project Setup dialog box that gives a complete overview of the project's files, in different categories.

Sheetset Setup
Opens the Sheetset Setup dialog box that allows to set up the complete set of sheets required to fit all bimsections in the project.

Enable Background Update
Calculates section results automatically, without interrupting the user workflow.

4.11.10.2 Sections

Displays a list of project sections organized by type: floor plans and elevations. You can expand a section type to see its sections.

A context menu displays when you right-click on a section:

- **Section Properties...**: opens the Section Properties dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: selects the section entity in the project model drawing.
- **Rename**: renames the section entity.
- **Update**: recalculates the section result.
- **Display Section Result**: displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

4.11.10.3 Schedules

Displays a list of project schedules.

A context menu displays when you right-click on the Schedules section.

- **Add Schedule (Wizard)...**: opens the Wizard Page dialog box that allows you to create a schedule based on a new or an existing data extraction definition.
- **Add Schedule (From dxd)...**: opens the Create Schedule dialog box that allows you to quickly add a BIM schedule, without the creation of a new .dxd file.

A context menu displays when you right-click on a schedule.

- **Schedule Properties...**: opens the Schedule Properties dialog box that allows you to modify the properties of a schedule.
- **Create/Update Table**: creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: displays the schedule in the resulting drawing. If not already open, the drawing is opened.

- **Edit**: opens the Wizard Page dialog box that allows you to edit the schedule.
- **Remove...**: displays the Remove dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

4.11.10.4 Sheets

Displays a list of project sheets. You can expand a sheet name to see its sheet views.

A context menu appears when you right-click on the Sheets section:

- **Sheetset Setup...**: opens the Sheetset Setup dialog box that allows you to create a layout from scratch.
- **Add Subset**: adds a subset in the Sheets tab.
- **Add Sheet...**: adds a new sheet in a subset in the Sheets tab.
- **Sheetset Options...**: opens the Sheet sets tab in the Settings dialog box that allows you to modify the sheet sets options.

A context menu displays when you right-click on a sheet name:

- **Sheet Properties...**: opens the Sheet Properties dialog box that allows you to modify the properties of a sheet.
- **Open Sheet**: opens the corresponding sheet.
- **Remove...**: opens the Remove dialog box that allows you to choose between **Result Drawing** and **Sheet** and removes the sheet.

A context menu displays when you right-click on a sheet view:

- **View Properties...**: opens the Sheet View Properties dialog box that allows you to modify the properties of a sheet view.
- **Display View**: displays the corresponding result and zooms in on its viewport.
- **Update**: updates the sheet view. It executes the BIMSECTIONUPDATE command for the related BIM section.

A context menu displays when you right-click on a subset name:

- **Subset Properties...**: opens the Subset Properties dialog box that allows you to modify the properties of a subset.

4.11.10.5 Models

Displays a list of project models.

A context menu displays when you right-click on a model with the following options:

- **Model Properties...**: opens the Model Properties dialog box that allows you to see more information about the model.
- **Close Drawing**: for a currently open model,it closes the model.
- **Open Drawing**: for a currently closed model, it opens the model.
- **Remove from Model List**: removes the drawing from the project model drawings.

4.11.11 BOM manager panel

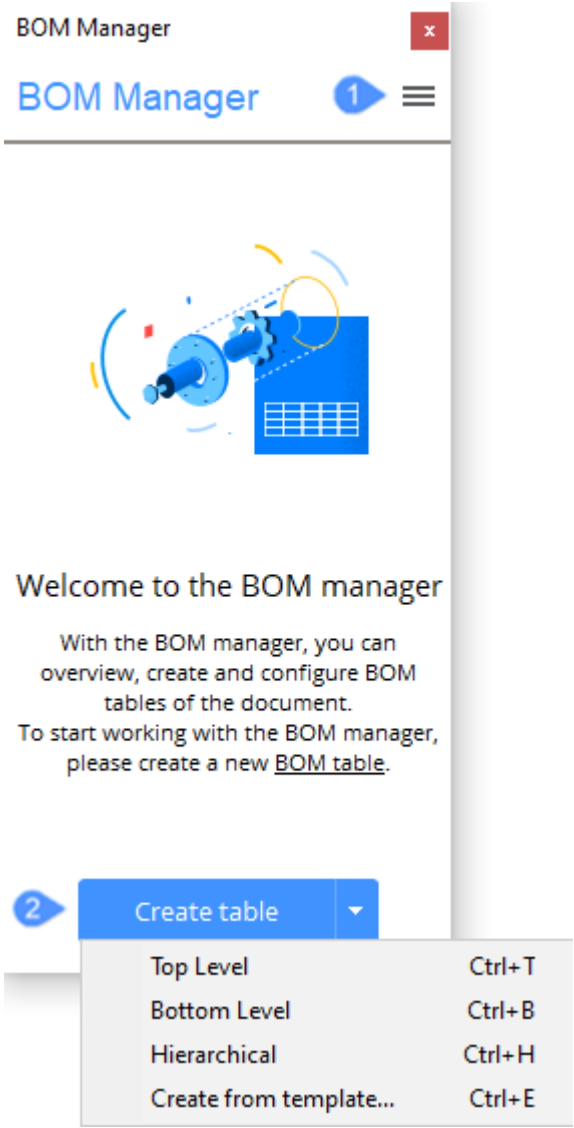
The BOM Manager panel allows you to browse, create and edit bills of materials (BOMs) in the current drawing.

The configuration of a bill of materials can be saved as a template file and reused later in other drawings.

With the Bill of Materials Manager, you can:

- Browse existing bills of materials in the drawing.
- Create a new bill of materials using either a standard type (such as top-level, bottom-level, hierarchical) or an existing BOM template.
- Edit bills of materials:
- Add or remove columns (includes formula columns).
- Move selected columns to new positions.
- Rename the columns.
- Configure how the BOM table is sorted.
- Configure the footer value for each individual column.
- Configure the units and how they are displayed for each column.
- Configure the width of each column with column properties.
- Configure the visibility of each individual column.
- Filter the components with a filter expression.
- Change the title of both the title row and the footer row.
- Search for components in the table.
- Highlight and select component instances.
- Save the BOM configuration as a template file.
- Load a BOM table configuration from a template file.
- Place the BOM table in a drawing.
- Delete BOM tables.

A Mechanical or Ultimate license is required to modify or create bills of materials. However, the manager is available with a Pro license.



1 Menu

2 Create a table

4.11.11.1 Menu

Opens the main menu.

New BOM

Opens a submenu, which includes:

- **Copy the current table**: adds a new BOM table using the template of the current table.

BOM tables in document

opens a submenu.

- **Go to first table**: switches to the first BOM table in the document.
- **Go to previous table**: Switches to the previous BOM table in the document.
- **Go to next table**: Switches to the next BOM table in the document.

- **Go to last table**: Switches to the last BOM table in the document.
- **<Name of the BOM>**: Switches to the corresponding BOM table in the document.

Place/Update BOM table
Starts the interactive placement of the table in the drawing. If the table is already in the drawing, this tool will update the table.

Export to CSV
Saves the content of the current table to a CSV file.

Replace from template…
Replaces the current BOM configuration with the one from the template.

Save as template…
Specify the location of the BOM template file.

Delete current BOM
Deletes the current BOM table from the document.

Set filter
Opens the Filter Configuration dialog box, which allows you to specify the filter configuration.

Selection set
Opens a submenu.

- **Entire model**: The BOM table will be created for the entire model.
- **Subassembly**: The BOM table will be created for a selected instance of a subassembly.
- **Drawing view**: The BOM table will be created for the parts selected for the corresponding drawing view, regardless of their visibility in the viewport.
- **Layout viewport**: The BOM table will be created for the selected layout viewport.
- **Custom selection**: The BOM table will be created only for the selected parts.
- **Show selection**: Shows the current selection for the BOM table.
- **Edit**: Allows you to edit the selection.

Refresh
Updates the current BOM and the list of all BOM tables in the drawing.

Insert column after
Inserts a column from the **Column selector** after the selected column.

Insert column before
Inserts a column from the **Column selector** before the selected column.

Nota: If no column is selected, a new column is added after the last column in the table.

Insert formula column
Opens a formula editor, which allows you to configure a formula column.

Insert template column
Opens a template editor, which allows you to configure a template column.

Edit evaluated column
Allows you to edit the selected formula or template column.

Delete column
Removes the selected column from the table.

Move column left
Moves the selected column(s) one column to the left.

Move column right
Moves the selected column(s) one column to the right.

Make column visible
Shows the selected column in the associated table in the drawing.

Make column invisible
Removes the selected column from the associated table. Invisible columns remain visible in the BOM manager but are marked with a diagonal hatch pattern in the background.

Associated table
Opens a submenu.

- **Zoom to**: Zooms to the table. If necessary, sets the table layout as the current one.
- **Select**: Selects the table. If necessary, sets the table layout as the current one.
- **Delete**: Deletes the table from the document.
- **Detach**: Detaches the table. The table in the document is no longer updated when the BOM table is modified.

Footer
Opens a submenu, for each of the selected columns.

- **No footer**: Removes the footer.
- **Sum**: Displays the sum in the footer.
- **Average value**: Displays the average in the footer.
- **Minimal value:**: Displays the minimum value in the footer.
- **Maximal value:**: displays the maximum value in the footer.

Sorting
Opens a submenu.

- **Automatic order**:Sorts the table automatically**.**
- **Natural order**: Sort the table by the natural order of component instances in the document.
- **Ascending order**: Sorts the selected columns in ascending order.
- **Descending order**: Sorts the selected columns in descending order.

Aggregate function
Opens a submenu.

- **No function**: Removes the aggregate function.
- **Sum**: Displays the sum of all values in the group.
- **Average value**: Displays the average of all values in the group.
- **Minimal value**: Displays the minimum of all values in the group.
- **Maximal value**: Displays the maximum of all values in the group.

- **Concatenation**: displays a concatenation of all values in the group in ascending order, separated by a specified delimiter.
- **Concatenation with count**: Displays a concatenation of all values in the group with their corresponding number of encounters in ascending order. Value and count pairs are separated by a specified delimiter.

Disable/Enable warnings
Controls the display of warning messages during the creation or update of BOM tables.

4.11.11.2 Create a table

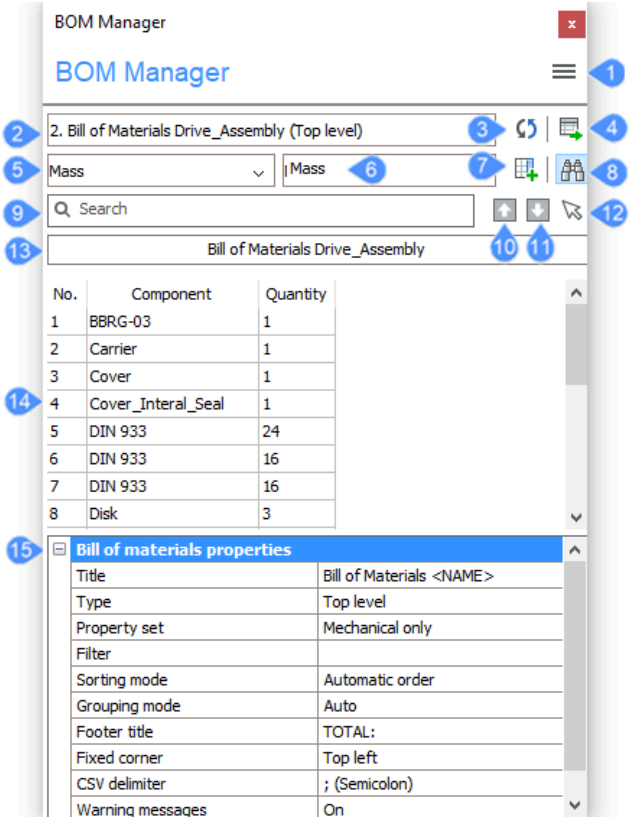
Top level
Creates a new Top level BOM.

Bottom level
Creates a new Bottom level BOM.

Hierarchical
Creates a new Hierarchical BOM.

Create from template
Opens a dialog box, which allows you to select a BOM Table template (*.BOM) file.

To start working with the BOM manager, you will have to create a new BOM table.



1 Menu

2 BOM Selector

3 Refresh

4 Place/update table

5 Category selector

6 Column selector

7 Insert column after

8 Find

9 Search

10 Find previous

11 Find next

12 Select all found

13 BOM title

14 BOM table

15 BOM and Column Properties

BOM Selector

Selects the current BOM table. The selection is stored in the drawing. When the drawing is closed and reopened, the selected BOM is restored in the manager. When selected, the table is updated in the current state of the drawing.

Refresh

Updates both the current BOM and the list of all BOM tables in the drawing.

Place/update table

Starts the interactive placement of a table in a drawing. If the table has already been placed in the drawing, it updates it. All edit operations automatically update the associated table in the drawing.

Category selector

Select the category of the properties that are displayed in the column selector. The category can be added to the current BOM pressing button no. 7.

Column selector

Select the column to add to the current BOM.

You can type here the formula for a formula field. To add a formula, type an expression starting with ”=”. To add a template expression type “$”, to refer columns, use <>. For example, to refer to the Mass column you will type: “=<Mass>”.

When you add a formula field and press button no. 7 the Formula column_1 dialog box will appear.

Insert column after

If a column is selected, it will insert the column from the Column selector after the selected column in the table.

If no columns are selected, the new column will be inserted after the last column in the table.

Find

Toggles the search field on and off. When you toggle on this button, buttons no. 9, 10, 11 and 12 will appear.

If you click on the BOM Manager panel, you can toggle the search field using **CTRL+F**.

Search

Type a string to search and press Enter; wildcards are supported.

Find previous

Selects the previous match on the BOM table.

You can also use **Shift+F3**.

Find next

Selects the next match in the BOM table.

You can also use **F3**.

Select all found

Selects all the items searched in the search field.

BOM title

Displays the title of BOM table. You can type the name you want instead of <NAME>. Press Enter or exit the field to apply the changes.

BOM table

Displays the content of the BOM table.

Right-click a column label offers additional tools:

- **Add template column from selected**: opens a template editor to configure a new template column. The selected columns will be automatically added to the template expression.
- **Column role**: opens a submenu.
  - **Number**: the column will be used as a source of numbers by the bmBalloon command.
  - **Name**: the column will be used as a source of names by the bmBalloon command.
  - **Quantity**: the column will be used as a source of quantities by the bmBalloon command.
  - **Regular**: the column will have no specific role.
- **Format**: opens the Format dialog box, which allows you to specify the format for the selected column(s).
- **Set column filter**: opens the Column Filter dialog box, which allows you to configure the column filter.
- **Fit column(s)**: fits the width of the selected column(s) to their content.
- **Fit all columns**: fits the width of all columns to their content.

Nota: The column width automatically fits the content unless the width has been changed manually.

Right-click on cells offers additional tools:

- **Zoom to**: zooms to the select part(s) in the model.
- **Select**: selects the part(s). Replaces an existing selection.
- **Add to Selection**: adds part(s) to the selection.
- **Remove from Selection**: removes the parts from the selection.
- **Remove highlighting**: removes highlighting.

BOM Properties

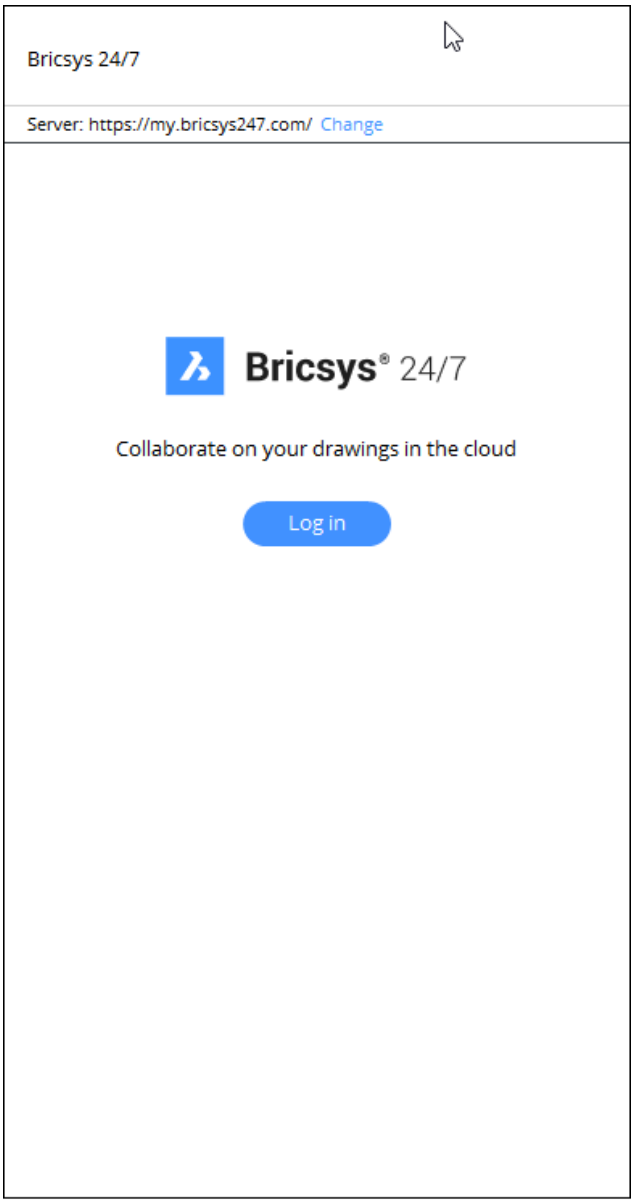
Displays the BOM and columns Properties. The columns properties will appear after you insert a column on the BOM table.

4.11.12 Bricsys 24/7 panel

The Bricsys 24/7 Panel helps team members to find and share their drawings.

Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

If you are logged out of your 24/7 account a screen will display:



1 Enter your Email address and Password and press the 'Log in' button.

2 (Option) Tick the**Remember Me**checkbox to store the username and password.

3 Once logged on, you will see an overview of your 'in the cloud' Projects, Folders, and Documents.

4.11.12.1 Project

The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.

4.11.12.2 Project Overview

At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.

4.11.12.3 Folder

When you open a project, a list of folders is shown.

4.11.12.4 Folder Overview

At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it.

4.11.12.5 Documents

In a folder you find documents. On the right-hand icons are displayed:

The profile picture or the initials from the author of the document.

The document is in the cloud.

Someone is currently working on the document.

The document is currently uploading.

The document is locked by someone, another person cannot change it.

Right-click on a folder to get a menu with 2 possible actions:

- Download files not locally available.
- Download out of date locally unmodified files.

Right-click on a document to get a menu with possible document actions

- Lock on server
- Unlock on server
- Upload local copy as a revision
- Download latest revision

4.11.13 Command line panel

The Command line panel allows you to launch commands or modify system variables by typing the command or variable name.

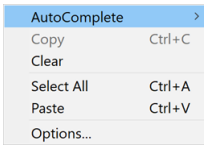
1 Command line

2 Command history

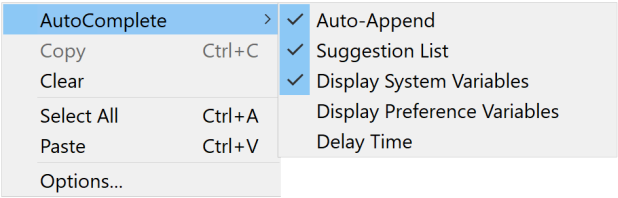
4.11.13.1 Command line

Displays the current command prompt. If no command is active, it prompts you to Enter Command. If a command is active, it displays prompts and options relevant to the current command. Prompt shortcuts are highlighted. You may enter the shortcut characters in the Command line or click on them with the left mouse button.

A right-click menu offers relevant tools.



AutoComplete
Hover over the AutoComplete option. Another context menu displays:



- If **Auto-Append** is checked, the entry is automatically completed if only one possible command is left.
- If **Suggestion List** is checked, the list of possible commands is displayed when typing in the command bar.
- If **Display System Variables** is checked, the system variables are included in the suggestion list.
- The **Delay Time** option sets the time in seconds before automated keyboard features display.

Copy
This option copies the selected text to the clipboard.

Clear
This option clears complete prompt history.

Select All
This option selects all the completed content of the prompt history.

Paste
This option pastes the text from the clipboard to the Command line.

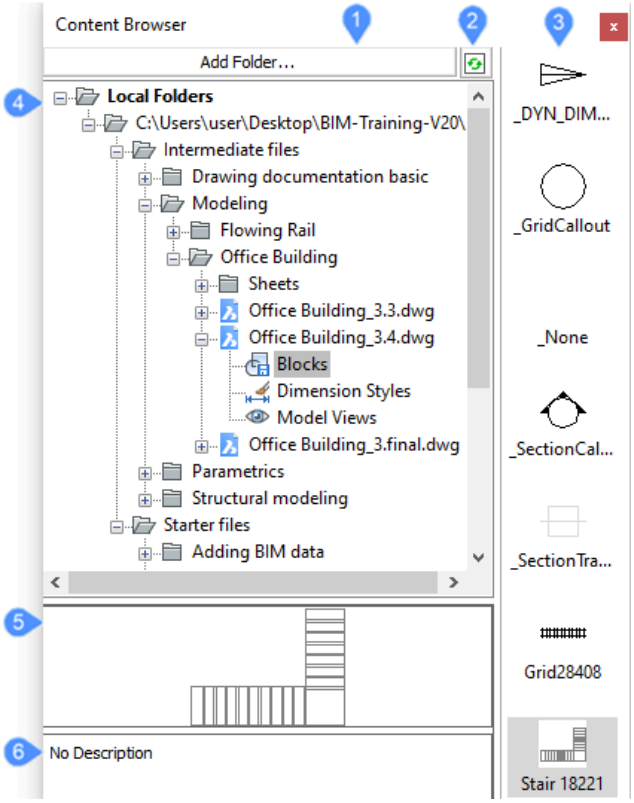
Options
This option displays the Settings dialog box at the Command line section.

4.11.13.2 Command History

Displays the history of the Command line input for the current BricsCAD session.

4.11.14 Content browser panel

The Content Browser panel offers you a quick and convenient way to open drawings, manage your block libraries, copy dimension styles, place views for your entire .dwg files stored on your computer.



1 Add Folder...

2 Refresh

3 Definitions in the selected drawing

4 Local Folders tree

5 Definitions' preview

6 Description

4.11.14.1 Add folder...

Displays the Choose a folder dialog box.

4.11.14.2 Refresh

Select a drawing and press the Refresh button to expand its content (Blocks, Dimension Styles, Model Views).

4.11.14.3 Definitions in the selected drawing

Select Blocks/Dimension styles/Model views to see the definitions used within the drawing.

You can drag & drop a block onto the current drawing and an Insert Block dialog box will appear.

If you drag & drop a dimension style to your drawing a Copy / Paste dialog box will appear.

4.11.14.4 Local Folders tree

Displays the content of a drawing/project. Right-click menus enable you to add or remove folders from within the Folders tree.

4.11.14.5 Definitions' preview

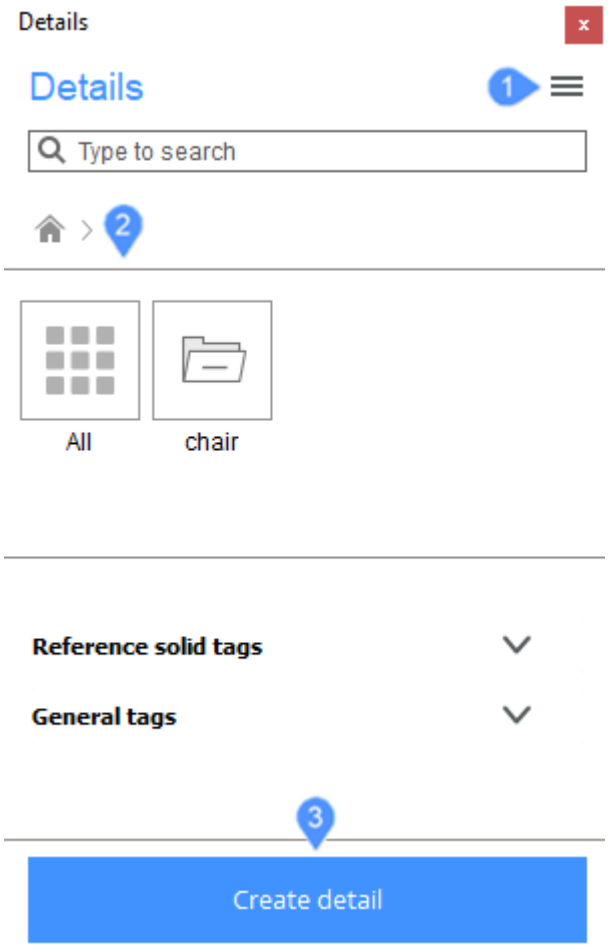
Shows a preview of your drawing/block/dimension style.

4.11.14.6 Description

Displays a description of your selected definition.

4.11.15 Details panel

The Details panel allows you to view details, add tags to details, propagate and adjust details.



1 Menu

2 Details library

3 Create detail

4.11.15.1 Menu

Grid View
Displays the details in a grid view.

List View
Displays the details in a list view.

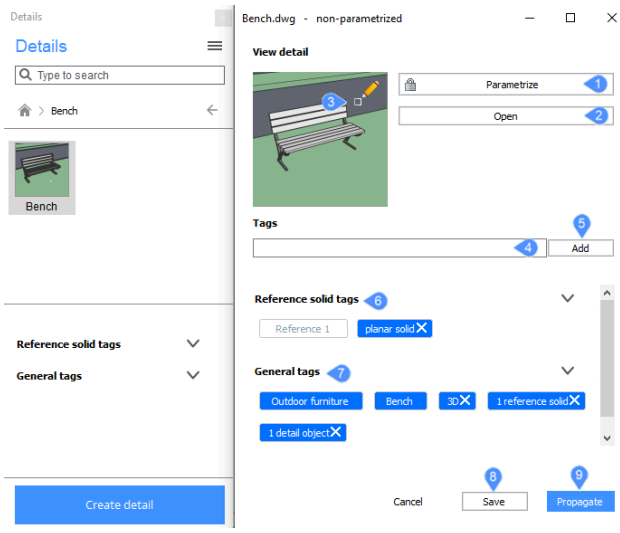
Generate thumbnails
Loads the new details in your library panel.

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

4.11.15.2 Details library

You can click on the category of the detail you want to view. The details assigned to that category will be displayed.

When you click on a detail, the below dialog box will appear.



1 Parametrize

2 Open

3 Edit detail

4 Tags

5 Add tag

6 Reference solid tags

7 General tags

8 Save

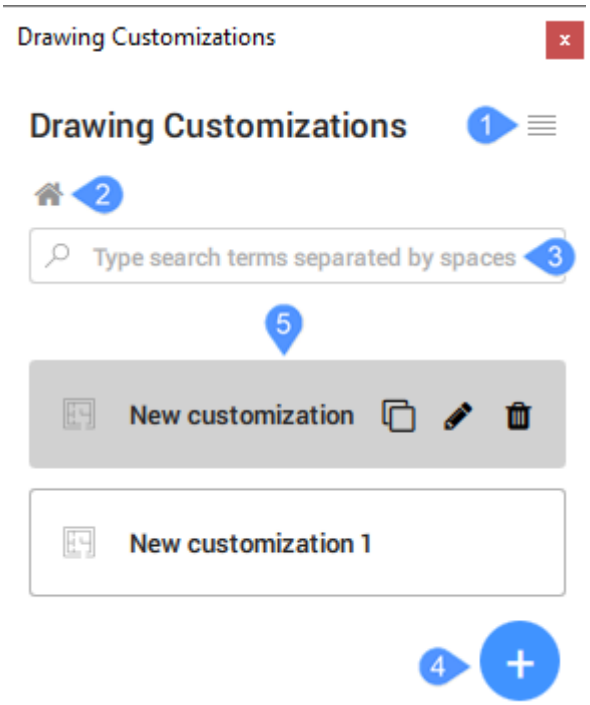
9 Propagate

4.11.15.3 Create detail

Opens the Create detail dialog box.

4.11.16 Drawing customizations panel

The Drawing Customizations panel allows you to create custom templates for drawings.



1 Menu

2 Home

3 Search

4 Add customization

5 Customizations List

4.11.16.1 Menu

Refresh
Refreshes the Drawing Customizations panel.

Manage libraries
Opens the Settings dialog box to change the section settings search path.

Load customizations
Allows you to choose a folder that contains customizations.

4.11.16.2 Home

Moves the cursor to the main user interface page of the panel.

4.11.16.3 Search

Searches all the drawing customizations that matches the words you enter in the search box.

4.11.16.4 Add customization

Creates a new template.

You can create a template by clicking on the button. By default, the template is named **New Customization** with the text highlighted. After you have added a relevant name for the template, press Enter to save and apply the changes. The inserted customizations will be sorted alphabetically automatically. Use the scroll button to navigate through the list.

4.11.16.5 Customizations List

Displays all the available drawing customizations.

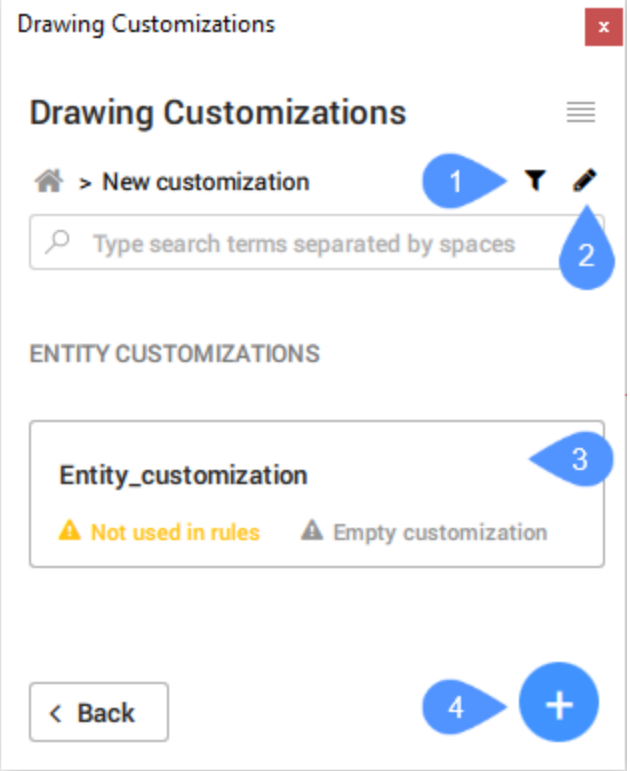
Duplicate
Creates a copy of your selected drawing customization.

Rename
Edits the name while the text is being highlighted. Press Enter to save and apply the changes.

Delete
Removes the drawing customization. The Removing a drawing customization dialog box appears.

Entity customizations
Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.
The **Entity customizations list** is empty for a newly created customization.

To create an entity customization left-click on the desired template, then click on the button. By default, the template is named **New Customization** with the text highlighted. Rename it while the text is being highlighted. Press Enter to save and apply the changes.
After creating your desired entity customization templates, click on the individual tabs to modify their respective settings.



1 To rule definitions

2 To style definitions

3 Entity customizations list

4 Add customization

To rule definitions

You can create a rule pressing the button. After creating the rule, the button offers you another 3 options:

- Entity customization
- Sub rule
- Filter

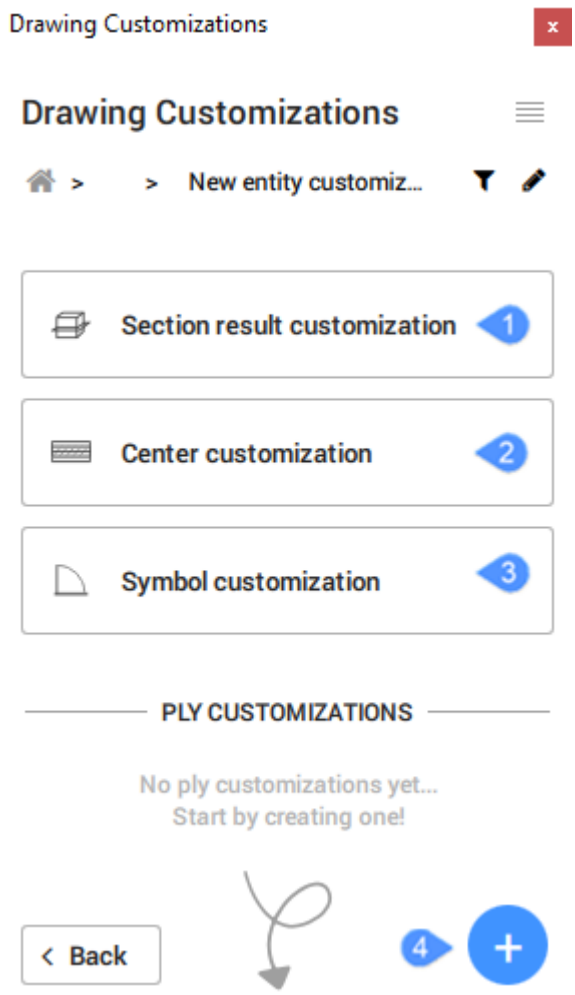
When creating an entity customization, you can add a sub rule for a rule already added.

To style definitions

To add a style definition, press the button.
When selecting a color, the Select Color dialog box will appear.
When selecting a line type and pressing Load…, the Load Linetypes dialog box will appear.

Entity customizations list
Displays the list of entity customizations.

Add customization



1 Section result customization

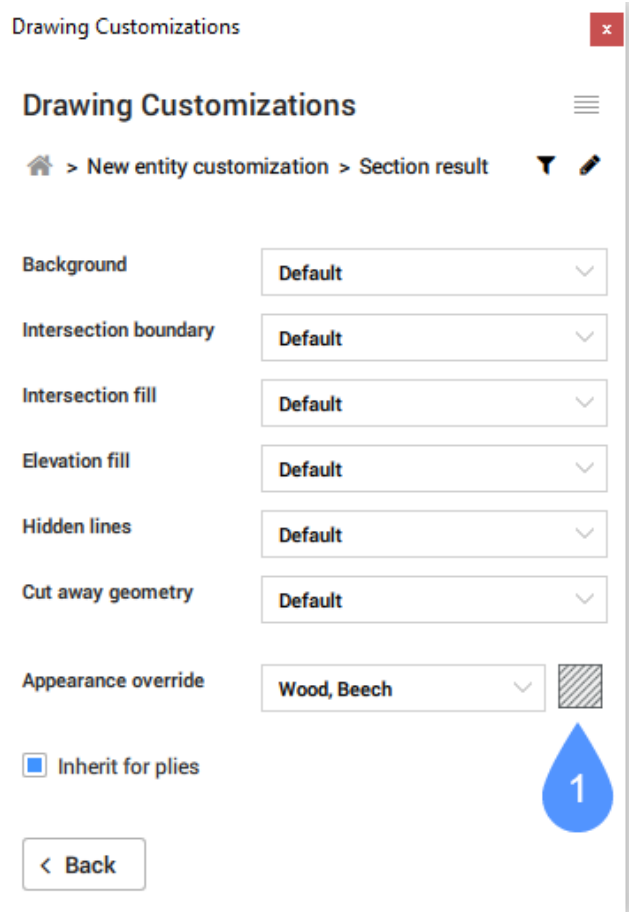
2 Center customization

3 Symbol customization

4 Add ply customization

Section result customization

Determines how the different pieces of geometry, generated by sectioning the entity with a section plane, are displayed.

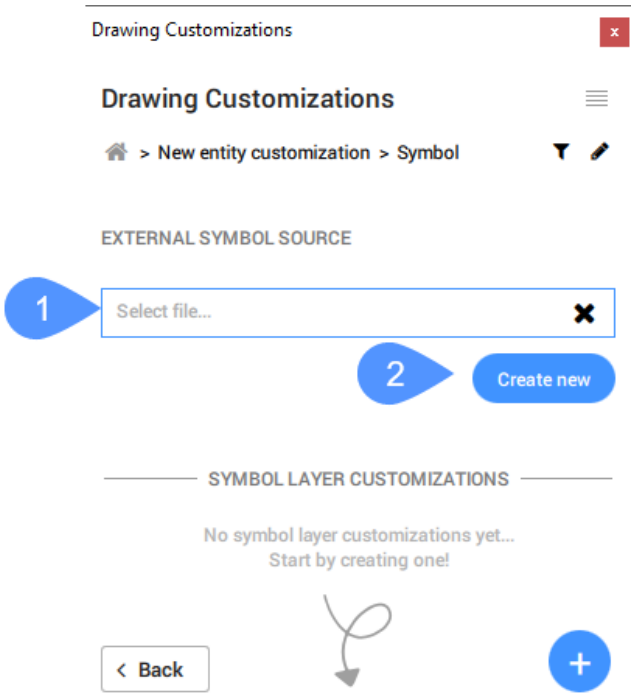


1 Physical Materials

Physical Materials
Opens the Physical Materials dialog box.

Center customization
Determines the appearance of the center of planar entities (center plane) or linear entities (center line).

Symbol customization
Determines the appearance of 2D symbols that are added for the entity and controls the source of those symbols.

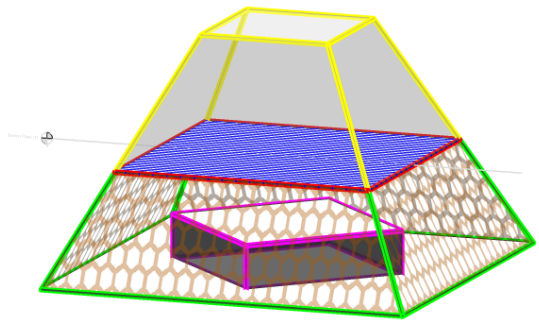


1 Select file…

2 Create new

Select file…
Opens the Set external symbol source dialog box.

Create new
Opens the New external symbol source dialog box.
The section result customization and center customization display a series of controls for which you can choose values from the drop-down list.



The diagram above illustrates how you could control the visibility and customization settings of an entity with the Drawing Customization tool. Notice that the section plane is aligned with the Blue square with Red outline.

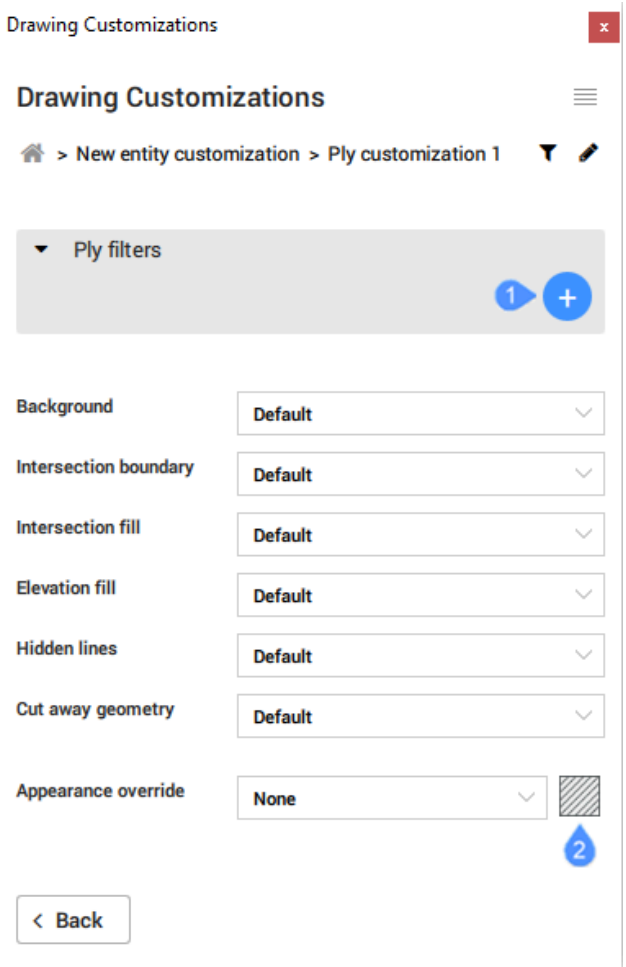
- Green outline represents **Background**, which is essentially the elevation outline of the cut object.
- Red outline represents **Intersection Boundary**, which traces over the cut area.

- Blue hatch represents **Intersection Fill**, as it shows the area of being cut by the section plane.
- Orange hexagons hatch represents **Elevation Fill**, which refers to the area bound by the elevation outline (see Background).
- Pink outline represents **Hidden Lines**, as it is below the blue hatch.
- Yellow outline represents **Cut away geometry**, as it is above the section plane.

Appearance Override allows you to specify how an entity's hatch appearances (section and elevation) would appear, in place of the default hatch patterns already defined within the Physical Materials library. When you create any new Physical Material entry in the "In Project" category as part of the Drawing Customization template, your desired configurations are independent of the drawing project and stored in the template instead. You can later apply this template to other project models as you wish.

Add ply customization

To add a ply customization, press the button.



1 Add default function

2 Physical Materials

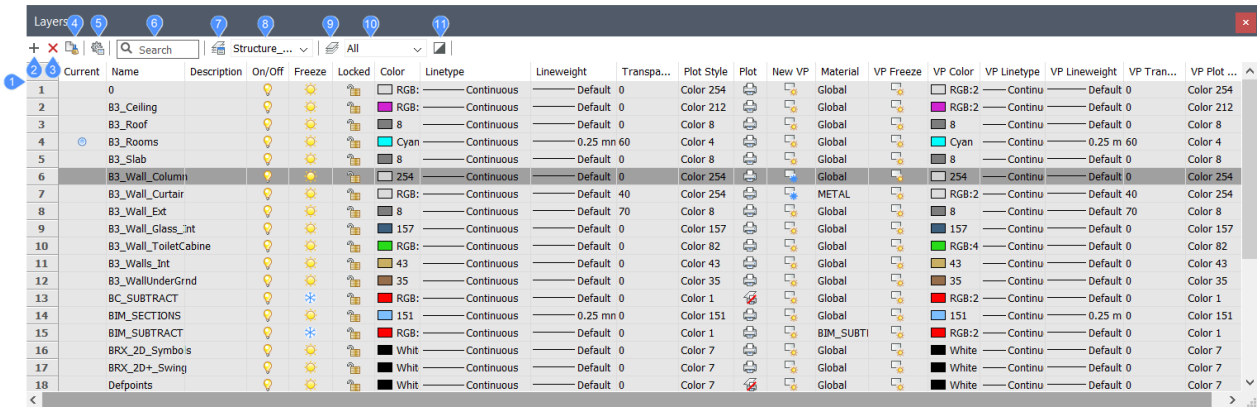
Add default function

Adds a default function for the ply customization.

Physical Materials
Opens the Physical Materials dialog box.

4.11.17 Layers panel

The Layers panel allows you to edit layer names and properties in the current drawing, as well as create, delete, edit, apply layers, layers states and layer filters.



1 Layer list

2 New layer

3 Delete

4 Purge

5 Settings

6 Search

7 Layer States Explorer

8 Layer State Control

9 Layers Explorer

10 Filter used

11 Invert filter

4.11.17.1 Layer list

Displays a list of layers and their properties. The list may include all layers in the current drawing or a subset of those layers based on selected layer states or filters. Column headers describe the properties of each layer. You may sort the layer list in order or reverse order for any of the properties by clicking once or twice on the column header.

A right-click menu on the column headers allows you to customize them. You can enable or disable the columns and restore the column positions.

A context menu displays when you right click menu on a layer name.

Merge to
Merges the selected layers to a target layer.

Nota: Merged layers will be purged from the drawing.

Remove From Group:
Removes the layer from group.

Select All:
Selects all available layers. You can also use the keyboard shortcut Ctrl+A.

Remove Viewport Overrides:
Removes overrides from properties that were set differently for viewports.

- **From Selected Layers**: Removes the overrides from the selected layers.
- **From All Layers**: Removes the overrides from all layers.
- **For Current Viewport**: Removes overrides from the current viewport.
- **For All Viewports**: Removes overrides from all viewports in the drawing

Isolate Selected Layers:
Isolates the selected layers. All other layers are either locked or turned off.

4.11.17.2 Current

Specifies the current working layer. Only one layer can be current at a time.

4.11.17.3 Name

Specifies the layer name. To rename a layer, double-click the layer name or select Rename from the right-click menu.

4.11.17.4 Description (optional)

Specifies the content or purpose of the layer. To create or edit a description, select one or more layers and then double-click the description field for one of the selected layers. The text you enter is applied to all selected layers.

4.11.17.5 On/Off

Specifies the On/Off status of the layers. Entities on layers are turned off are not visible in the drawing. However, they are processed during operations that require a regeneration of the drawing. Turning a layer on and off does not require the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for only a short time while you edit the drawing. To change the status, select one or more layers and then click a lightbulb icon for one of the selected layers. The status is applied to all selected layers.

To change the status, select one or more layers and then click a light bulb icon:

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

4.11.17.6 Freeze

Specifies the Thaw/Freeze status of the layers. Entities on frozen layers are not visible in the drawing and are not processed during operations that require a regeneration of the drawing. This can save processing time if the layer includes many entities. However, the process of freezing and thawing a layer requires the drawing to regenerate. Therefore, this toggle is most useful when you want entities on a layer to be hidden for an extended time while you edit the drawing. To change the status, select one or more layers, and then click a sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer.
- **Freeze**: Hides entities that are on that layer.

4.11.17.7 Locked

Specifies the lock status of the layers. Entities on locked layers cannot be edited. They remain visible; however, they are dimmed based on the LAYLOCKFADECTL System Variable. To change the status, select one or more layers, and then click a padlock icon for one of the selected layers. The status is applied to all selected layers.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

4.11.17.8 Color

Specifies the color of the layers. To change the color, select one or more layers and then click the current color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

4.11.17.9 Linetype

Specifies the linetype of the layers. To change the linetype, select one or more layers and then double-click the linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

4.11.17.1 Lineweight
0

Specifies the Lineweight of the layers. To change the lineweight, select one or more layers and then double-click the lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

4.11.17.1 Transparency
1

Specifies the transparency of the layers. To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0

(fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY System Variable must be turned on to experience transparency.

4.11.17.1 Plot style 2

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 3

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 4

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 5

Specifies the material of the layers. To change the material, select one or more layers and then double-click the material for one of the selected layers. The material you select is applied to all selected layouts. If the material you want is not displayed in the drop-down list, you can add it to the current drawing using the MATERIALS command.

4.11.17.1 VP Freeze

6

Specifies the Thaw/Freeze status of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space Thaw/Freeze status. To change the status, select one or more layers and then click the sun/snowflake icon for one of the selected layers. The status is applied to all selected layers.

- **Thaw**: Displays entities that are on that layer in the current layout or viewport.
- **Freeze**: entities that are on that layer in the current layout or viewport.

4.11.17.1 VP Color

7

Specifies the color of layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space color.

To change the color, select one or more layers and then click the current vp color for one of the selected layers. The Select Color dialog box is displayed. The color you select is applied to all selected layers.

4.11.17.1 VP Linetype

8

Specifies the linetype of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space linetype.

To change the linetype, select one or more layers and then double-click the vp linetype for one of the selected layers. If the linetype you want is not displayed in the drop-down list, choose Load to display the Load Linetypes dialog box. The linetype you select is applied to all selected layers.

4.11.17.1 VP Lineweight

9

Specifies the Lineweight of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space lineweight.

To change the lineweight, select one or more layers and then double-click the vp lineweight for one of the selected layers. The lineweight you select is applied to all selected layers.

4.11.17.2 VP Transparency

0

Specifies the transparency of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space transparency.

To change the amount of transparency, select one or more layers and then double-click the transparency for one of the selected layers. You can enter a value between 0 (fully opaque) and 90 (almost fully transparent) or select a predefined value from the drop-down list. The value you specify is applied to all selected layers.

The TRANSPARENCYDISPLAY system variable must be turned on to experience transparency.

4.11.17.2 VP Plotstyle

1

Specifies the plot style of the layers for the current layout tab or viewport. This property is not available from the Model tab and it overrides the model space plot style. If plot style mode (PSTYLEMODE system variable) for the current drawing is set to color dependent plot styles, the layers' plot styles use the layers' colors and are read-only.

If plot style mode for the current drawing is set to named plot styles, you can change the plot style. To specify a different named plot style, select one or more layers and then click the current plot style for one of the selected layers. The Select Plot Style dialog box is displayed. The plot style you select is applied to all selected layers.

4.11.17.2 New layer

2

Creates new layer with the generic name **NewLayer1**. The new layer that is created copies the properties of the currently selected layer.

4.11.17.2 Delete

3

Deletes layers, except for the layer with the name **0** and any layer with entities, which cannot be deleted.

4.11.17.2 Purge

4

Purges layers from the current drawing, except for the following ones, which cannot be purged:

- The layer 0.
- The current layer.
- Any layer with entities.

4.11.17.2 Settings

5

Displays the Layer Settings dialog box.

4.11.17.2 Layer States Explorer

6

Displays the Drawing Explorer dialog box with the Layer States category selected.

4.11.17.2 Layer State Control

7

Displays the current layer state if there are any defined in the drawing. Click the down-arrow button to choose a different layer state.

4.11.17.2 Layers Explorer

8

Displays the Drawing Explorer dialog box with the Layers category selected.

4.11.17.2 Filter used

9

Displays the current layer filter if there are any defined in the drawing. Click the down-arrow button to choose a different layer filter. Some filters are automatically created by BricsCAD®: All, All Used Layers.

4.11.17.3 Invert filter

0

Inverts the content of the current filter so that the dialog box displays all layers not included in the filter.

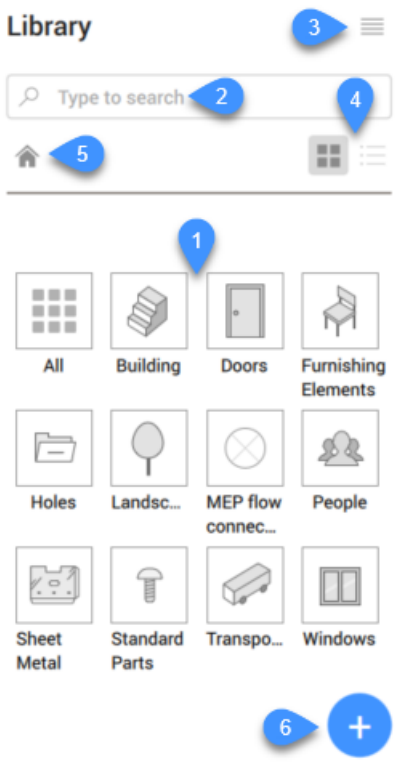
4.11.17.3 Search

1

Allows you to search for specific layer names. The list is shortened to display only the layers that contain the entered sequence of characters, regardless of where they are located in the layer name.

4.11.18 Library panel

The Library panel offers a central location to access 2D and 3D block libraries.



1 Block library

2 Search

3 Menu

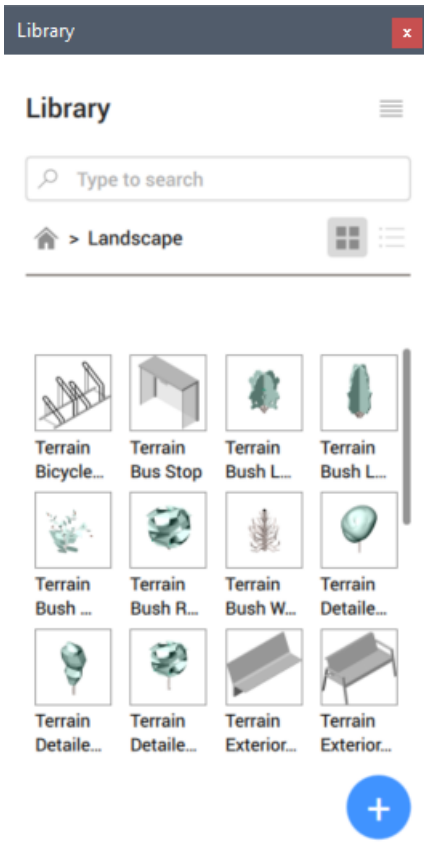
4 View

5 Home

6 Add

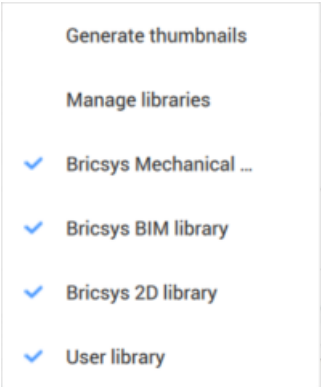
4.11.18.1 Block library

The block library is organized by categories. Select a category to view its blocks. Categories and blocks may be user-defined or pre-defined by BricsCAD®. You can scroll through the categories and blocks using the mouse wheel or the scroll bar. The availability of pre-defined blocks depends on your license level.



4.11.18.2 Menu

The Library panel menu enables you to control what is displayed in the block library.



Generate thumbnails
Generates or updates the thumbnail images of all blocks.

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

Bricsys BIM library
When checked, the panel displays BIM components. The default path for BIM components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\Bim\Components*..

Bricsys Mechanical library

When checked, the panel displays Mechanical components. The default path for Mechanical components is: *C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\DesignLibrary*.

User library

When checked, the panel displays the user defined components. The path for user library is defined by the COMPONENTSPATH system variable.

4.11.18.3 Search

Searches the library for words you enter in the search box.

4.11.18.4 View

Toggles between grid and list views of the block content.

4.11.18.5 Home

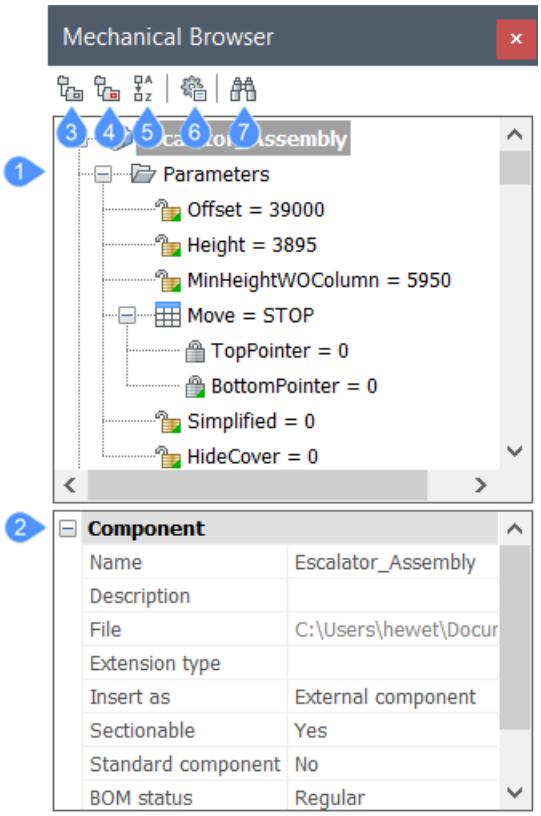
Returns the block library to the home screen.

4.11.18.6 Add

Opens the Add Block to Library dialog box.

4.11.19 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

4.11.19.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

Update
Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.

Visual style > All by Viewport
Applies the current visual style to all components in the assembly.

Insert standard hardware
Inserts a standard hardware part as a mechanical component in the current drawing.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create exploded view
Creates a block with an exploded representation of the current.

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Delete
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

Disable
The feature will remain in the Mechanical Browser, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SMCONVERT.

Right-clickmenusoffer additional toolsfor components:

Open
Opens the referenced drawing.

Open a copy
Opens a copy of a component insert as a new drawing.

Update
Reloads all referenced components from external files and updates BOM tables.

Replace…
Replaces a component insert.

Nota: Replacing a local insert turns it into an external insert.

Set material to component
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

BOM status
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Link to parameter
Links subcomponent parameter to the main level parameter.

4.11.19.2 Properties

Specifies the properties of the selected item.

4.11.19.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

4.11.19.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

4.11.19.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

4.11.19.6 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

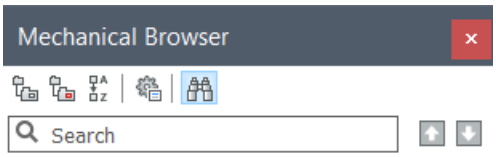
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

Sub-components of standard parts
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

4.11.19.7 Show search

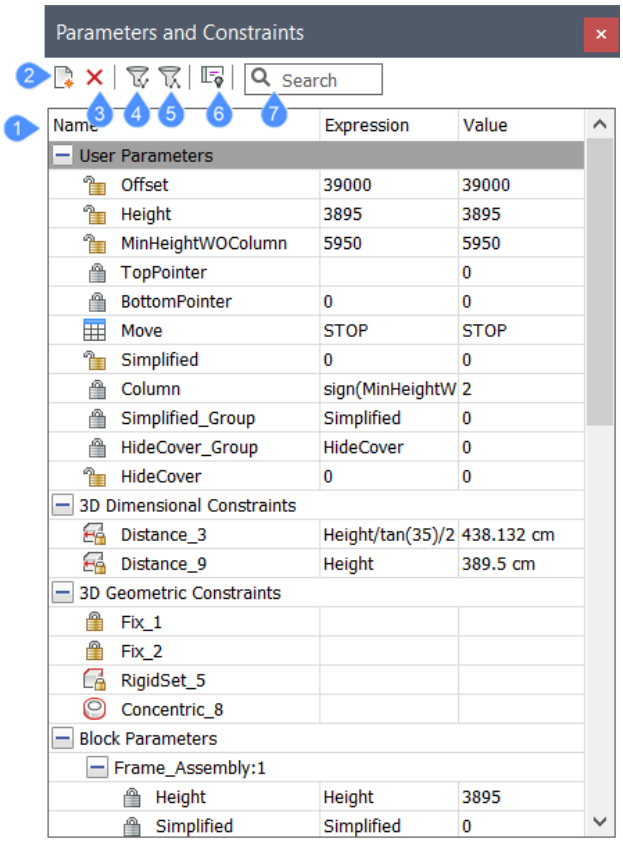
Toggles the search field and associated tools on and off. You can also press CTRL+F to toggle Search.

Search field
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.



4.11.20 Parameters and constraints panel

The Parameters and Constraints panel offers a central location to view and modify parameters and constraints in the current drawing.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

4.11.20.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters, 2D dimensional constraints, 3D dimensional constraints,

3D geometric constraints, block parameters and array parameters. Right-click menus offer additional tools for some parameters and constraints:

Units
Sets the dimension of the parameter. You can choose between: none, linear, square and cubic.

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates parameter value within some range.

Exposed
Defines whether the parameter is visible and can be modified when the component is inserted in an assembly

Clean unused variables
Erases the unused variables.

Link to parameter
Links subcomponent parameter to the main level parameter.

4.11.20.2 Name

Specifies the name of the parameter.

4.11.20.3 Expression

Specifies the mathematical expression of the parameter.

4.11.20.4 Value

Specifies the value resulting from the parameter's expression.

4.11.20.5 New user parameter

Inserts a new user parameter with default values for the Expression and Value.

4.11.20.6 Delete a user parameter

Removes selected parameter and constraints from the drawing.

4.11.20.7 Show dependent parameters

Toggles the display of dependent parameters on or off.

4.11.20.8 Show defining parameters

Toggles the display of defining parameters on or off.

4.11.20.9 Hide geometrical constraints

Toggles the display of geometrical constraints on or off.

4.11.20.1 Search

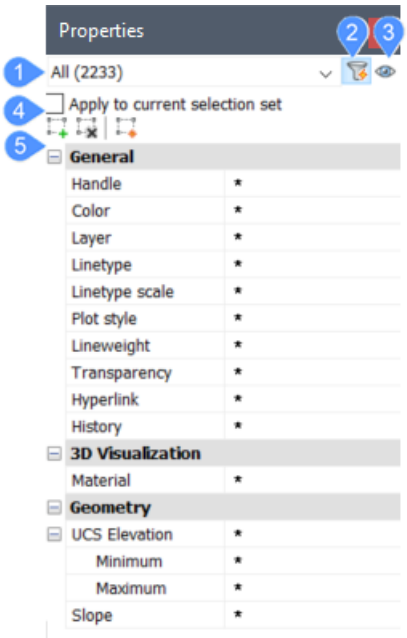
0

Filters the parameter list to display only the parameter names that match the character string you enter.

4.11.21 Properties panel

TheProperties panel offers a central location to access drawing, viewport, and entity properties.

It can be used in either Properties mode or Quick Select mode.



1 Entities list

2 Properties/Quick Select toggle

3 Property Preview toggle

4 Selection tools (Quick Select mode only)

5 Properties list

Entities list

Displays the type and number of selected entities.

Properties/Quick Select toggle

Switches between Properties and Quick Select mode.

Property Preview toggle

Turns the PROPERTYPREVIEW setting on or off.

Selection tools

Creates a selection set based on properties. These tools are only available in Quick Select mode.

Properties list

Displays relevant property categories based on the current mode and selection set. You can expand and collapse categories by clicking on the + and – icons.

4.11.21.1 Properties mode

In Properties mode, the Properties panel enables you to view and modify drawing, viewport, and entity properties. Drawing and viewport properties are accessible when the current selection set is empty. Entity properties are accessible when the current selection set has at least one entity.

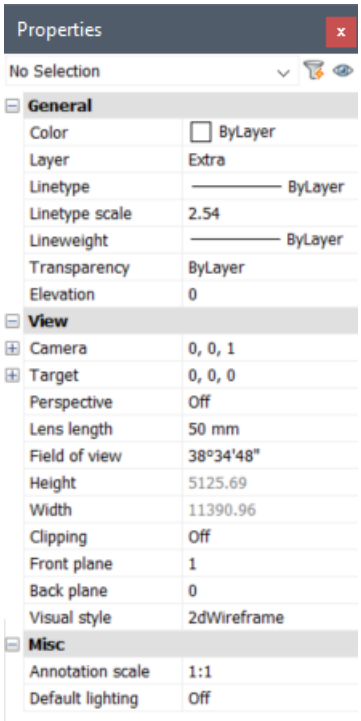
Entities list

When no entities are selected in Properties mode, the entities list shows No Selection and the properties list displays drawing and viewport properties.

When entities are selected in Properties mode, the entities list describes the type and number of entities in the selection set. You can filter the selection set by choosing a type of entity from the drop-down list. The types of entities in the selection set determine which properties are accessible.

Properties list (no selection set)

The drawing and viewport properties list, available when no entities are selected, displays General, View, and Misc properties.



General Properties

In Properties mode, when no entities are selected, General properties describe the current entity settings. View and modify current entity settings to specify the properties that apply to new entities as they are created. Changing the current entity settings in the Properties list has the same effect as changing the relevant system variables.

Color

Describes the current color (CECOLOR system variable). You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer

Describes the current layer (CLAYER system variable). You can specify a new layer from the drop-down list.

Linetype
Describes the current linetype (CELTYPE system variable). You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Describes the current linetype scale (CELTSCALE system variable). You can enter a new linetype scale.

Lineweight
Describes the current lineweight (CELWEIGHT system variable). You can specify a new lineweight from the drop-down list.

Transparency
Describes the current transparency level (CETRANSPARENCY system variable). You can specify a new transparency level from the drop-down list or enter a value.

Elevation
Describes the elevation relevant to the current UCS (ELEVATION system variable). You can enter a new elevation.

View Properties
When no entities are selected, View properties describe the current view and relevant view settings.

Camera
Describes the absolute coordinates of the camera in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Target
Describes the absolute coordinates of the camera target in the current UCS, based on the cartesian coordinate system (X,Y,Z). These values automatically update when you orbit or change the viewpoint in other ways.

Perspective
Describes the current perspective setting (PERSPECTIVE system variable). You can turn Perspective on or off from the drop-down list.

Lens Length
Describes the current lens length (LENSLENGTH system variable). You can enter a new lens length.

Field of View
Describes the current angle for the field of view. You can enter a new angle for the field of view.

Height (read only)
Describes the height of the current view.

Width (read only)
Describes the width of the current view.

Clipping
Describes the clipping mode for the current view. You can specify a new clipping mode from the drop-down list. When clipping is turned on, parts of the view before the front clipping plane and/or behind the back clipping plane are hidden. This property is read only for layouts.

Front Plane
Describes the location of the front clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

Back Plane

Describes the location of the back clipping plane. This property is only available when the Clipping property is turned on. This property is read only for layouts.

Visual Style

Describes the current visual style of the drawing. You can specify a new visual style from the drop-down list. This property is read only for layouts.

Misc Properties

When no entities are selected, Misc properties describe the current annotation scale and default lighting status.

Annotation Scale

Describes the current annotation scale (CANNOSCALE system variable). You can specify a new annotation scale from the drop-down list.

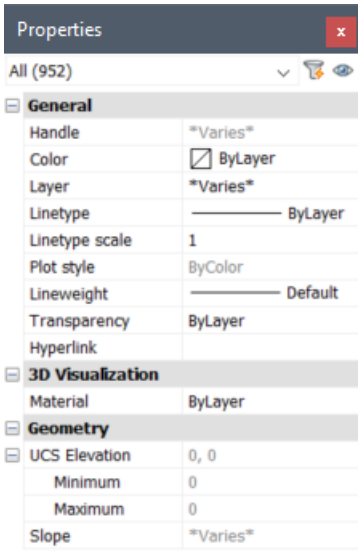
Default Lighting

Describes the default lighting status (DEFAULTLIGHTING system variable). You can turn Default Lighting on or off from the drop-down list.

Properties list (current selection set)

The Properties list only displays properties that are valid for all the selected or filtered entities. The more types of entities you select, the fewer properties are available on the Properties list. Filtering the entity list enables you to view and modify properties for multiple entities of the same type with minimal effort. When two or more entities are selected, the properties show their common values. Properties that differ between the selected entities are shown as *varies*.

All entities display General, 3D Visualization (Model space only), and Geometry properties. Additional properties may display for certain types of entities.



General Properties

In Properties mode, when entities are selected, General properties describe properties that are applied to the selected entities and are valid for almost any type of entity.

Handle (read only)

Describes the unique hexadecimal value by which the selected entity is identified in the DWG database.

Color
Describes the color of the selected entity. You can specify a new color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer
Describes the layer of the selected entity. You can specify a new layer from the drop-down list.

Linetype
Describes the linetype of the selected entity. You can specify a new linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Describes the linetype scale of the selected entity. You can enter a new linetype scale.

Plot Style
Describes the plot style of the selected entity. You can specify a new plotstle from the drop-down list. Note that this property is only available when plot styles are enabled. Also see the PlotStyle command

Lineweight
Describes the lineweight of the selected entity. You can specify a new lineweight from the drop-down list.

Transparency
Describes the transparency level of the selected entity. You can specify a new transparency level from the drop-down list or enter a value.

Hyperlink
Describes the URL of the selected entity. You can enter a new URL or choose the Browse button to access the Edit Hyperlink dialog box.

History
Enables you to undo editing operations for a selected entity. You can specify a previous step from the drop-down list to undo the entity back to one of the previous steps without undoing all the other commands and view operations that have been executed since. The History property only displays when the selection set includes only one entity.

Thickness
Describes the thickness of the selected entity. You can specify a new thickness by entering a value or selecting the picker icon and picking two points in the drawing.

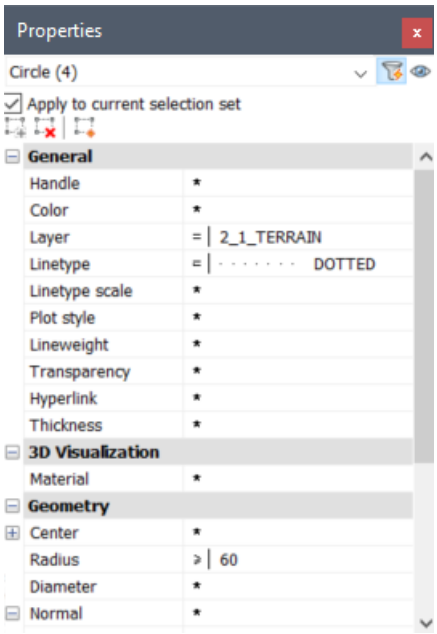
3D Visualization Properties
3D Visualization properties describe the materials assigned to the selected entities.

Material
Describes the current material of the selected entities. You can specify a new material from the drop-list.

Geometry Properties
Geometry properties describe the geometric characteristics of the selected entities. These properties can vary for different types of entities. See Drawing Entities for more information.

4.11.21.2 Quick Select mode

In Quick Select mode, the Properties panel enables you to create a selection set based on entity properties. It includes selection tools and a operators.



Selection tools

Create a selection set based on entity properties.

Apply to current selection set

This option is only valid if there's an existing selection set. When enabled, the Entities list is limited to the entities in the selection set. When disabled, the Entities list includes all entities in the current viewport or layout.

Add to current selection set

Adds entities to the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is enabled since all possible entities are already included in the current selection set.

Remove from current selection set

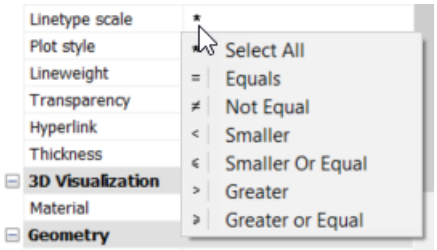
Removes entities from the current selection set based on the entity types and property values you specify in the Entities and Properties lists. This option is unavailable when the Apply to current selection set is disabled since entities can only be removed from an existing selection set.

Add to new selection set

Creates a new selection set based on the entity types and property values you specify in the Entities and Properties lists.

Operators

A list of operators enables you to filter properties based on values. You can click on the current operator to display the list and change it to a different operator.



Select All
Does not filter the entity list based on the property.

Equals
Filters the entity list to include only those entities whose property equals the specified value.

Not Equal
Filters the entity list to include only those entities whose property does not equal the specified value.

Smaller
Filters the entity list to include only those entities whose property is less than the specified value.

Smaller Or Equal
Filters the entity list to include only those entities whose property is less than or equal to the specified value.

Greater
Filters the entity list to include only those entities whose property is greater than the specified value.

Greater or Equal
Filters the entity list to include only those entities whose property is greater than or equal to the specified value.

Properties list
The properties list displays entity properties on which you can filter. The available properties depend on the types of entities in the drawing and whether the option to Apply to current selection set is enabled. General, 3D Visualization (Model space only), and Geometry properties are available for all entities. Additional properties may display for certain types of entities.

General Properties
In Quick Select mode, General properties enable you to filter drawing entities based on their general properties.

Handle
Enables you to filter entities based on the unique hexadecimal value by which the entity is identified in the DWG database.

Color
Enables you to filter entities based on the color of the entity. You can specify a color from the drop-down list or choose Select color to access additional colors from the Select Color dialog box.

Layer
Enables you to filter entities based on the layer of the entity. You can specify a layer from the drop-down list.

Linetype
Enables you to filter entities based on the linetype of the entity. You can specify a linetype from the drop-down list or choose Load to access additional linetypes from the Load Linetypes dialog box.

Linetype Scale
Enables you to filter entities based on the linetype scale of the entity. You can enter a linetype scale.

Plot Style
Enables you to filter entities based on the plot style of the entity. You can specify a plotstle from the drop-down list.

Nota: This property is only available when plot styles are enabled.

Lineweight

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

Transparency

Enables you to filter entities based on the transparency level of the entity. You can specify a transparency level from the drop-down list or enter a value.

Hyperlink

Enables you to filter entities based on the URL of the entity. You can enter a URL or choose the Browse button to access the Edit Hyperlink dialog box.

Thickness

Enables you to filter entities based on the thickness of the entity. You can specify a thickness by entering a value or selecting the picker icon and picking two points in the drawing.

3D Visualization Properties

In Quick Select mode, 3D Visualization properties enable you to filter drawing entities based on their 3D visualization properties.

Material

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

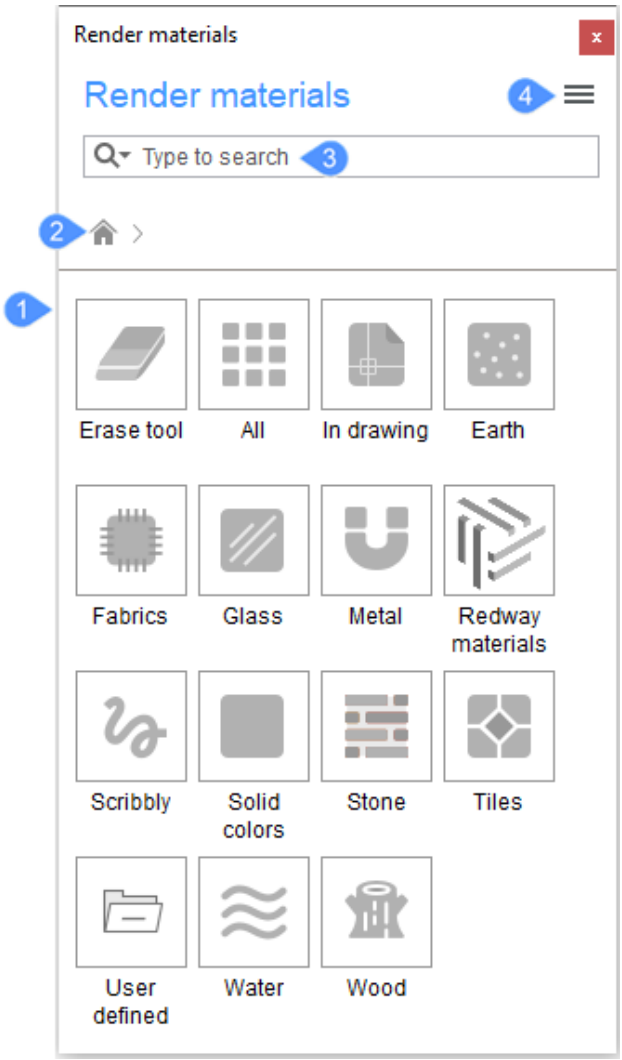
Geometry Properties

In Quick Select mode, Geometry properties enable you to filter drawing entities based on their geometric properties. These properties can vary for different types of entities.

4.11.22 Render materials panel

The Render Materials panel offers access to materials used for renderings.

Material definitions are saved in the drawing. Each drawing contains the Global material. It is not possible to delete or rename the Global material, however you can edit the properties of the Global material.



1 Material library

2 Home

3 Search

4 Menu

4.11.22.1 Material library

The material library is organized by categories. You can select a material from a category and then apply it to an entity using the brush that appears. You can scroll through the materials using the mouse wheel or the scroll bar.

Nota: Right-click on a material to add it to library, to remove it from drawing or to add it to drawing.

4.11.22.2 Home

Returns the material library to the home screen.

4.11.22.3 Search

Searches the library for words you enter in the search box.

4.11.22.4 Menu

Grid View
Displays the materials in a grid view.

List View
Displays the materials in a list view.

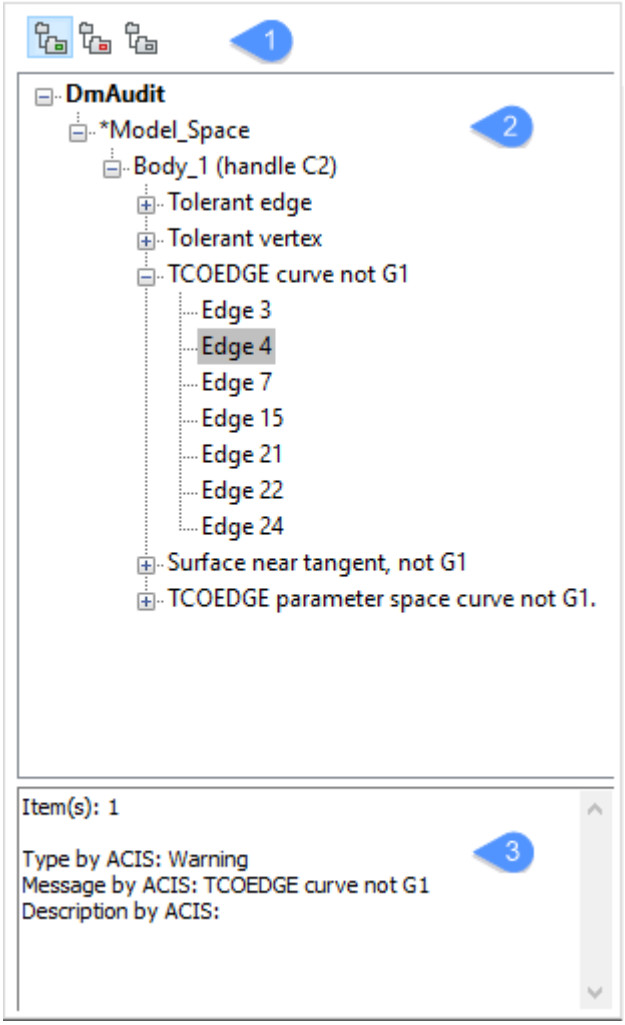
Manage libraries
Opens the Settings dialog box to change the render materials directory path.

Get more materials
Opens the Render Materials documentation in a browser, where you can download the complete Render Material Library.

4.11.23 Report panel

The Report panel offers feedback for relevant commands including DMAUDIT, DMSTITCH, and all sheet metal (SM*) commands.

The appearance of the Report panel is determined by the REPORTPANELMODE system variable.



1 Display grouping

2 Report output

3 Detailed items output

4.11.23.1 Display grouping

Specifies how information is grouped for the report.

- Sort by Structure/Message/Subentity
- Group by Message/Structure/Subentity
- Group by Structure/Subentity/Message

4.11.23.2 Report output

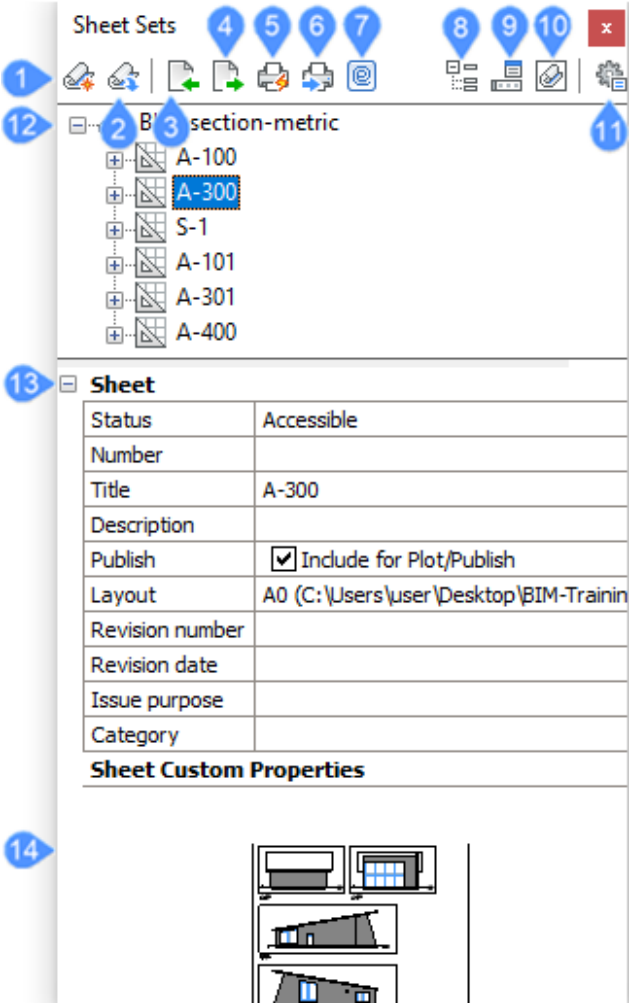
Displays command output.

4.11.23.3 Detailed items output

Shows detailed command output for selected item in the report output.

4.11.24 Sheet sets panel

The Sheet Sets panel offers a central location to create and manage sheet sets.



1 New sheet set...

2 Open sheet set...

3 Import from XML...

4 Export to XML...

5 Plot

6 Publish...

7 eTransmit...

8 Sheet selections sets...

9 Custom properties...

10 View categories

11 Sheet set options...

12 Sheets

13 Sheet properties

14 Sheet preview

4.11.24.1 New sheet set...

Opens the Create New Sheet Set dialog box.

4.11.24.2 Open sheet set...

Opens the Select a Sheet Set File dialog box.

4.11.24.3 Import from XML...

Allows you to import an XML file that was created in BricsCAD® with **Export to XML...**. Opens the Import Sheet Set dialog box.

4.11.24.4 Export to XML...

Exports a sheet set in XML format. Opens the Export sheet set as dialog box.

4.11.24.5 Plot

Plots the currently selected sheet set(s), subset(s) or sheet(s).

4.11.24.6 Publish...

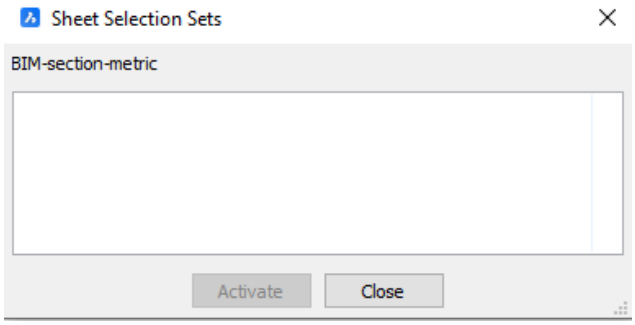
Publishes the currently selected sheet set(s), subset(s) or sheet(s). Opens the Publish dialog box.

4.11.24.7 eTransmit...

Launches the eTransmit command. The currently selected sheet set(s), subset(s) or sheet(s) are included in the eTransmit procedure. Opens the eTransmit dialog box.

4.11.24.8 Sheet selections sets...

Opens the Sheet selection sets dialog box.



4.11.24.9 Custom properties...

Allows you to add custom properties for sheet sets and sheets. Opens the Custom properties dialog box.

4.11.24.1 View categories

0

Expands and collapses the currently loaded sheet sets.

4.11.24.1 Sheet set options...

1

Opens the Sheet Sets options in the Settings dialog box.

4.11.24.1 Sheets

2

Displays the opened and created sheet sets.

Right-clickmenusoffer additional tools:

Renumber

Opens the Renumber Sheet Set dialog box, which allows you to renumber all sheets in the sheet set.

New Sheet

Displays the New Sheet(s) dialog box, which allows to add new sheets to the current sheet set.

New Subset

Adds a new subset to the current sheet set.

Insert Sheet List Table

Inserts the sheet list table into the current drawing.

Open

Opens the drawing with the layout of the selected sheet.

Open read-only

Opens the drawing with the layout of the selected sheet in read-only mode to protect it

Nota: You can double-click on a sheet set or a subset to expand/collapse. Double-click on a sheet to open it.

4.11.24.1 Sheet properties

3

Displays the properties and variables of the selected sheet set, subset or sheet.

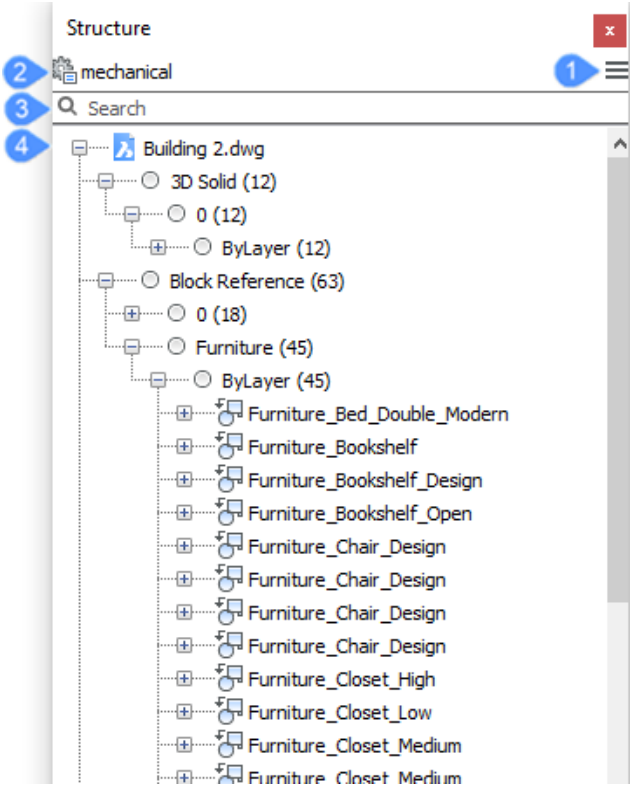
4.11.24.1 Sheet preview

4

Displays a thumbnail of the currently selected sheet set without opening it.

4.11.25 Structure panel

The Structure panel enables you to view and select drawing content from a structured tree.



1 Menu

2 Configure Structure Tree

3 Search

4 Content tree

4.11.25.1 Menu

Displays the Structure panel main menu.

bim
Displays a BIM entity-based structure.

default
Displays a layer-based structure.

mechanical
Displays an entity-based structure.

Configure structure tree
Opens the Configure Structure Tree dialog box, which allows you to configure the structure tree.

Select Configuration…
Allows you to load Structure Tree Configuration Files (*.cst).

4.11.25.2 Configure Structure Tree

Displays the Configure structure tree dialog box.

4.11.25.3 Search

Searches the structure tree for character strings that match what you enter.

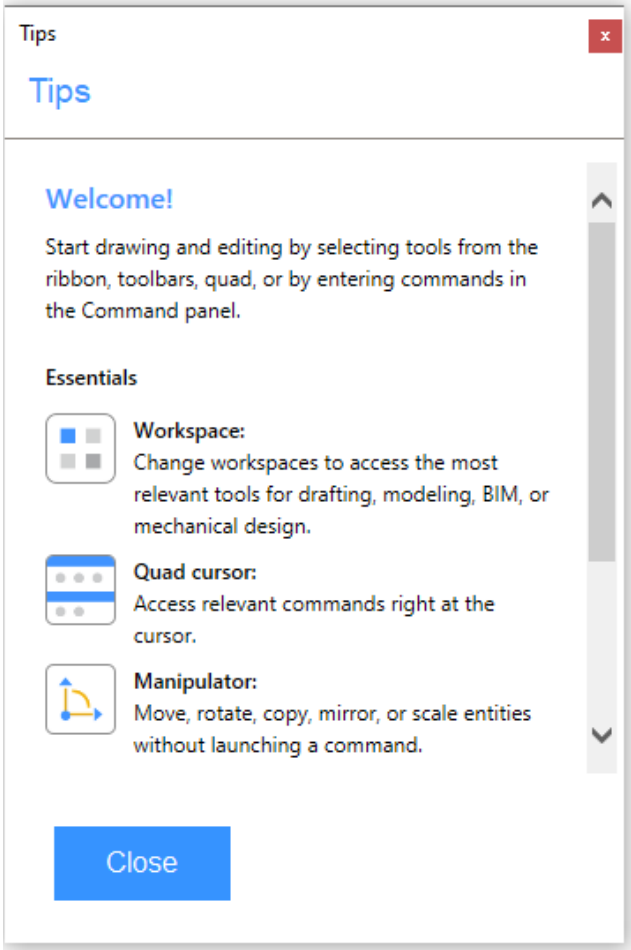
4.11.25.4 Content tree

Displays a structured tree view of the drawing content. The configuration of the structure tree is defined in a .CST file and specified by the STRUCTURETREECONFIG system variable. The location of CST files is specified by the SRCHPATH system variable. You can create custom CST files to meet your needs.

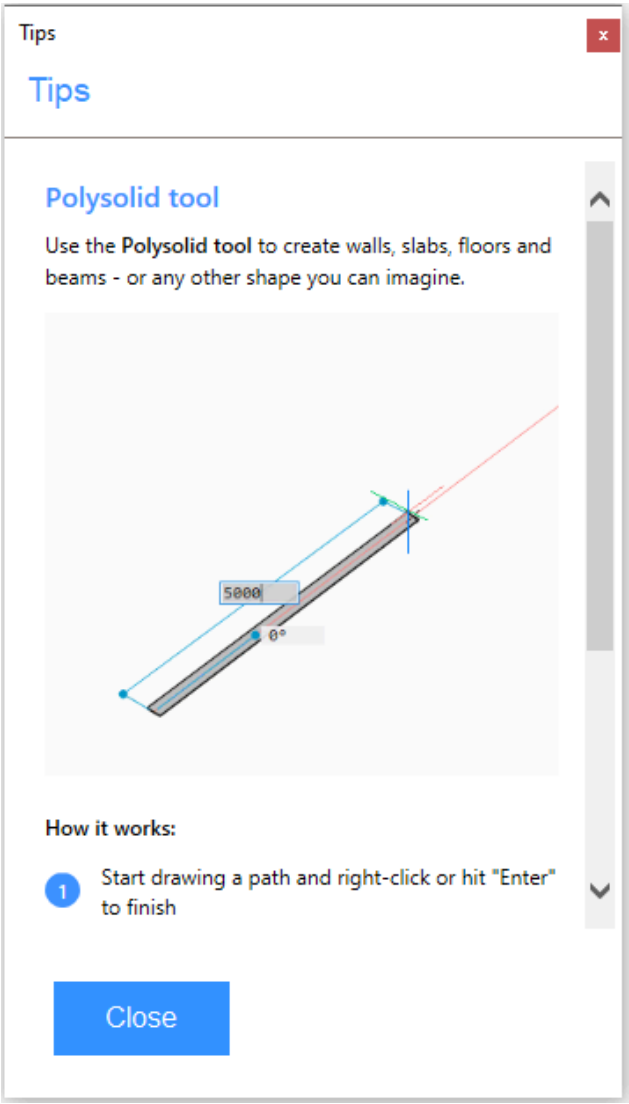
A right-click menu on a tree node or on an entity offers relevant tools.

4.11.26 Tips panel

The Tips panel offers relevant advice based on the context in which you are working.



When you start certain commands, the panel displays a short animation and additional information about the command.



4.11.27 Tool palettes panel

The Tool palettes panel offers a central location to access blocks, hatches, and command tools.

It includes default tabs with sample tools. You can easily create and customize your own tabs and tools using the right-click menu.

Right-clickmenusoffer additional tools:

Update tool image:
Updates tool image for hatches and blocks.

Specify Image…:
Allows you to specify another image for the tool.

> ***Nota:*** To restore the original tool image, right-click the tool and choose **Remove Image** in the context menu.

Properties…:
Opens the Tool Properties dialog box, which allows you to edit a tool.

View Options…:
Opens the View options dialog box, which allows you to edit the display of the tool icons.

Add Text:
Adds a text separator.

Add Separator:
Adds a line separator.

Customize Palettes…:
Opens the Customize dialog box, which allows you to customize the palettes. You can import and export palettes or create, import and export palettes groups.

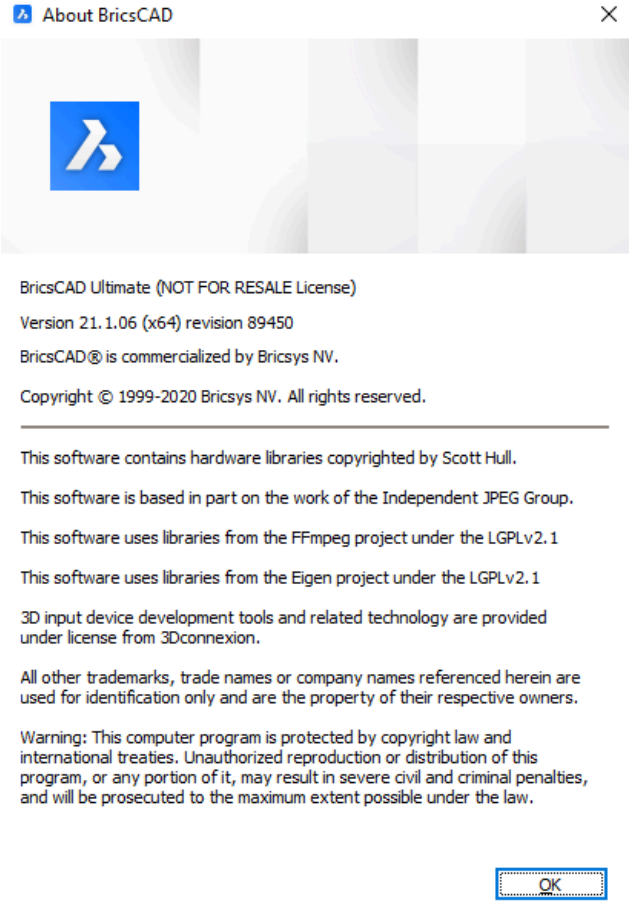
Add Tool…:
Opens the Customize dialog box, which allows you to add a new tool.

4.12 Dialog boxes

4.12.1 About BricsCAD dialog box

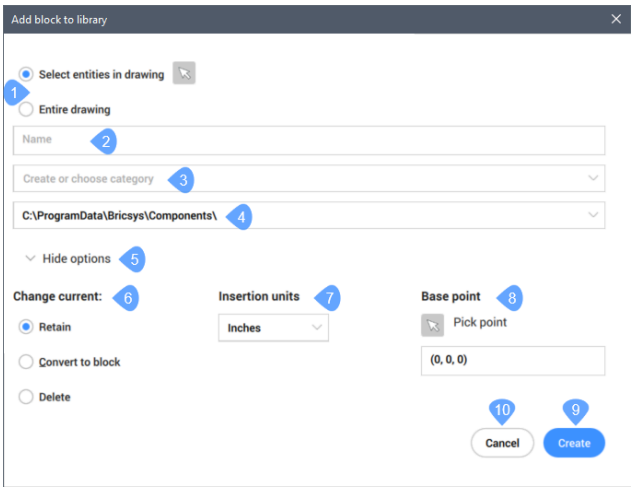
The About BricsCAD dialog box displays copyright and product information.

The product information includes the license type, version and revision number.



4.12.2 Add block to library dialog box

The Add block to library dialog box creates a block definition and adds it to the Library.



1 Selection set

2 Name

3 Category

4 Location

5 Options

6 Change current

7 Insertion units

8 Base point

9 Create

10 Cancel

4.12.2.1 Selection set

Specifies which entities are included in the block definition. You can select entities in the drawing or use the entire drawing.

4.12.2.2 Name

Specifies the name of the block.

4.12.2.3 Category

Specifies a category for the block. You can choose from a list of existing categories or enter a new category name.

4.12.2.4 Location

Specifies the location where the block dwg will be saved. Available locations are controlled by the Library directory path setting (COMPONENTSPATH system variable).

4.12.2.5 Show/Hide Options

Shows or hides additional options. The available options depend on the selection set.

4.12.2.6 Change current

Species what happens to selected entities after the block is created. You can choose to retain the individual entities, convert them to a block insertion using the new definition, or delete them. This option is only available if you define the selection set by choosing to Select entities in drawing.

4.12.2.7 Insertion units

Specifies the units for the block insertion. You can choose from the following list of standard units:

- Inches
- Feet
- Miles
- Millimeters
- Centimeters
- Meters
- Kilometers
- Microinches
- Mils
- Yards
- Angstroms
- Microns
- Decimeters
- Dekameters
- Hectometers
- Gigameters
- Astronomical Units
- Light-years
- Parsecs
- US Survey Feet
- Us Survey Inch
- US Survey Yard
- US Survey Mile
- Base point

Specifies the base point for block definition. You can pick a point in the drawing or enter X,Y,Z coordinates. This option is only available if you define the selection set by choosing to Select entities in drawing.

4.12.2.8 Create

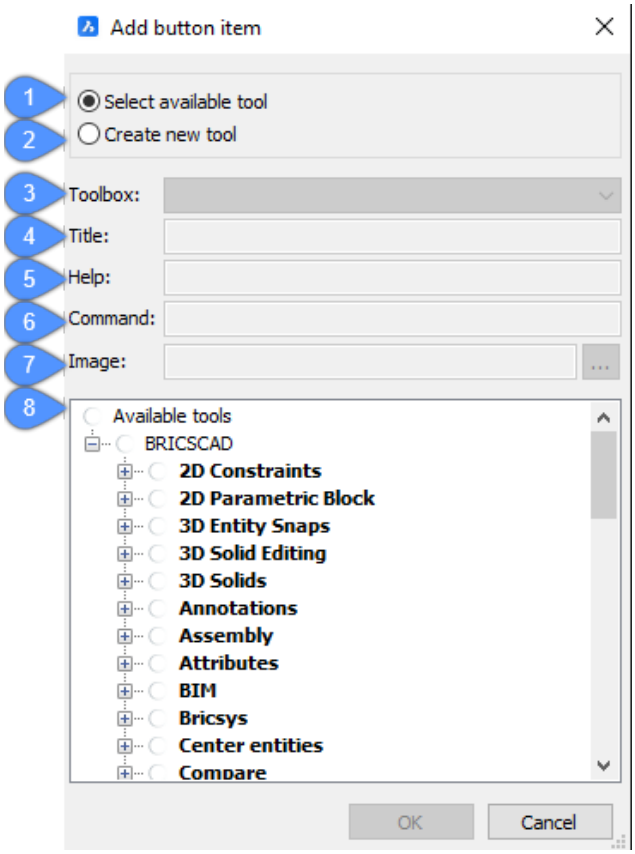
Creates a DWG file for the block definition in the specified location, adds the block to the library panel under the specified category, and closes the dialog box.

4.12.2.9 Cancel

Closes the dialog box without creating the block definition.

4.12.3 Add button item dialog box

The Add button item dialog box allows you to add a button item by selecting an available tool or by creating a new tool.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

4.12.3.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

4.12.3.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

4.12.3.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

4.12.3.4 Title

Specifies the name of the new tool.

4.12.3.5 Help

Specifies the help string displayed on the status bar.

4.12.3.6 Command

Specifies the command(s) or macros.

4.12.3.7 Image

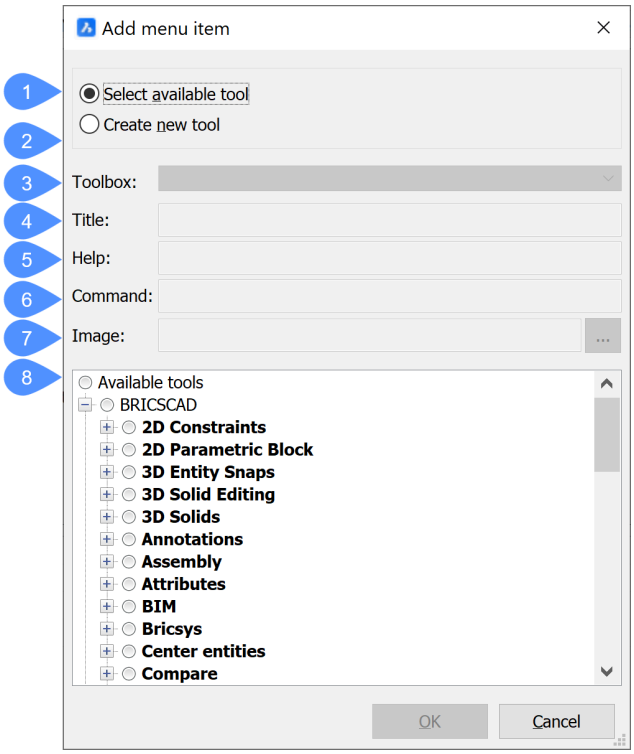
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

4.12.3.8 Available tools

Selects an existing tool.

4.12.4 Add menu item dialog box

The Add button item dialog box allows you to add a menu item by selecting an available tool or by creating a new tool.



1 Select available tool

2 Create new tool

3 Toolbox

4 Title

5 Help

6 Command

7 Image

8 Available tools

4.12.4.1 Select available tool

Assigns an existing command to thenewbutton or button group. If this option is selected, the other options are grayed out, except for Available tools.

4.12.4.2 Create new tool

Assigns a new command/macro to the newbutton or button group. If this option is selected, Available toolsisgrayed out.

4.12.4.3 Toolbox

Specifies the toolbox to which to add the new command. You can choose a toolboxinthedrop-downlist.

4.12.4.4 Title

Specifies the name of the new tool.

4.12.4.5 Help

Specifies the help string displayed on the status bar.

4.12.4.6 Command

Specifies the command(s) or macros.

4.12.4.7 Image

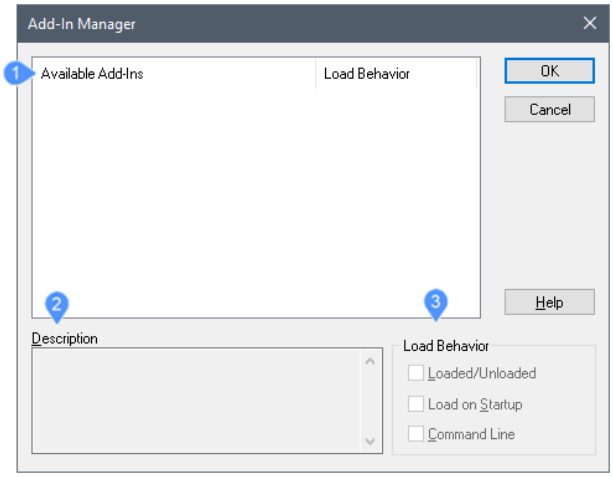
Specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image.

4.12.4.8 Available tools

Selects an existing tool.

4.12.5 Add-in manager dialog box

The Add-In Manager dialog box controls how Visual Basic for Applications COM (Common Object Model) modules are loaded. You can either toggle the modules between loaded and unloaded or start modules when BricsCAD starts up.



1 Available Add-ins

2 Description

3 Load Behavior

4.12.5.1 Available Add-ins

Lists the names and status of VBA COM modules available to BricsCAD. "VBA COM" is short for Visual Basic for Applications Common Object Model.

4.12.5.2 Description

Describes the selected VBA COM module, if any.

4.12.5.3 Load Behavior

Determines how modules are loaded.

Loaded/Unloaded
Toggles the module between loaded and unloaded.

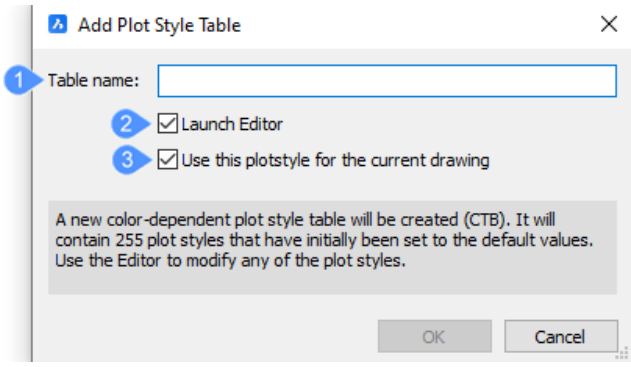
Load on Startup
Loads modules when BricsCAD starts up.

Command line
Loads modules through the Command line.

4.12.6 Add plot style table dialog box

The Add plot style table dialog box allows you to create new plot styles.

If the drawing uses named plot styles, the wizard creates a new named plot style table (STB file). If the current drawing does not use named plot styles, the wizard creates a new color-dependent table (CTB file).



1 Table name

2 Launch Editor

3 Use this Plotstyle for the current drawing

4.12.6.1 Table name

Enter the name you want to give to the plot style table.

4.12.6.2 Launch Editor

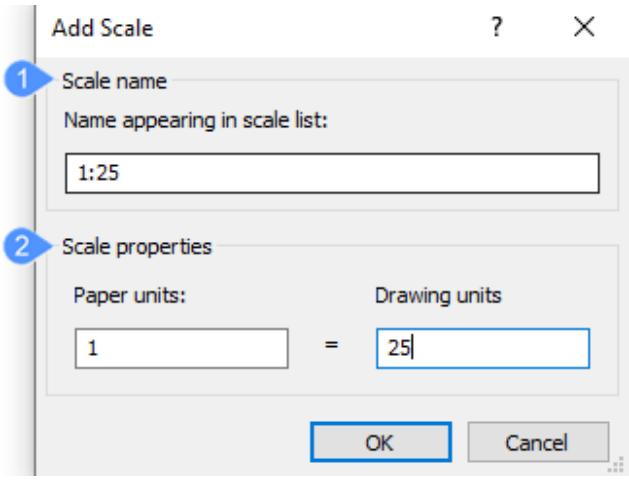
Opens the Plot style table editor dialog box after you enter the name and press the OK button.

4.12.6.3 Use this Plotstyle for the current drawing

If you do not check the Launch Editor option, it saves and sets the table name you have entered as current plot style table in the current layout.

4.12.7 Add scale dialog box

The Add Scale dialog box allows you to add a new scale factor to the list.



1 Scale name

2 Scale properties

4.12.7.1 Scale name

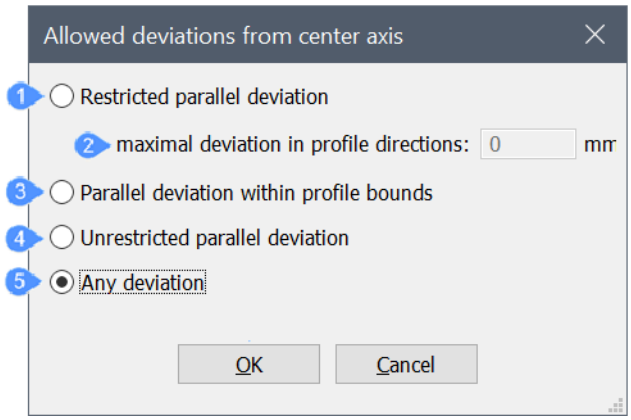
Specifies the scale name as it will appear in the scale list.

4.12.7.2 Scale properties

Specifies the scale factor in paper (size when plotted) and drawing units.

4.12.8 Allowed deviations from center axis dialog box

The Allowed Deviation from Center Axis dialog box allows you to set the level of freedom used when generating an analytical model.



1 Restricted parallel deviation

2 Maximal deviation in profile directions

3 Parallel deviation within bounds

4 Unrestricted parallel deviation

5 Any deviation

4.12.8.1 Restricted parallel deviation

Axes remain parallel to center axis and within the specified distance from the center axis in both profile directions.

4.12.8.2 Maximal deviation in profile directions

Set the maximal deviation in profile directions.

4.12.8.3 Parallel deviation within bounds

Axis remain parallel to center axis and do not exceed profile bounds.

4.12.8.4 Unrestricted parallel deviation

Axes remain parallel to center axis, but can be placed outside the profile bounds if is useful to avoid node splits or rigid links.

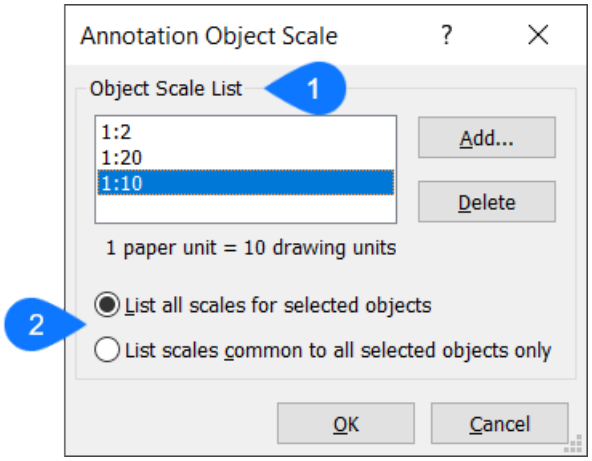
4.12.8.5 Any deviation

The main axes remain parallel to the center axis, but tilted axis (with respect to vertical and main horizontal directions) can be allowed to deviate from parallel if useful to avoid node splits and rigid links. Other axes can move outside of profile bounds.

Click **OK** to open the Visual settings dialog box.

4.12.9 Annotation Object Scale dialog box

The Annotation Object Scale dialog box allows you to add or remove scale factors used with annotative entities.



1 Object Scale List

2 Options

4.12.9.1 Object Scale List

Shows all the scales supported by the selected annotative objects.

Add

Adds annotation scale factors to the selected entities.

Delete

Removes annotation scales from the selected entities. Choose **No** in the **Annotative** field on the Properties Bar to remove the annotative property.

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

4.12.9.2 Options

List all scales for selected objects

Lists all the scale factors.

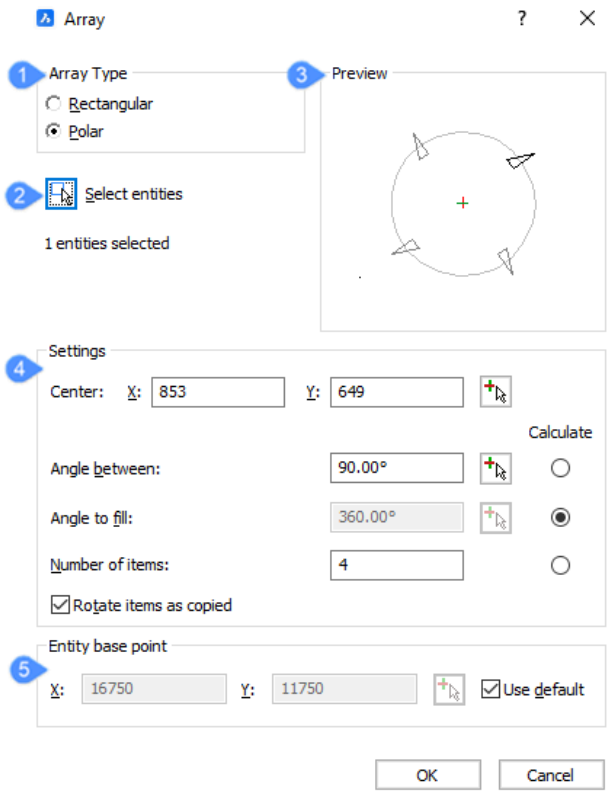
List scales common to all selected objects only

Lists only the scale factor shared by the selected entities.

Nota: This option is meaningless when only one entity is selected.

4.12.10 Array dialog box

The Array dialog box allows you to create a 2D rectangular or polar array of entities.



1 Array Type

2 Select entities

3 Preview

4 Settings

5 Entity base point (Polar only)

4.12.10.1 Array Type

Specifies whether a rectangular or polar array will be created.

4.12.10.2 Select entities

Temporarily closes the Array dialog box and prompts you to select entities. Press Enter to complete your select and return to the Array dialog box.

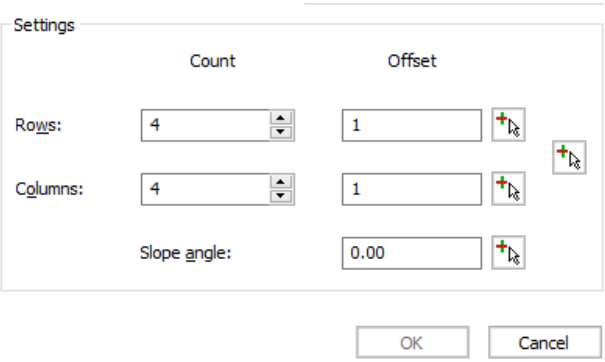
4.12.10.3 Preview

Displays a preview of the array based on the specified settings.

4.12.10.4 Settings

Specifies the number and location of items in the array. The available settings differ based on the array type.

Rectangular
Copies the selected entities in a rectangular pattern.



Count
Specifies the number of rows and columns in the array.

Offset
Specifies the distance between rows and columns. You can type a distance in the Offset fields or click the buttons to pick the offset distance in the drawing.

Offset field
Enables you to type distances for the offsets. Positive distances add rows along the positive Y axis and add columns along the positive X axis. Negative distances add rows along the negative Y axis and add columns along the negative X axis.

Offset buttons
Enables you to specify the offset distance by picking two points in the drawing. Clicking one of these buttons temporarily closes the Array dialog box and prompts you to pick points.

Pick Row Offset
Specifies the distance between rows. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if rows are added along the positive or negative Y axis.

Pick Column Offset

Specifies the distance between columns. A temporary line displays between the two points you pick. The length of the line determines the offset distance. The angle of the line determines if columns are added along the positive or negative X axis.

Pick Both Offsets

Specifies the distance between rows and the distance between columns. A temporary rectangle displays between the two points you pick. The length of the rectangle determines the distance between columns and the width of rectangle determines the distance between rows.

Slope Angle

Specifies the angle of the array from the X axis. You can type an angle in the Angle field or click the button to pick an angle in the drawing.

Angle field

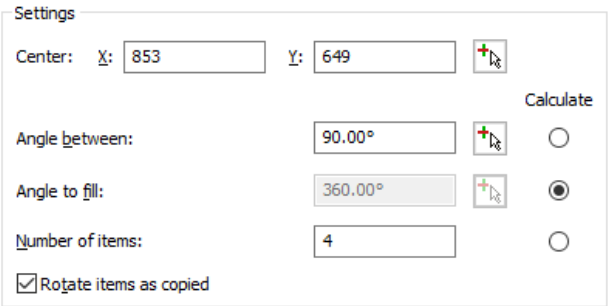
Enables you to type an angle for the slope.

Pick Angle of Array button

Enables you to specify the angle of the array by picking two points in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick points.

Polar

Copies the selected entities in a circular pattern.



Center

Specifies the center point of the polar array. You can type coordinates in the X and Y fields or click the button to pick the center in the drawing.

X field

Enables you to type a coordinate to place the center of the array along the X axis.

Y field

Enables you to type a coordinate to place the center of the array along the Y axis.

Pick Center Point button

Enables you to specify the center of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

Angle between

Specifies the angle between each item in the polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field

Enables you to type a value to specify the angle between each item of the array.

Pick Angle Between Items button

Enables you to specify the angle between each time by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the angle between items.

Calculate

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

Angle to fill

Specifies the fill angle and direction for a polar array. You can type an angle in the field, click the button to pick an angle in the drawing, or choose the Calculate option.

Angle field

Enables you to type a value to specify the fill angle of the array. Enter 360 for a full circle of copies. Enter a smaller number to draw a partial polar array. Positive values draw the array counterclockwise and negative values draw the array clockwise.

Pick Angle to Fill button

Enables you to specify the fill angle of the array by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point. A temporary line displays between the center point of the array and the point you pick. The angle of the line determines the fill angle.

Calculate

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

Number of items

Specifies the number of items in the polar array. You can type a number in the field or choose the Calculate option.

Number field

Enables you to enter the number of items to include in the array.

Calculate

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

Rotate items as copied

Specifies whether array items are rotated when they are copied to form the polar array.

- No: does not rotate items; all copies have the same orientation as the original item.
- Yes: rotates copies.

4.12.10.5 Entity base point (Polar only)

Specifies the base point of items in a polar array.

X field

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

Y field

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

Pick base point button

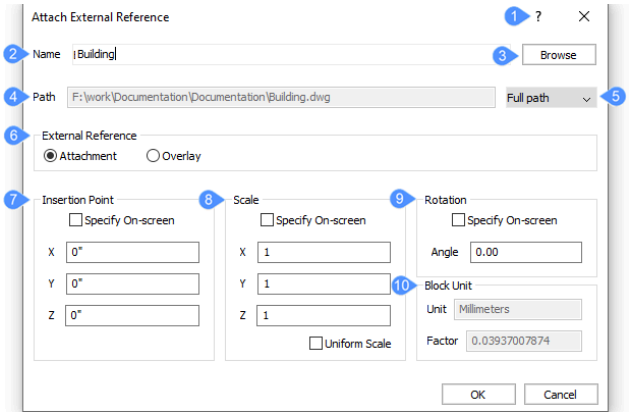
Enables you to specify the base point of array items by picking a point in the drawing. Clicking the button temporarily closes the Array dialog box and prompts you to pick a point.

Use Default

Uses the default basepoint for array entities. Choosing the Use default option disables the other entity base point options.

4.12.11 Attach external reference

The Attach External Reference dialog box allows you to attach externally - referenced drawings to the current drawing.



1 Help

2 Name

3 Browse

4 Path

5 Path type

6 External reference

7 Insertion Point

8 Scale

9 Rotation

10 Block Unit

4.12.11.1 Help

Opens the Bricsys Help article about the XATTACH command.

4.12.11.2 Name

Specifies the name of the DWG file attached to the current drawing.

4.12.11.3 Browse

Opens the Select Reference File dialog box to allow you to choose a different *.dwg file

4.12.11.4 Path

Displays the path of the drawing file.

4.12.11.5 Path type

Determines how much of the path is stored:

Full path
Stores the drive and folder names of the DWG file as an absolute reference, such as *c:\cad\dwg\filename.dwg*

Relative path
Stores the path from the parent drawing's location to the referenced drawing's location, such as *..\filename.dwg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the DWG file name, such as *filename.dwg*.

4.12.11.6 External reference

Specifies how the external reference is to be attached.

Attachment
Attaches the xref and nested xrefs.

Overlay
Attaches only the first level of xref.

4.12.11.7 Insertion Point

Specifies the location of the xref's lower-left corner:

Specify On-screen
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y, and Z fields.

X, Y, or Z
Specifies the x, y, and/or z coordinates for the xref's insertion point. Use 0,0,0 to insert the xref at the drawing's origin.

4.12.11.8 Scale

Specifies the size of the xref.

Specify On-screen
Determines how you specify the scale factors.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X, Y, and Z fields.

X, Y, or Z
Specifies the x, y, and/or z scale factors of the xref in the x, y, and z directions. Use 1,1,1 to keep the xref at its original size.

Uniform Scale
Makes the y and z scale factors equal to x.

4.12.11.9 Rotation

Specifies the rotation angle of the xref.

Specify On-screen
Determines how you specify the rotation angle.

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

Angle
Specifies the rotation angle about the insertion point. Positive angles rotate the xref counterclockwise. Negative angles rotate the xref clockwise. Use 0 to keep the xref at its original orientation.

4.12.11.1 Block Unit

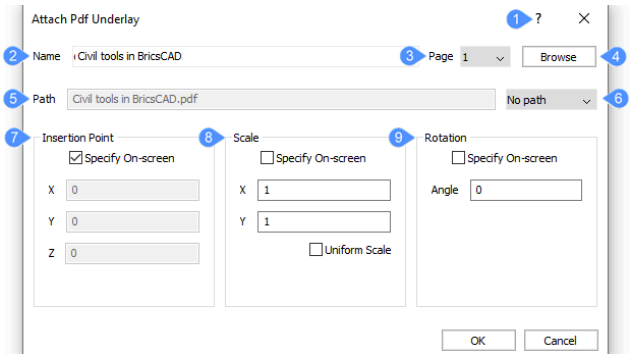
0
Specifies the units with which the drawing will be inserted, usually a form of Imperial or metric units, such as inches or mm:

Unit
Reports the INSUNITS setting of the drawing being attached.

Factor
Reports the calculated scaling factor with respect to the INSUNITS setting of the drawing being attached and the current drawing.

4.12.12 Attach PDF underlay dialog box

The Attach PDF underlay dialog box allows you to attach a PDF file as an underlay into the current drawing.



1 Help

2 Name

3 Page

4 Browse

5 Path

6 Path type

7 Insertion Point

8 Scale

9 Rotation

4.12.12.1 Help

Opens the Bricsys Help article about the -PDFATTACH command.

4.12.12.2 Name

Displays the file name of the PDF file to attach. When more than one PDF file has been attached to the drawing, their names are listed in the drop-down list.

4.12.12.3 Page

Allows you to display the page you want from the whole PDF file. You can change the number of the page from the drop-down arrow list.

4.12.12.4 Browse

Opens the Select PDF underlay file dialog box from where you can select another PDF file than the one already selected.

4.12.12.5 Path

Displays the path of the file according to the path type you choose.

4.12.12.6 Path type

Choose the way the program remembers the path to the PDF file. You can select from the Path drop-down list one of the following options:

Full path
Stores the drive and the folder(s) name to the PDF file as an absolute reference such as: C:\Users\user \PDFATTACH\filename.pdf

Relative path
Stores the part of the path to PDF file's folder, such as .\filename.pdf. This option does not work until the drawing has been saved at least once. (..\ means one folder higher)

No path
Strips the drive and folder names, leaving just the PDF file name, such as filename.pdf.

4.12.12.7 Insertion Point

Specifies the location of the underlay's lower-left corner.

Specify On-screen
Determines how the insertion point is specified.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box, using the X, Y and Z fields.

X, Y, Z
Specifies the x, y, z coordinates for the PDF underlay's insertion point. Use 0,0,0 to insert the underlay at the drawing's origin.

4.12.12.8 Scale

Specifies the size of the PDF underlay:

Specify On-screen
Determines how the scale factors are specified:

- On: specify the scale factor in the drawing after the dialog box is closed.
- Off: specify the scale factor in the dialog box using the X and Y fields.

X,Y
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

4.12.12.9 Rotation

Specifies the rotation angle of the PDF underlay:

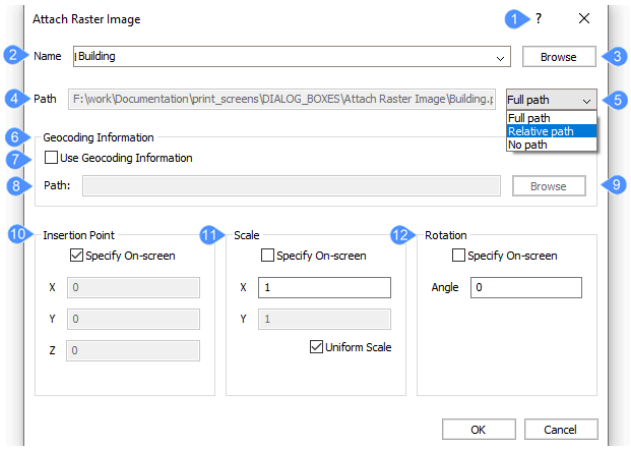
Specify On-screen
Determines how the rotation angle is specified:

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

Angle
Specifies the rotation angle about the insertion point. Positive angles rotate the underlay counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the underlay at its original orientation.

4.12.13 Attach raster image dialog box

The Attach image raster dialog box allows you to attach raster images to the drawing.



1 Help

2 Name

3 Browse

4 Path

5 Path type

6 Geocoding Information

7 Use Geocoding Information

8 Path

9 Browse (image position file)

10 Insertion Point

11 Scale

12 Rotation

4.12.13.1 Help

Opens the Bricsys Help article about the IMAGEATTACH command.

4.12.13.2 Name

Specifies the name of the file to attach.

4.12.13.3 Browse

Opens the Select Image File dialog box to allow you to choose a different raster file.

4.12.13.4 Path

Displays the path of the image file.

4.12.13.5 Path type

Specifies how much of the path is stored.

Full path
Stores the full path to the image file as an absolute reference such as: *C:\foldername\imagename.jpg*

Relative path
Stores the path from the drawing's location to the image's location. For example: *..\foldername \imagename.jpg*. The .. refers to the folder above the current one. The drawing must be saved before you can use this option.

No path
Strips the drive and folder names, leaving just the image file name such as *imagename.jpg*.

4.12.13.6 Geocoding Information

Position files or world files have an extension that depends on what kind of image file they are associated with.

Use Geocoding Information
Toggles use of a position file, which specifies the size, location and rotation of the image file.

- Yes: uses a position file
- No: does not use a position file.

Path
Displays the path of the position file.

Browse (image position file)
Opens the Select Image Position File dialog box.

4.12.13.7 Insertion Point

Specifies the location of the image's lower left corner.

Specify On-screen
Determines how you specify the insertion point.

- On: specify the insertion point in the drawing after the dialog box is closed.
- Off: specify the insertion point in the dialog box using the X, Y and Z fields.

X,Y,Z
Specifies the x, y, and z coordinates for the image's insertion point.

4.12.13.8 Scale

Specifies the size of the image.

Specify On-screen
Determines how you specify the scale factor.

- On: specify the scale factors in the drawing after the dialog box is closed.
- Off: specify the scale factors in the dialog box using the X and Y fields.

XY
Specifies the scale factors along the X and Y axes.

Uniform scale
Makes the Y scale factor equal to X.

4.12.13.9 Rotation

Specifies the rotation angle of the image.

Specify On-screen
Determines how you specify the rotation angle.

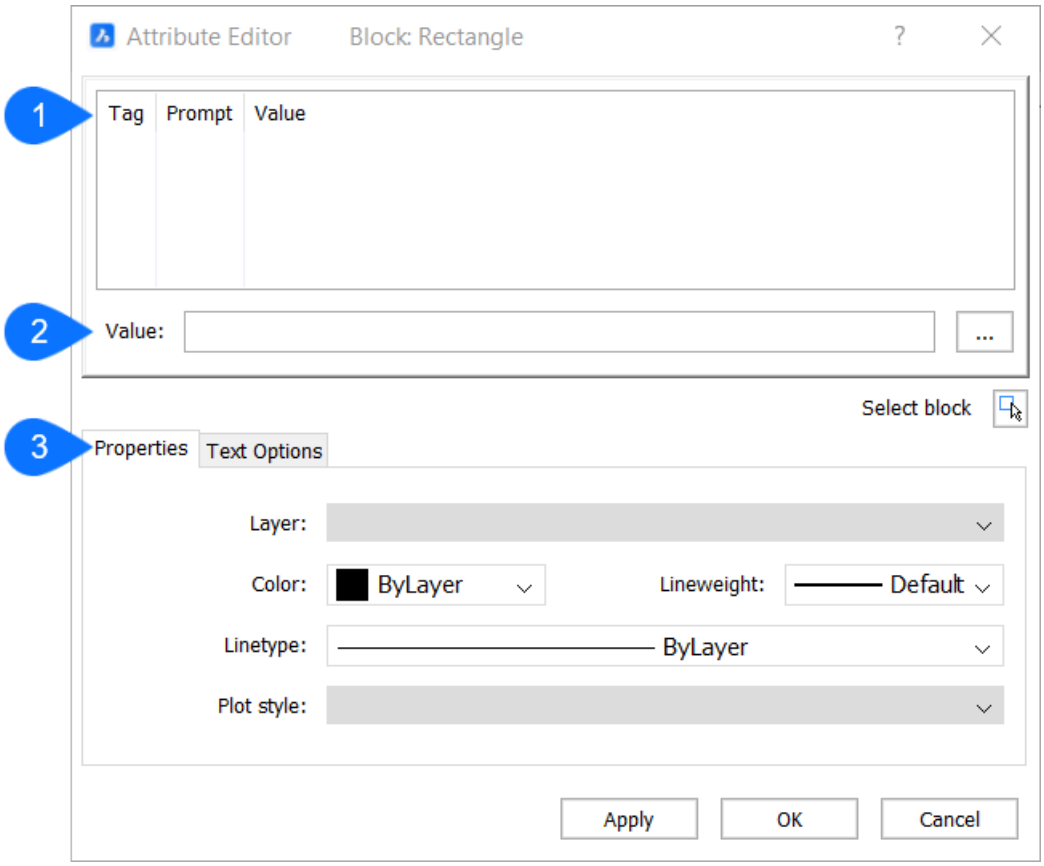
- On: specify the rotation angle in the drawing after the dialog box is closed.
- Off: specify the angle in the dialog box using the Angle field.

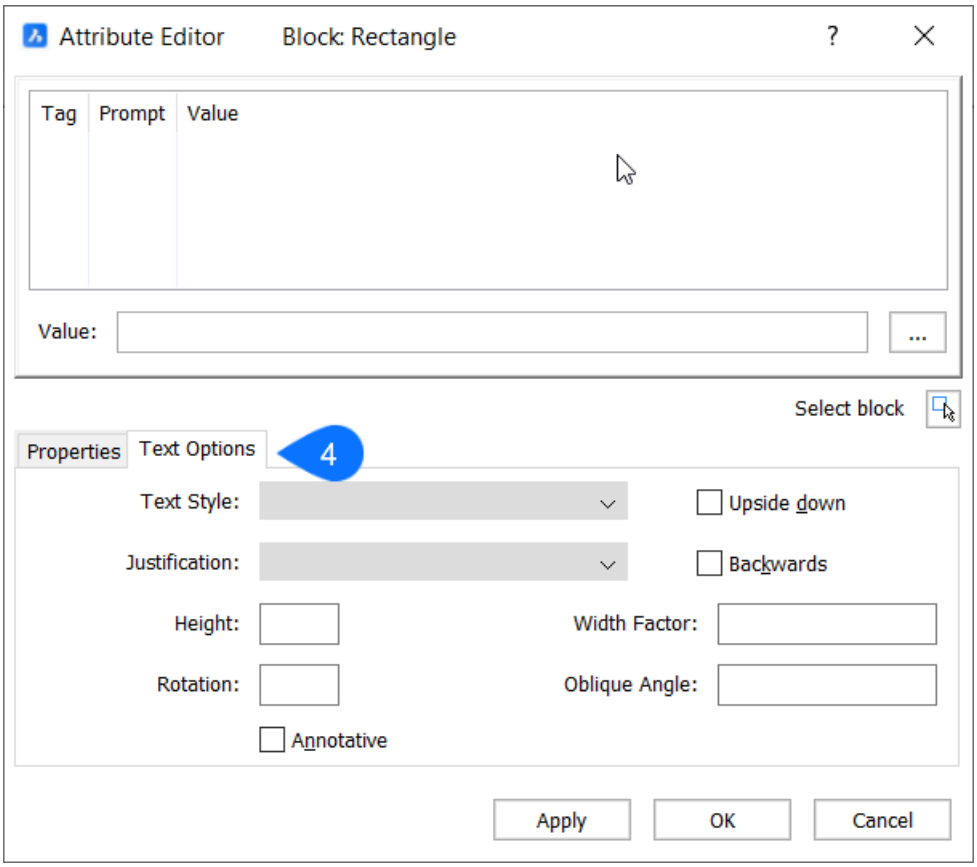
4.12.13.1 Angle

0
Specifies the rotation angle of the image about the insertion point. Positive angles rotate the image counterclockwise. Negative angles rotate the image clockwise. Use 0 to keep the image at its original orientation.

4.12.14 Attribute editor dialog box

The attribute Editor dialog box allows you to edit attributes within blocks.





1 Overview list

2 Value

3 Properties

4 Text Options

4.12.14.1 Overview list

Displays an overview of all attributes used in the block.

4.12.14.2 Value

Displays the currently selected attribute from the list. A new value can be typed.

4.12.14.3 Properties

Specifies properties of the selected attribute.

Layer
Specifies the layer of the selected attribute.

Color
Specifies the color of the selected attribute.

Linetype
Specifies the linetype of the selected attribute.

Plot style
Specifies the plotstyle of the selected attribute.

4.12.14.4 Text Options

Text style
Specifies the font of the text in the selected attribute.

Justification
Specifies the alignment of the text in the selected attribute.

Upside down
When checked, text is displayed upside down.

Backwards
When checked, text is displayed backwards.

Height
Specifies the height of the text in the selected attribute.

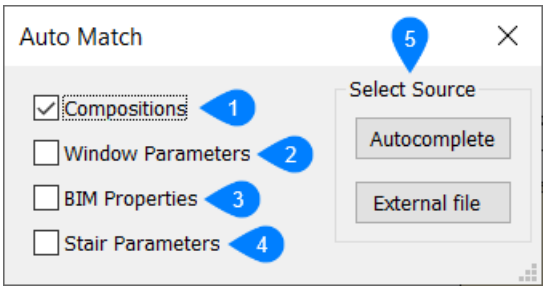
Rotation
Specifies the rotation angle of the text in the selected attribute.

Width factor
Specifies the width of the text in the selected attribute.

Oblique angle
Specifies the oblique angle of the text in the selected attribute.

4.12.15 Auto Match dialog box

The Auto Match dialog box defines which characteristics of example entities are matched and lets you select a source.



1 Compositions

2 Window Parameters

3 BIM Properties

4 Stair Parameters

5 Select Source

4.12.15.1 Compositions

When checked on, the BIM properties of entities will be copied.

Entities without a composition will be matched with the other compositions in your drawing, based on type, spatial location and orientation of the entity, as well as on the 'Is external'-property.

If all entities have compositions, you can overwrite some by choosing a source entity. AutoMatch will then copy this source's composition to the rest of the valid entities.

4.12.15.2 Window Parameters

When checked on, the parameters of the windows will be copied.

If there are unchanged windows in your drawing, AutoMatch will match the properties based on width, height, type, spatial location and orientation of the windows and type and composition of the entities they are in.

If all windows have changed parameters, you can overwrite some by choosing a source window. AutoMatch will then copy this source's parameters to the rest of the valid windows.

Nota: Some window parameters will never be copied along.

- W (= width of the window)
- H (= height of the window)

4.12.15.3 BIM Properties

When checked on, the BIM properties of entities will be copied.

If there are entities with unchanged properties in your drawing, AutoMatch will match the properties based on BIM type, composition and spatial location of the other entities in your drawing.

If all entities have a changed property, you can overwrite it by choosing a source entity. AutoMatch will then copy this source's property to the rest of the valid entities.

Some general BIM properties will never be copied along:

- GUID

- Composition
- Profile
- SpatialLocation
- IsExternal

Some beam and column related BIM properties will never be copied along:

- eomHash
- athString
- XAxis
- YAxis
- ZAxis
- ProfileName
- EndPoint
- StartPoint
- Valid
- InstancePropertySets

Some window and door related BIM properties will never be copied along:

- Name
- SubtractAllSolids
- Translucency
- OverallWidth
- OverallHeight

Some slab related BIM properties will never be copied along:

- RoomBounding

Some room related BIM properties will never be copied along:

- Number
- RoomArea
- RoomRepresentation

Some stair related BIM properties will never be copied along:

- StairType
- NumberOfRisers
- NumberOfTreads

4.12.15.4 Stair Parameters

When checked on, the parameters of a stair will be copied.

If there are unchanged stairs in your drawing, AutoMatch will match the properties based on number of treads, number of risers, height, width, length and type of the stair.

If all stairs have changed parameters, you can overwrite some by choosing a source stair. AutoMatch will then copy this source's parameters to the rest of the valid stairs.

Nota: Some stair parameters will never be copied along:

- Height_Staircase
- Stair_Width
- Length_Staircase

4.12.15.5 Select Source

Autocomplete
Matches all chosen options for entities in the open drawing with similar entities in the open drawing.

External file
Opens the Select file to insert dialog box. Here you can choose a file to match all chosen options for entities in the chosen drawing with similar entities in the open drawing.

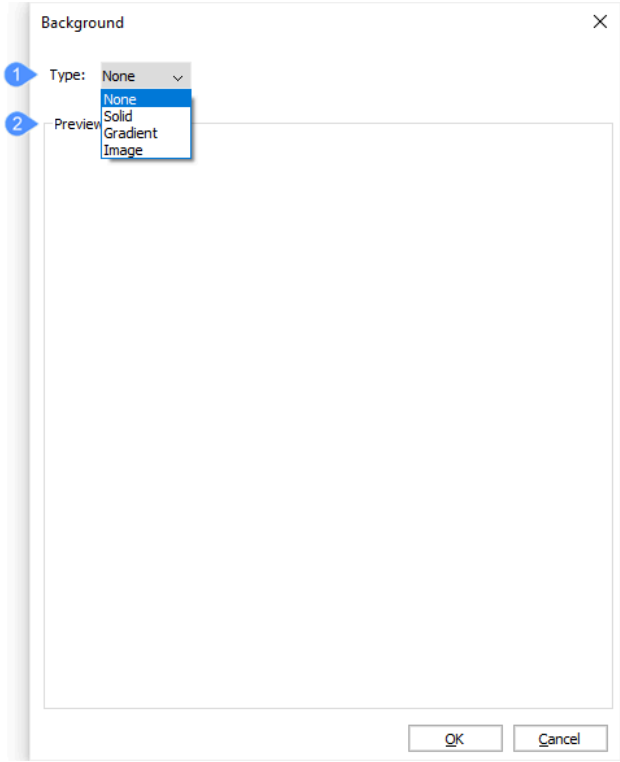
4.12.16 Background dialog box

The Background dialog box enables you to apply a background to the current viewport. You can choose from 4 types: None, Solid, Gradient, and Image.

Nota: Set Backgrounds = On in Visual Styles in Drawing Explorer dialog box to apply the changes you may make in Background dialog box. This option is available in all visual styles, except 2DWIREFRAME.

4.12.16.1 None

When background type is set to None, the Background dialog box includes 2 areas:



1 Type

2 Preview

Type
Specifies the background type for the current viewport. When the background type is None, default colors are applied to the current viewport.

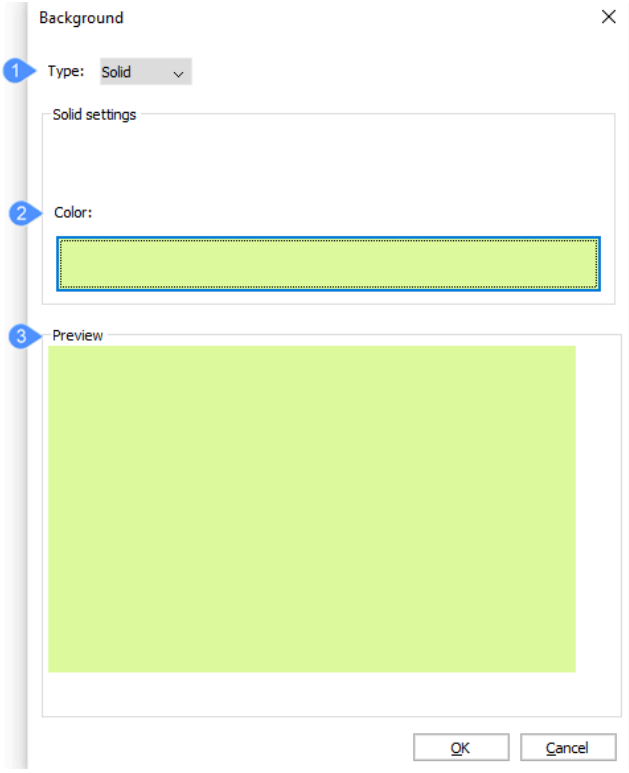
Preview
Displays a preview of the background. When set to None, the background depends on the PERSPECTIVE variable:

- **Off**: background is the color specified by the BKGCOLOR variable, most often black or white.
- **On**: two-color gradient that simulates the ground with the sky; the colors are specified by variables that begin with Gradient-, such as GRADIENTCOLORTOP, GRADIENTCOLORMIDDLE and GRADIENTCOLORBOTTOM.

Nota: After changing the variables' values, rerun the GRADIENTBKGON command.

4.12.16.2 Solid

When the background type is set to Solid, the Background dialog box includes 3 areas:



1 Type

2 Solid settings

3 Preview

Type

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

Solid settings

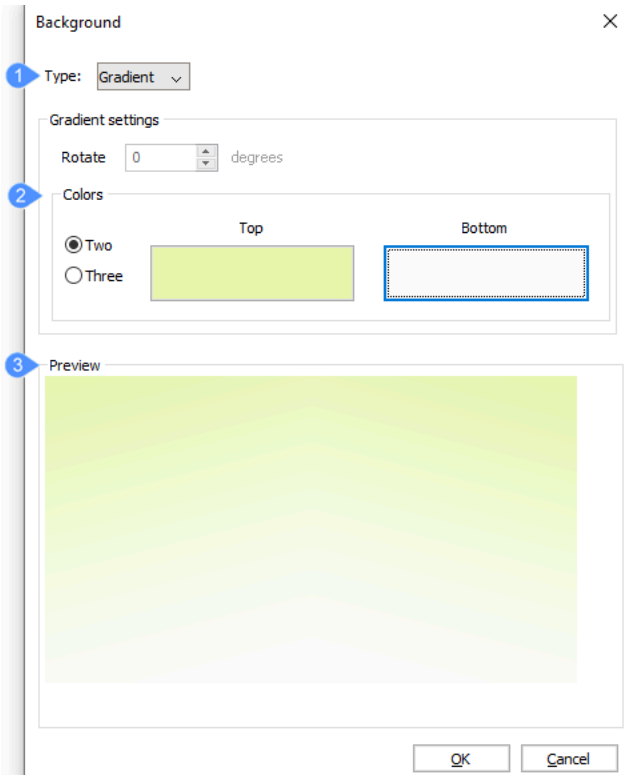
Specifies the current color for the solid background. You can click on the color swatch to display the Select Color dialog box and specify a different color.

Preview

Displays a preview of the background using the color you selected.

4.12.16.3 Gradient

When the background type is set to Gradient, the Background dialog box includes 3 areas:



1 Type

2 Gradient settings

3 Preview

Type

Specifies the background type for the current viewport. When the background type is Gradient, a gradient between 2 or 3 colors is applied to the current viewport.

Gradient settings

Specifies the rotation and colors for the gradient background.

Rotate field

Specifies the rotation angle of the gradient background. You can type an angle or use the arrows to increase or decrease the angle.

Colors

Specifies the colors for the gradient background.

Two

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

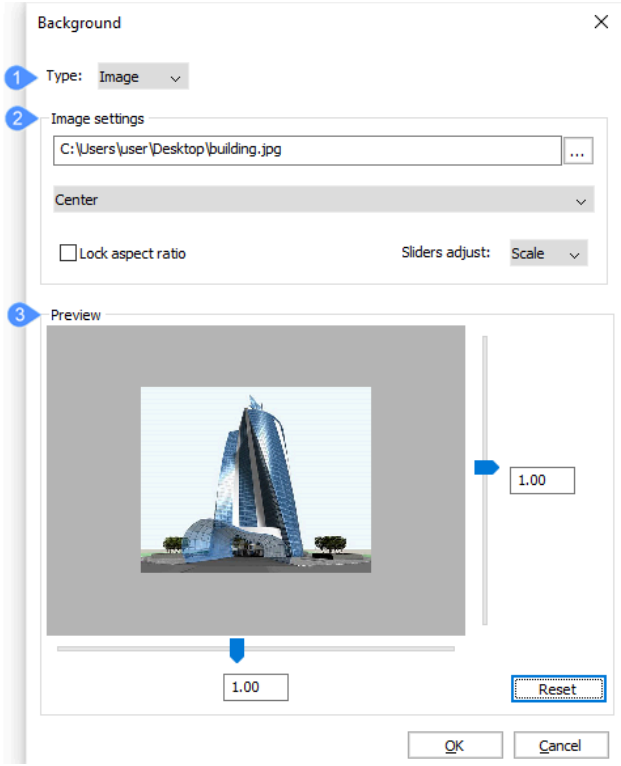
Three

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

Preview
Displays a preview of the background using the colors you have selected.

4.12.16.4 Image

When the background type is set to Image, the Background dialog box includes 3 areas:



1 Type

2 Image settings

3 Preview

Type
Specifies the background type for the current viewport. When the background type is Image, a raster image is applied to the current viewport.

Image settings
Specifies the file and position for the image background.

Path field
Specifies the name and location of the image file. You can enter the path by typing or choose the browse button to open the Select Image File dialog box.

Position
Specifies the position of the image within the current viewport. Options include Center, Stretch and Tile.

Center
Centers the image in the viewport.

Stretch
Stretches small images to fit the size of the viewport.

Tile
Tiles the image as needed to fill the viewport.

Lock aspect ratio

Specifies whether the aspect ratio of the image is maintained when the image is scaled. This option is only available when the Sliders Adjust option is set to **Scale**.

- **ON**: maintains the image's aspect ratio; the x and y sliders move together.
- **OFF**: allows the image to be distorted; the x and y sliders move independently of each other.

Sliders adjust

Specifies the function of the 2 slides that display in the Preview pane.

- **Offset**: moves the image in the viewport.
- **Scale**: resizes the image; allows you to make the image the same size as the viewport.

Preview

Displays a preview of the background using the image file and options you specify.

X and Y sliders

Adjusts the offset and/or the scale of the image in the viewport. Which one is adjusted, offset or scale, is determined by the Slides adjust setting.

- **X**: moves the slider left and right. The X field updates accordingly. You can also type a value into the X field.
- **Y**: moves the slider up and down. The Y field updates accordingly. You can also type a value into the Y field.

Reset

Resets the offset to a default value of 0 and resets the scale to a default value of 1.

4.12.17 BIM BCF panel - Custom login dialog box

The BIM BCF Custom Log In dialog box allows you to connect to a cloud service to start collaborating on issues in the drawings.

Source URL

It is generally easier to use cloud services to manage BCF issues. If you wish to connect to these services from within BricsCAD BIM, you will need an account of these services first. This allows you to create issues and manage them in real-time from anywhere.

Nota: It is not possible tocreateissues within BricsCAD BIM. The BCF panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

To connect to a cloud service, insert in the Source URL field one of the following Cloud services:

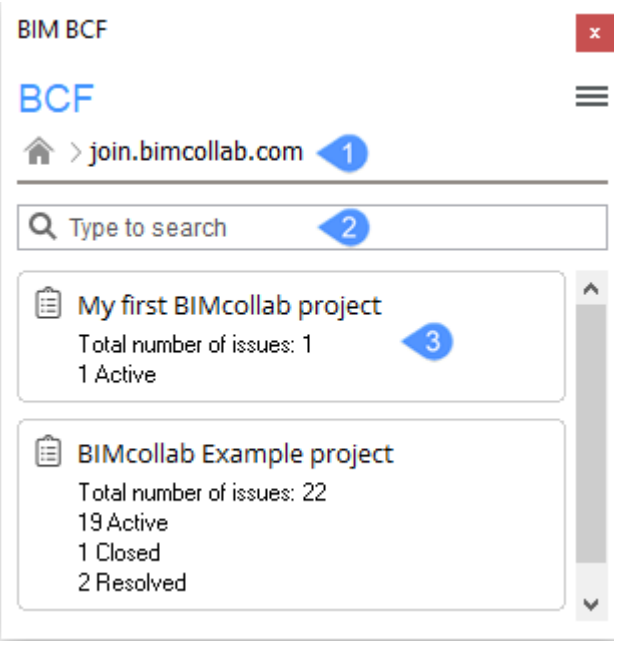
Cloud Service	Source URL
BIMcollab	<space name>.bimcollab.com *
BimSync	bcf.bimsync.com
BIMtrack	Bcfrestapi.bimtrackapp.co

Nota: * The Source URL for BIMcollab includes the name of the space you bought. If using a free account, use join.bimcollab.com.

4.12.17.1 Log in

Connects you to the cloud service. You are asked to insert the e-mail address and the password to connect.

After you successfully log in, you can close the browser window and start collaboration on issues in BricsCAD.



1 Source URL

2 Search

3 Project list

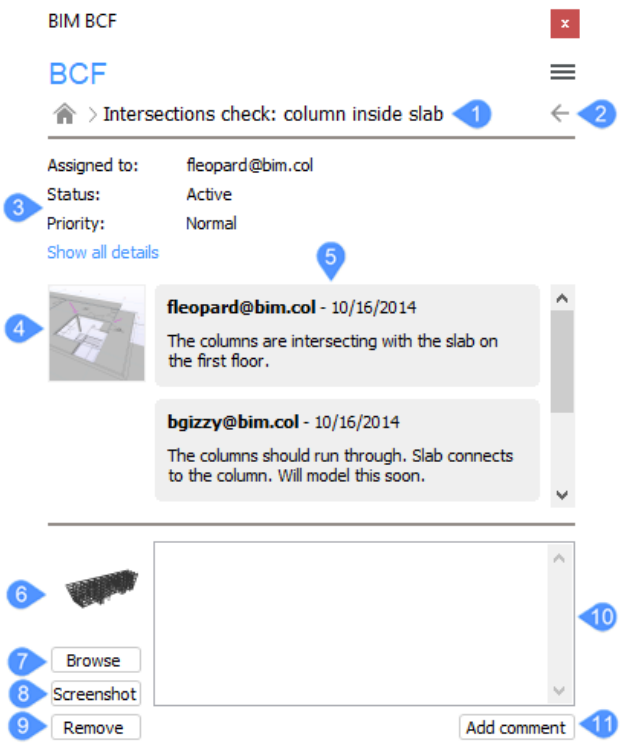
Source URL
Displays the source URL.

Search
Displays only the projects filtered by the word you type in the search bar.

Project list
Displays the list of projects that you are involved in.

4.12.17.2 Project list

After you left-click the project will open it and show a list of the issues. When you left-click on an issue, this dialog box will open.



1 Issue name

2 Issue overview

3 Details

4 Thumbnail image

5 Comments

6 Preview

7 Browse

8 Screenshot

9 Remove

10 Comment

11 Add comment

Issue name
Displays the name of the opened issue.

Issue overview
Returns you to the issue overview.

Details
Displays detailed information about the issue, such as status, author and creation date.

Thumbnail image
If the comment has an associated camera position, then clicking the thumbnail image will take the camera in the current drawing to this position.

This is useful if you want to know where exactly the issue is, and you do not want to manually search through the model.

Comments
Displays the comments on the issue.

Preview
Displays a preview of the issue after you press the button no. 8.

Browse
Opens the Select a viewpoint for this comment dialog box.

Screenshot
Insert a screenshot from model space in the preview field (no. 6).

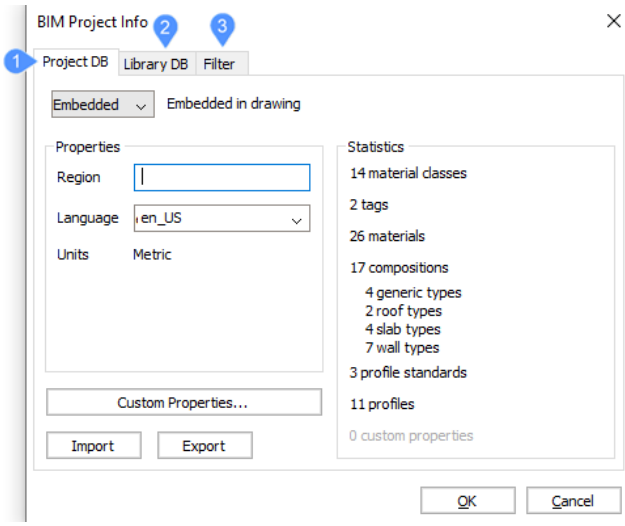
Remove
Removes the screenshot preview.

Comment
Allows you to insert a comment for an issue.

Add comment
Adds the comment you have inserted in field no. 10 to an existing issue.

4.12.18 BIM project info dialog box

The BIM Project Info dialog box allows you to set up your project and library database of your BIM models, to control the location, properties and content of the Project and Library databases.



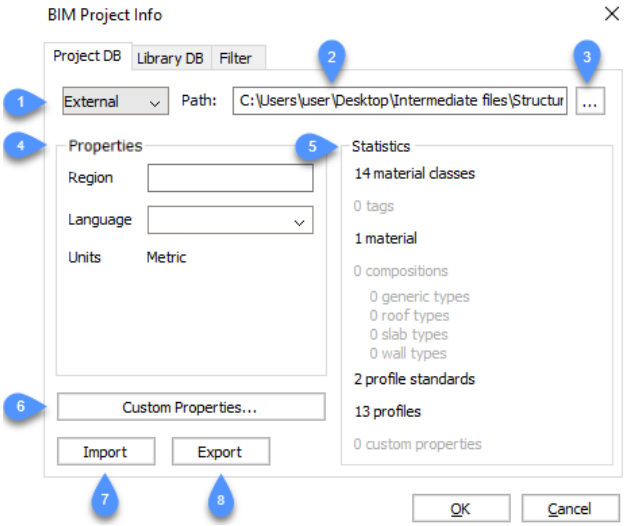
1 Project DB

2 Library DB

3 Filter

4.12.18.1 Project DB

Shows the BIM project info.



1 Project database drop-down list

2 Path

3 Browse

4 Properties

5 Statistics

6 Custom properties...

7 Import

8 Export

Project database drop-down list
Choose between Embedded or External.

Embedded
Saves the project library in the drawing file.

External
Saves the project library in a BIM database file (.bsyslib).

Nota: The external database keeps the database independent from the current drawing. This is especially useful if you want to share the library between multiple drawings.

When switching from External to Embedded the content of the external library is copied to the embedded project library.

Path
Displays the path where the project library can be found.

Browse
Opens the Select or create a Bricsys Library dialog box.

Properties
Region, Language and Units are the properties of the BIM project.

Region
Fill out the region field.

Language
You can choose one of the language options from the drop-down list.

Units
Displays the system of measurement used in the BIM project (metric, imperial).

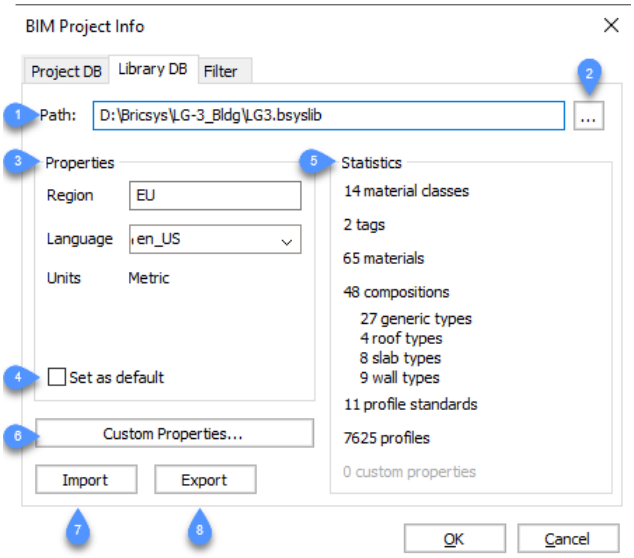
Statistics
Shows the location, properties, and the content of the current project library of the BIM model.

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

Import
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

Export
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

4.12.18.2 Library DB



1 Path

2 Browse

3 Properties

4 Set as default

5 Statistics

6 Custom Properties…

7 Import

8 Export

Path
Displays the path where you the project library can be found.

Browse
Opens the Select or create a Bricsys Library dialog box.

Properties
Region, Language and Units are the properties of the project.

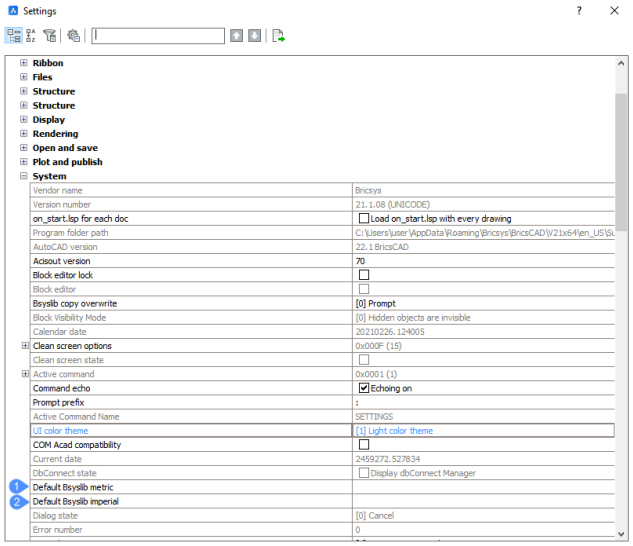
Region
Fill out the region field.

Language
You can choose one of the language options from the drop-down list.

Units
Displays the system of measurement used in the BIM project (metric, imperial).

Set as default
This option allows you to set the selected database as default library database for next projects.

Nota: You can edit the default library settings in the Settings dialog box under Program options/ System.



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

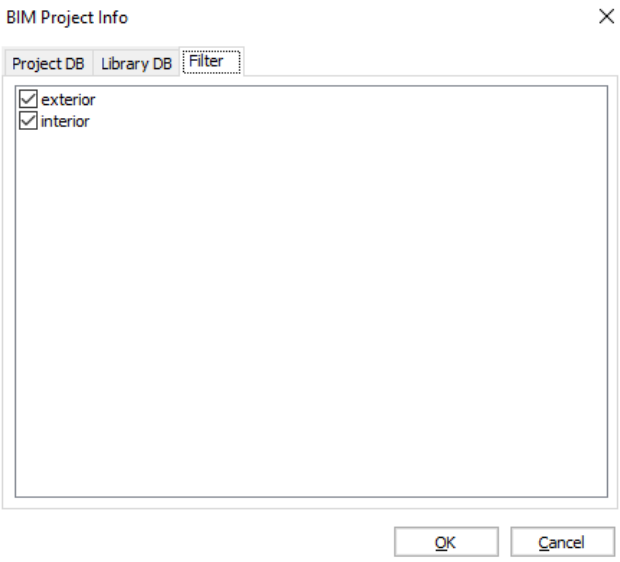
Custom Properties…
Adds custom properties to materials and compositions. Opens the Edit Custom Properties dialog box.

Import
Imports the selected *.xml library files. Opens the Select xml/csv file to import dialog box.

Export
Saves .xml library files in the defined location. Opens the Select location to save xml file dialog box.

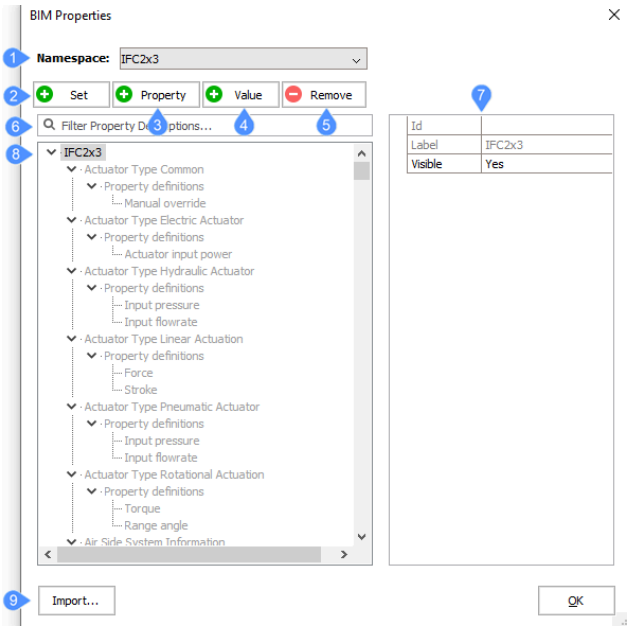
4.12.18.3 Filter

Check the tags you want to use in the filter.



4.12.19 BIM properties dialog box

The BIM Properties dialog box allows you to create, edit and delete property definitions and to organize properties in property sets.



1 Namespace

2 Add Set

3 Add Property

4 Add Value

5 Remove

6 Filter

7 Properties

8 Properties Tree

9 Import

4.12.19.1 Namespace

By default, 3 namespaces are available:

- **IFC2X3**: shows the IFC2x3 properties; is read-only. This namespace contains properties that are defined by the IFC2X3 schema.
- **Quantity**: shows the quantities properties; is read-only.
- **User**: allows you to create user defined properties.

It is also possible to import namespaces, using the **Import** button.

- **Classification System**: allows you to organize your BIM models with standard classification codes that are used in the construction industry.

4.12.19.2 Add Set

Creates a new property set.

To create a custom property set, first select the user option in the namespace drop-down list, then click the button.

Do the following:

- Type a name in the **Id** field.
- Type a name in the **Label** field.

  The Label name appears in the properties tree. If the **Label** field is empty, the **Id** name is displayed. If you type in the **Label** field, the **Id** field will not be displayed any more. The Id must be unique.
- Per Instance: Allows to enable/disable user-defined properties, on a per-entity basis. When set, the BIM/Entity property sets property is available in the Properties panel when a BIM entity is selected.

4.12.19.3 Add Property

Creates a new property set.

4.12.19.4 Add Value

Creates a new value definition.

4.12.19.5 Remove

Removes the selected property set, property definition or value definition.

4.12.19.6 Filter

Displays only the properties of which the Id and Label contain the search string typed. The properties tree is filtered as you type.

4.12.19.7 Properties

- **Id**: name of the userspace. This field is not editable.
- **Label**: the label of the namespace is used in the properties tree. If the Label field is empty, then the Id is used instead.
- **Visible**: defines whether the namespace properties are visible in the Properties panel.

4.12.19.8 Properties Tree

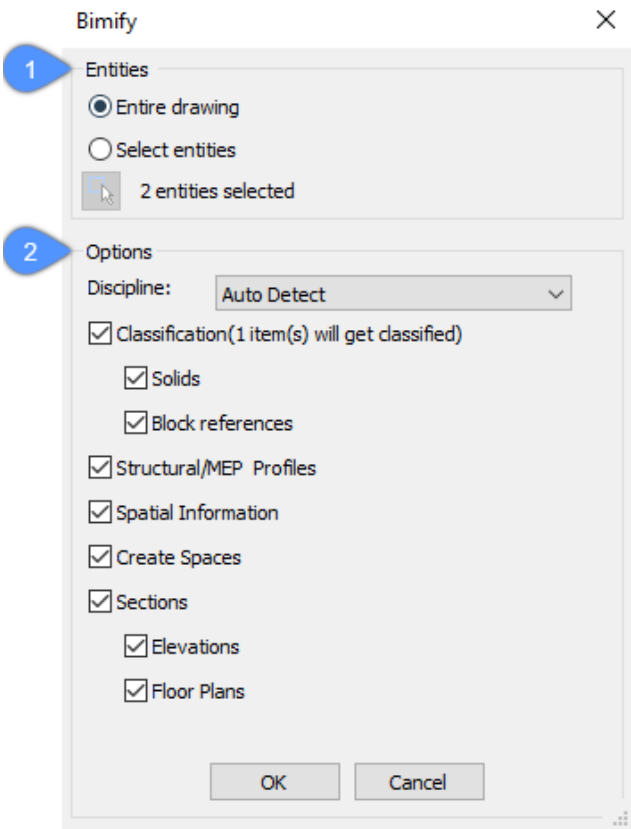
Displays the property sets, property definitions and value definitions.

4.12.19.9 Import

Opens the Select XML file to import dialog box.

4.12.20 Bimify dialog box

The Bimify dialog box allows you to analyze the model and to run an automatic classification and a spatial location assignment for the entire model.



1 Entities

2 Options

4.12.20.1 Entities

Specifies if the BIMIFY command needs to be run for all entities in the drawing or just several selected entities.

4.12.20.2 Options

Discipline
Determines to what discipline the model belongs to in order to improve the automatic classification.

- **Auto Detect**: an AI algorithm determines the discipline automatically.
- **Architectural**: the model is classified as an architectural model.
- **MEP**: the model is classified as a MEP model.
- **Mixed Model**: the model cannot be assigned to 1 discipline and is perceived as a mixed model.

Classification
Defines the classification of solids and blocks.

- **Solids**: makes sure that 3D solids are classified as walls, slabs, etc.
- **Blocks**: makes sure that blocks are classified as windows, doors, etc.

Structural/MEP profiles
Assigns columns, beams, members or flow segments profiles that are available in BricsCAD BIM profile panel.

Nota: If no match can be found for a profile element in the standard library, the *BIMIFY* command creates a new custom profile in the project's library.

Spatial information
Assigns spatial locations (buildings and floors).

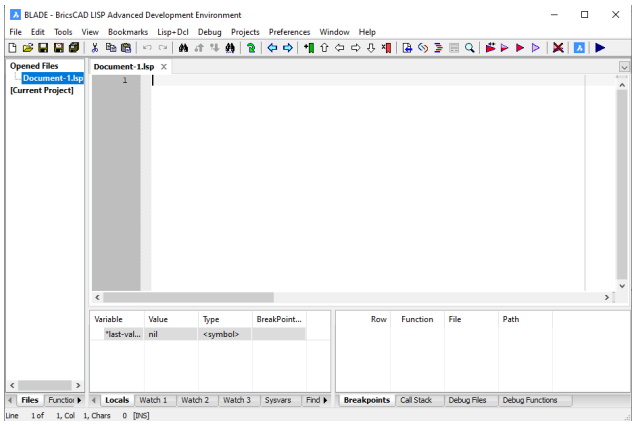
Create Spaces
Detects external walls and sets the property Wall Common/is External = ON.

Sections
Creates BIM Section entities.

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

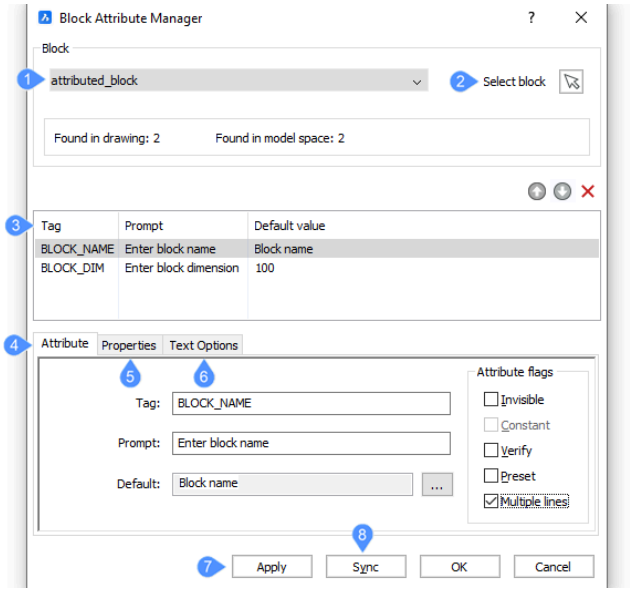
4.12.21 Blade - BricsCAD LISP advanced development environment dialog box

The Blade - BricsCAD LISP Advanced Development Environment dialog box allows you to edit and debug LISP applications.



4.12.22 Block attribute manager dialog box

The Block attribute manager dialog box allows you to edit all aspects of attributes in a block definition and then optionally apply the changes to all blocks of the same name in the drawing.



1 Block name

2 Select block

3 Attribute list

4 Attribute tab

5 Properties tab

6 Text Options Tab

7 Apply

8 Sync

4.12.22.1 Block name

Select a block from the drawing whose attributes you want to edit.

You can select a block from the drop-down list.

4.12.22.2 Select block

Allows you to select a block in the drawing.

4.12.22.3 Attribute list

Select an attribute from the list. It includes the tag, the prompt and the default value.

: moves the attribute up the list.

: moves the attribute down the list.

: deletes the attribute up the block.

4.12.22.4 Attribute tab

Allows you to edit the definition of the selected attribute.

There are 5 attribute flags options:

- **Invisible**: hides the attributes from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the *ATTDISP* command.
- **Constant**: specifies a default value that the user cannot change.
- **Verify**: forces the user to enter the value a second time; this helps to ensure the value is entered correctly.
- **Preset**: inserts attributes without prompting the user; the attributes can be changed later with the ATTEDIT command.
- **Multiple lines**: when checked, allows the use of multiline text.

4.12.22.5 Properties tab

The**Properties**tab allows you to edit the properties of the selected attribute.

4.12.22.6 Text Options Tab

The**Text Options**tab allows you to edit the text properties of the selected attribute.

4.12.22.7 Apply

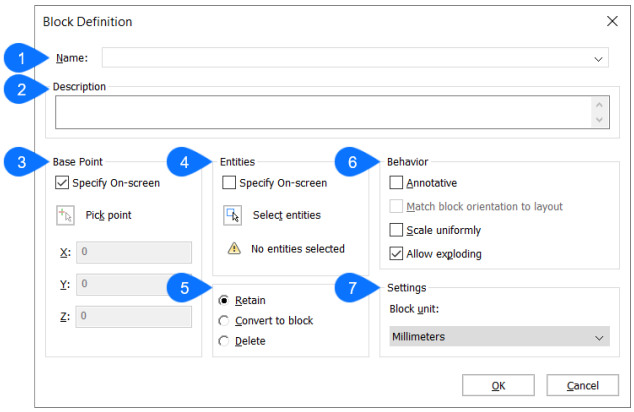
Applies the changes to the block definition, and keeps the dialog box open; choose another block whose attributes you wish to edit.

4.12.22.8 Sync

Applies the changes to all instances of the same block in the drawing.

4.12.23 Block definition dialog box

The block definition dialog box allows you to create a new block definition.



1 Name

2 Description

3 Base Point

4 Entities

5 Block Options

6 Behavior

7 Settings

4.12.23.1 Name

Specifies the name of the block.

4.12.23.2 Description

Provides a description of the block (Optional).

4.12.23.3 Base Point

Modifies the properties of the base point of the block definition. This is the point at which the block will be inserted with the INSERT command.

Specify On-screen
Allows you to define the base point in the model space after you click the **OK** button, if the box is checked.

Pick point
Allows you to pick a point in the drawing for the block's base point. You can also enter the X,Y,Z coordinates in the drawing.

X/Y/Z
Defines the coordinates where the block should be inserted. The Z coordinate is usually optional.

4.12.23.4 Entities

Selects the entities that make up the block.

Specify On-screen
Allows you to select the entities in the model space after you click the **OK** button, if the box is checked.

Nota: Optionally, include Reference Curves in the selection, which allows to automatically align a block during insertion.

Select entities
Selects one or more entities in the drawing.

4.12.23.5 Block Options

Defines how the entities are turned into a block.

Nota: The Convert to block option is the most efficient one.

Retain
Selected entities are retained as entities, and the block definition is created in the drawing.

Convert to block
Selected entities are converted to a block.

Delete
Selected entities are deleted after the block definition is created.

4.12.23.6 Behavior

Annotative
Sets the annotative property of the block. This kind of block should be created when the annotation scale in model or paper space is 1:1. By being annotative, the block automatically scales itself according to the current annotation scale factor.
Choose whether you want the block to follow annotative scaling:

- **On** – the block will scale itself to whatever annotative scale is in effect. The Scale Uniformly option is grayed out (unavailable).
- **Off** - the block follows the scale factor you give the block during the *Insert* command.

Match block orientation to layout
Determines if annotative blocks match the orientation of the layout.

- **On**: annotative blocks display upright, regardless of the orientation of the viewport.
- **Off**: annotative blocks match the orientation of the viewport.

Scale uniformly
Determines if blocks can be scaled non-uniformly. This option is not available to annotatively-scaled blocks.

- **On** - X, Y, and Z scale factors of blocks are the same. This prevents blocks from being distorted.
- **Off** - blocks can be inserted with different X, Y, and Z scale factors. This is useful for objects that can have different dimensions, such as differently sized table tops.

Allow exploding
Determines if users can explode the block after it is inserted. When a block is exploded, it loses its block status, and the individual entities can be edited.

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

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

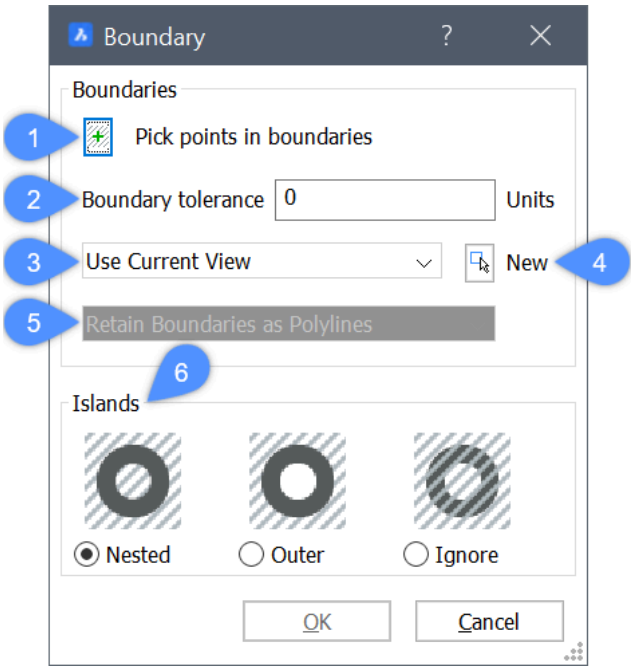
- **Off** - blocks cannot be exploded; you can change this property with the *EXPBLOCKS* command, Blocks section.

4.12.23.7 Settings

Block unit
Scales the block correctly when the block definition is inserted in a drawing whose units are different from the drawing in which the block was created. More specifically, in which the INSUNITS variable differs.

4.12.24 Boundary dialog box

The Boundary dialog box allows you to create closed polylines, defined by surrounding entities.



1 Pick points in boundary

2 Boundary tolerance

3 Boundary set

4 New

5 Retain Boundaries as Polylines

6 Islands

4.12.24.1 Pick points in boundary

It allows you to specify a point inside an enclosed area for which you want to create a boundary. It temporarily dismisses the Boundary dialog box.

4.12.24.2 Boundary tolerance

Specify the largest gap that can exist in the boundary for BricsCAD to consider the area closed.

Nota: When boundary tolerance is 0, no gaps can exist.

4.12.24.3 Boundary set

Specify where BricsCAD should search for entities that make up the boundary.

- **Use Current View**: searches all entities in the current viewport.
- **Use Boundary Set**: searches only the current selection set for possible boundaries.

4.12.24.4 New

It allows you to create a new selection set of entities that make up the boundary. It temporarily dismisses the Boundary dialog box.

4.12.24.5 Retain Boundaries as Polylines [Read only]

Indicates how boundaries are retained.

4.12.24.6 Islands

Specify island detection. An island is an enclosed area inside a boundary.

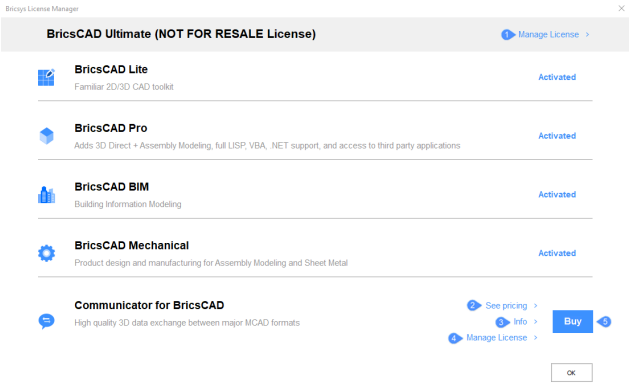
Nested
Treats every island as a boundary.

Outer
Creates a boundary only of the outermost entities.

Ignore
Ignores the innermost areas. A boundary is created between the outermost area and islands.

4.12.25 Bricsys license manager dialog box

The Bricsys License Manager dialog box allows you to activate/deactivate the software licenses for BricsCAD and Communicator.



1 Manage License

2 See pricing

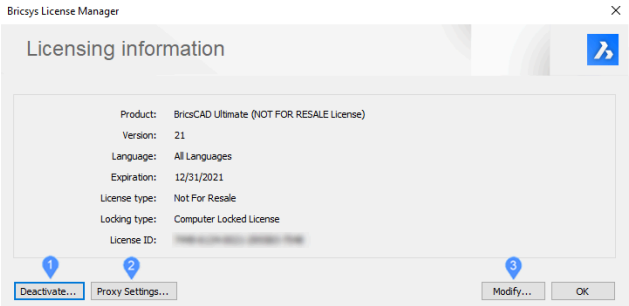
3 Info

4 Manage License for Communicator

5 Buy

4.12.25.1 Manage License

Opens the Bricsys License Manager dialog box.



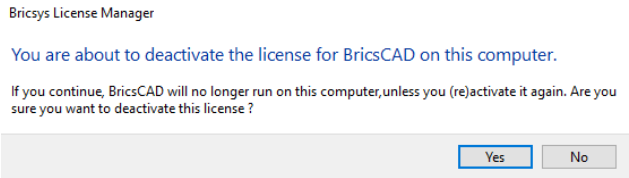
1 Deactivate

2 Proxy Settings...

3 Modify...

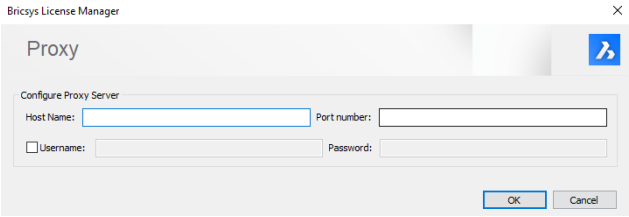
Deactivate

Opens the Bricsys License Manager dialog box for confirming deactivation.



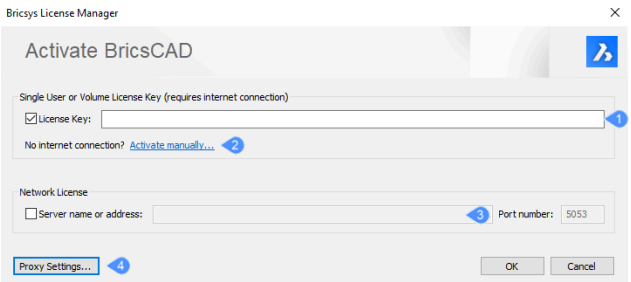
Proxy Settings…

Opens the Bricsys License Manager dialog box where you can configure the proxy server.



Modify…

Opens the Bricsys License Manager dialog box for activating BricsCAD where you can insert your license key, your network license or activate it manually.



1 License Key

2 Activate manually…

3 Network License

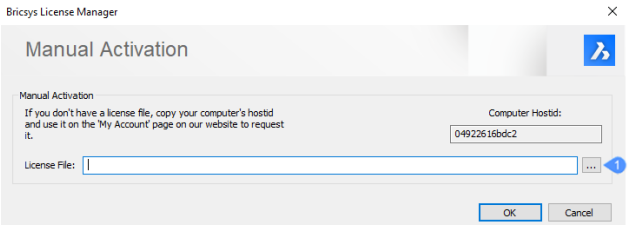
4 Proxy Settings…

License Key

If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually…

Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic



1 Select license file

Select license file

Opens the Select license file dialog box.

Network License

If you are using a network license, enter the host name or the IP address of the network license server.

Proxy Settings…

Opens the Bricsys License Manager dialog box to configure the proxy server.

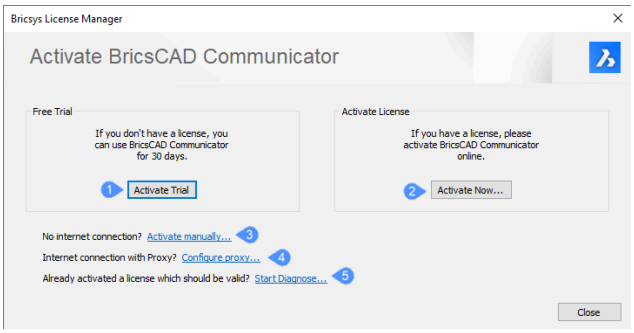
4.12.25.2 See pricing

Opens the Bricsys official website (Bricsys), where you can see the prices for Communicator and 1 year/3 years/lifetime BricsCAD subscription.

4.12.25.3 Info

Opens the Communicator Info dialog box.

4.12.25.4 Manage License for Communicator



1 Activate Trial

2 Activate Now

3 Activate manually...

4 Configure proxy...

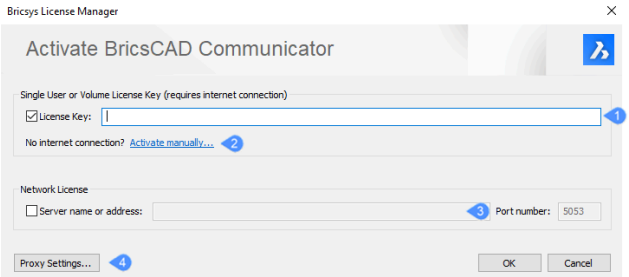
5 Start Diagnose...

Activate Trial

Activates the 30 days free trial of BricsCAD Communicator.

Activate Now

Opens the Activate BricsCAD Communicator dialog box.



1 License Key

2 Activate manually

3 Network License

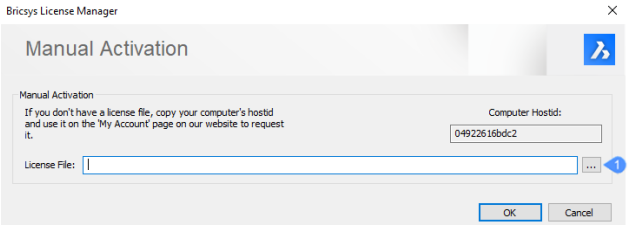
4 Proxy settings

License Key

If you are connected to the internet, you can enter in this field a single user or volume license key.

Activate manually…

Opens the Bricsys License Manager dialog box for Manual Activation, where you can select the license file. The .lic file can be found in this path: C:\ProgramData\Bricsys\BricsCADV21.lic



1 Select license file

Select license file

Opens the Select license file dialog box.

Network License

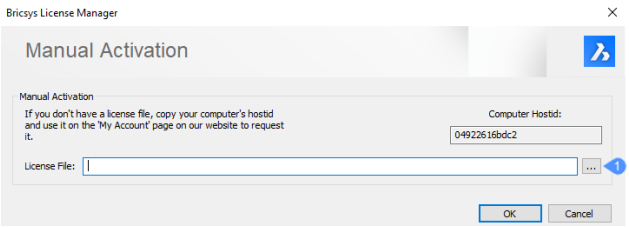
If you are using a network license, enter the host name or the IP address of the network license server.

Proxy Settings…

Opens the Bricsys License Manager dialog box to configure the proxy server.

Activate manually…

Opens the Bricsys License Manager dialog box for activating BricsCAD Communicator.



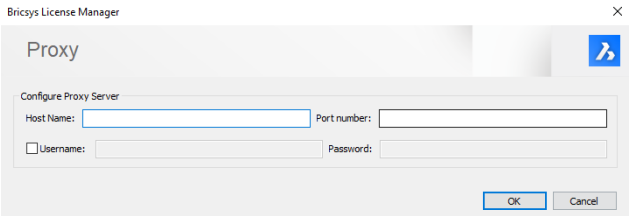
1 Select license file

Select license file

Opens the Select license file dialog box.

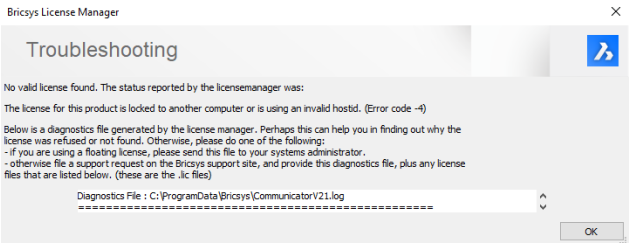
Configure proxy…

Opens the Bricsys License Manager dialog box where you can configure the proxy server.



Start Diagnose…

Opens the Bricsys License Manager dialog box for Troubleshooting.



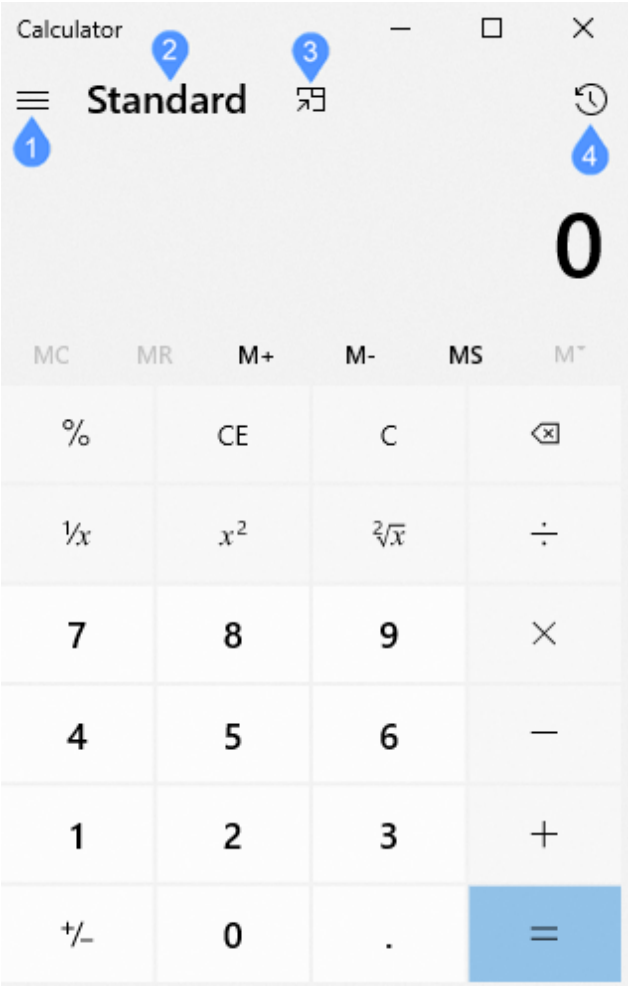
4.12.25.5 Buy

Opens the Bricsys official website (Bricsys) where you can buy or subscribe to BricsCAD.

4.12.26 Calculator dialog box

The Calculator dialog box displays the operating system's software calculator.

When you are using the Standard view, it includes 4 areas.



1 View

2 View's name

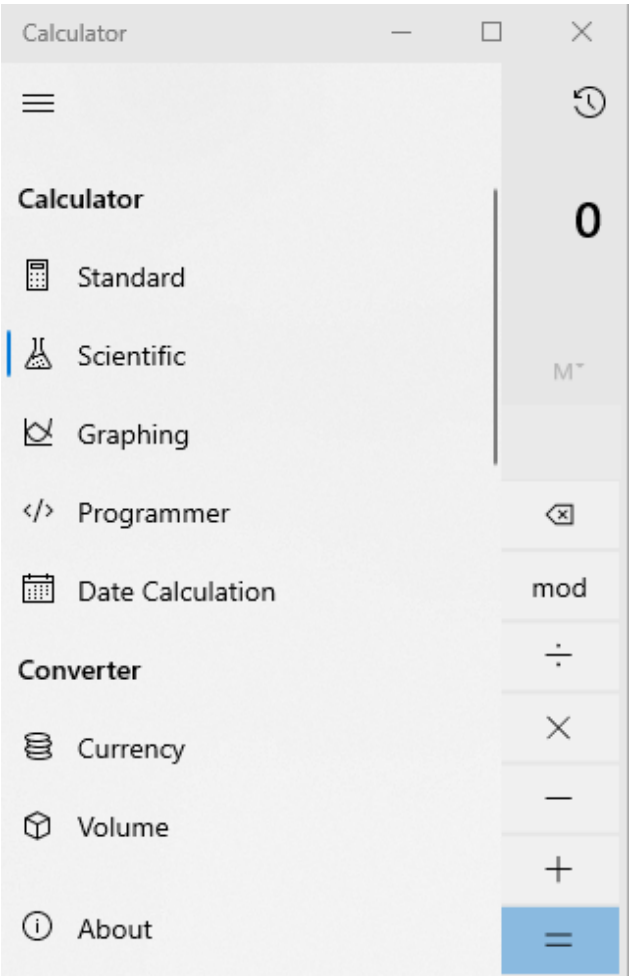
3 Keep on top

4 History

4.12.26.1 View

Displays the possible views you can choose from.

It has 2 categories to choose from: **Calculator** (Standard, Scientific, Graphic, Programmer, Date Calculation) and **Converter** (Currency, Volume, Length, Weight and Mass, Temperature, Energy, Area, Speed, Time, Power, Data, Pressure, Angle).



4.12.26.2 View's name

Displays the current view name.

4.12.26.3 Keep on top

You can also use the keyboard shortcut ALT + Up to access this function.

Note that this function is available only for the Standard view.

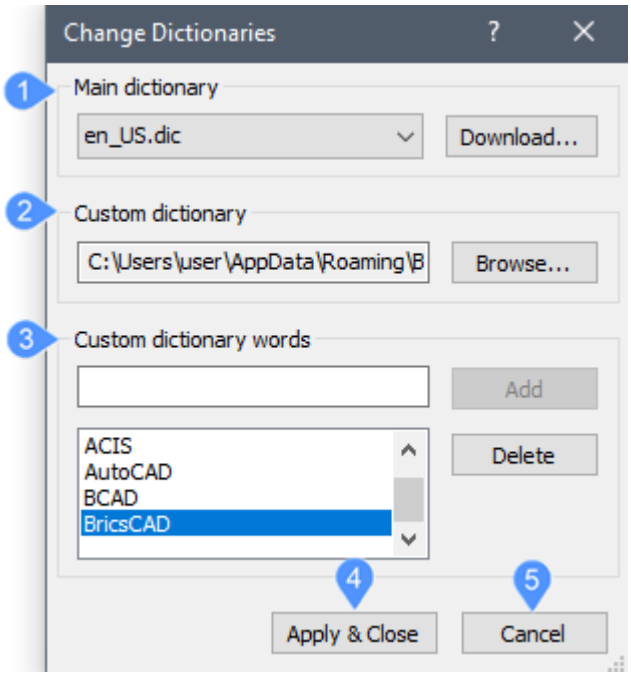
4.12.26.4 History

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

Nota: This function is available only for the Standard and Scientific calculator.

4.12.27 Change dictionaries dialog box

The Change dictionaries dialog box allows you to change dictionaries.



1 Main dictionary

2 Custom dictionary

3 Custom dictionary words

4 Apply & Close

5 Cancel

4.12.27.1 Help

Opens the Bricsys Help article for the SPELL command.

4.12.27.2 Main dictionary

Specifies the main dictionary to use for spell checking.

File name
Specify the dictionary file to use as the main dictionary. Available files are displayed in the drop-down list.

Download
Download additional dictionaries from OpenOffice.org. Opens the Web browser at: https://extensions.openoffice.org/dictionaries

4.12.27.3 Custom dictionary

Specifies a custom dictionary to use for spell checking. Use custom dictionaries for discipline-specific words, such as medical or mechanical terms.

File location
Specifies the file name and location of a custom dictionary to use in addition to the main dictionary.

Browse
Open the Select Custom Dictionary dialog box to select a custom dictionary file.

4.12.27.4 Custom dictionary words

Specifies words to be included in the custom dictionary.

Word field
Allows you to enter a word to be included in the custom dictionary.

Add
Add the word you entered in the word field to the custom dictionary.

Words list
Displays all the words included in the custom dictionary.

Delete
Delete a selected word from the custom dictionary.

4.12.27.5 Apply & Close

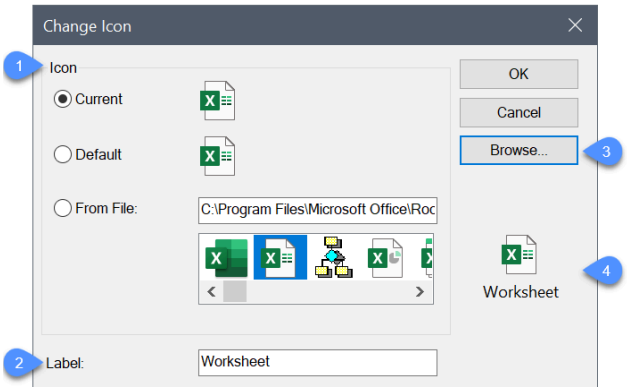
Apply your changes and close the dialog box. The custom dictionary file is only updated with added or deleted words after you choose Apply and Close.

4.12.27.6 Cancel

Close the dialog box without applying changes.

4.12.28 Change Icon dialog box

The Change Icon dialog box allows you to change the icon of an OLE Frame entity before inserting it into the drawing. The icon is related to the document type and it is defined by the source applications.



1 Icon

2 Label

3 Buttons

4 Preview

4.12.28.1 Icon

Specifies an icon to represent an embedded or linked document.

Current
Displays the current icon.

Default
Displays the default icon.

From File
Choose another icon.

4.12.28.2 Label

Optionally edit the label text.

4.12.28.3 Buttons

The Buttons area contains OK, Cancel and the Browse button.

OK
Click OK to accept the changes and to return to the previous dialog box (Insert Object).

Cancel
Click Cancel to discard the changes and to return to the previous dialog box (Insert Object).

Browse
Displays the Browse dialog box (a standard file selection dialog box), in which you can select an icon from a file.

4.12.28.4 Preview

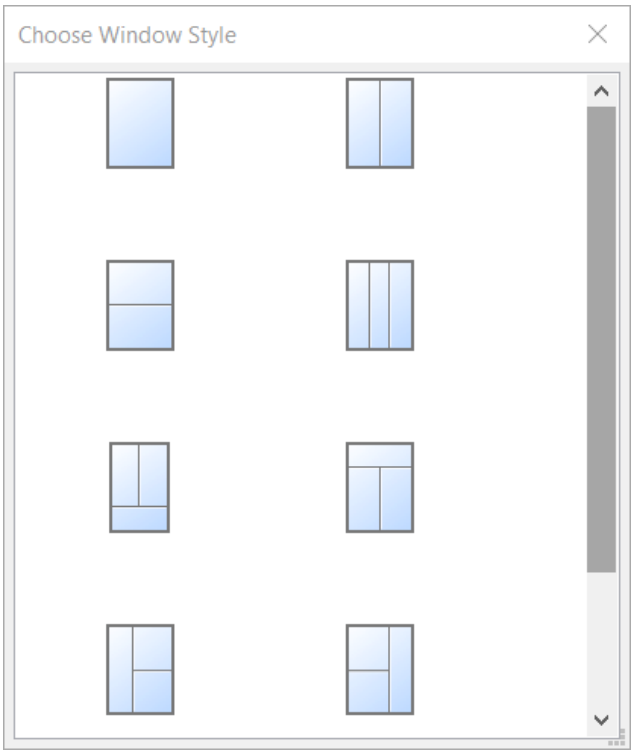
Displays the icon preview.

4.12.29 Choose window style dialog box

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

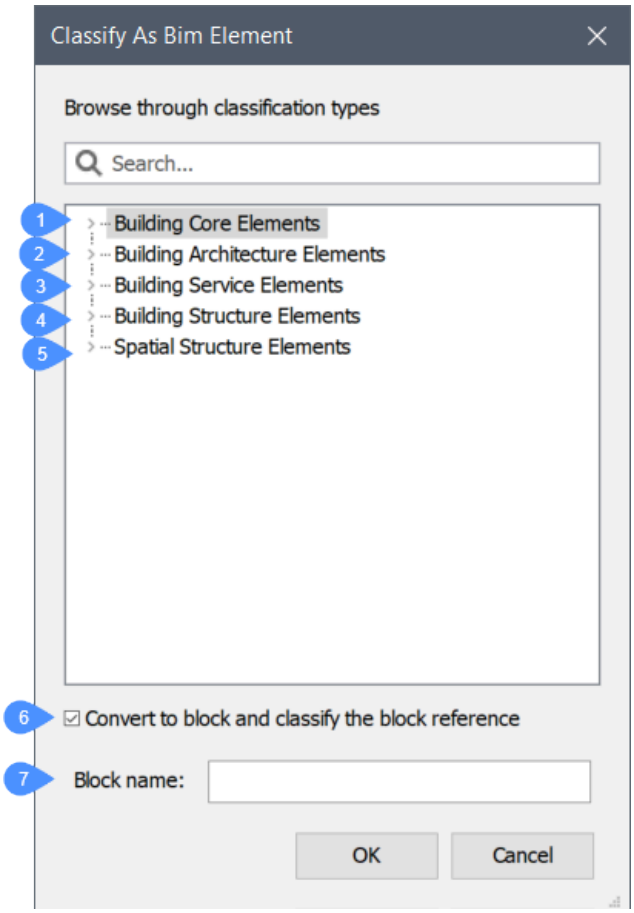
A series of parameters is created, which can be edited in the Parameters section of the Properties panel.

Rectangular windows and openings have a W (Width) and H (Height) parameter.



4.12.30 Classify As Bim Element dialog box

The **Classify As Bim Element** dialog box specifies the types of classification or converts the selection to a block and classifies the block reference.



1 Building Core Elements

2 Building Architecture Elements

3 Building Service Elements

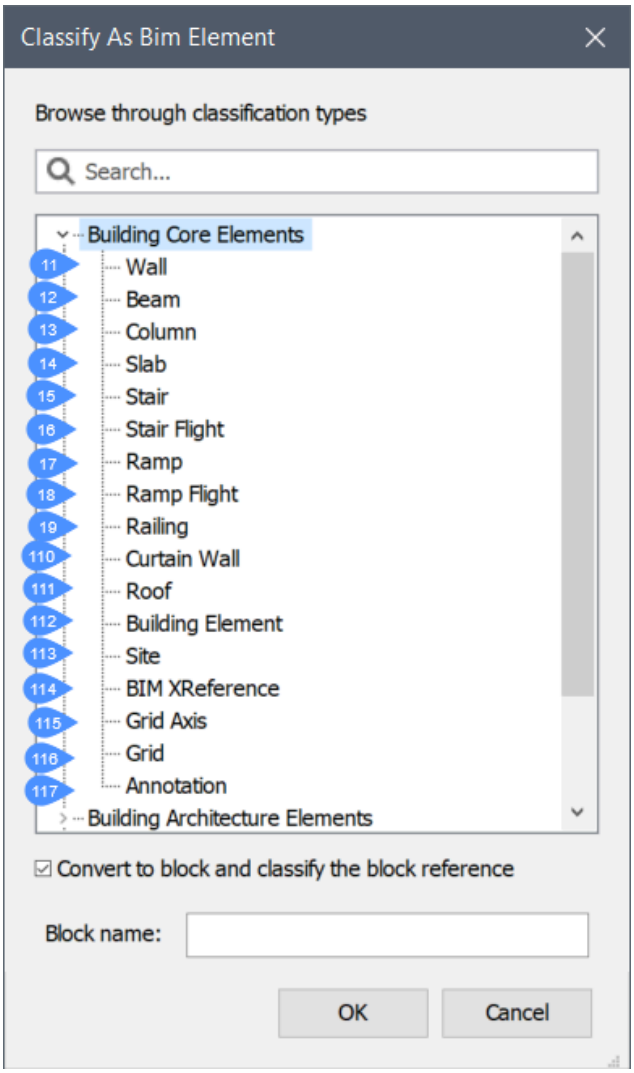
4 Building Structure Elements

5 Spatial Structure Elements

6 Convert to block and classify the block reference

7 Block name

4.12.30.1 Building Core Elements



11. Wall

12. Beam

13. Column

14. Slab

15. Stair

16. Stair Flight

17. Ramp

18. Ramp Flight

19. Railing

110. Curtain Wall

111. Roof

112. Building Element

113. Site

114. BIM XReference

115. Grid Axis

116. Grid

117. Annotation

Wall
Vertical construction that subdivides spaces.
Example: separation wall, bearing wall, interior wall.

Beam
Structural member that carries loads between or beyond points of support, usually narrow in relation to its length and (nearly) horizontal.
Example: I-beam, L-beam, T-beam, double T-beam.

Column
Structural member of slender form, usually vertical, that transmits to its base the forces, primarily in compression, that are applied to it.
Example: Doric column, Ionic column, Corinthian column.

Slab
Component of the construction that normally encloses a space vertically. The slab may provide the lower support (floor) or upper construction (roof slab) in any space in a building.
Example: floor slab, roof slab.

Stair
Construction comprising a succession of horizontal stages (steps or landings) that make it possible to pass on foot to other levels.
Example: stairs, escalator, spiral staircases interrupted with landings.

Stair Flight
Assembly of building components in a single "run" of stair steps (not interrupted by a landing). The stair steps and any stringers are included in this object.
Example: stairs, escalator, spiral staircases NOT interrupted with landings.

Ramp
Inclined way or floor joining 2 surfaces at different levels.
Example: Slope/Ramp interrupted with landings.

Ramp Flight
Inclined slab segment, normally providing a human circulation link between 2 landings, floors or slabs at different elevations.
Example: Slope/Ramp NOT interrupted with landings.

Railing
The railing is a frame assembly adjacent to human circulation spaces and at some space boundaries where it is used instead of walls or to complement walls. It is designed to aid humans, either as an optional physical support, or to prevent injury by falling.
Example: handrail, balustrade, grab bar, guard rail.

Curtain Wall
Exterior wall of a building which is an assembly of components, hung from the edge of the floor/roof structure rather than bearing on a floor.

Roof

Construction enclosing the building from above.

Example: flat roof, gable roof, gambrel roof, hip roof.

Building Element

The building element comprises all elements that are primarily part of the construction of a building.

Example: structural and space separating system like walls, beams, doors.

Site

A defined area of land, possibly covered with water, on which the project construction is to be completed.

BIM XReference

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

Grid Axis

An individual axis that is defined in the context of a design grid. The axis definition is based on a curve of dimensionality 2.

Example: X-axis, Y-axis, Z-axis.

Grid

A planar design grid defined in 3D space used as an aid in locating structural and design elements.

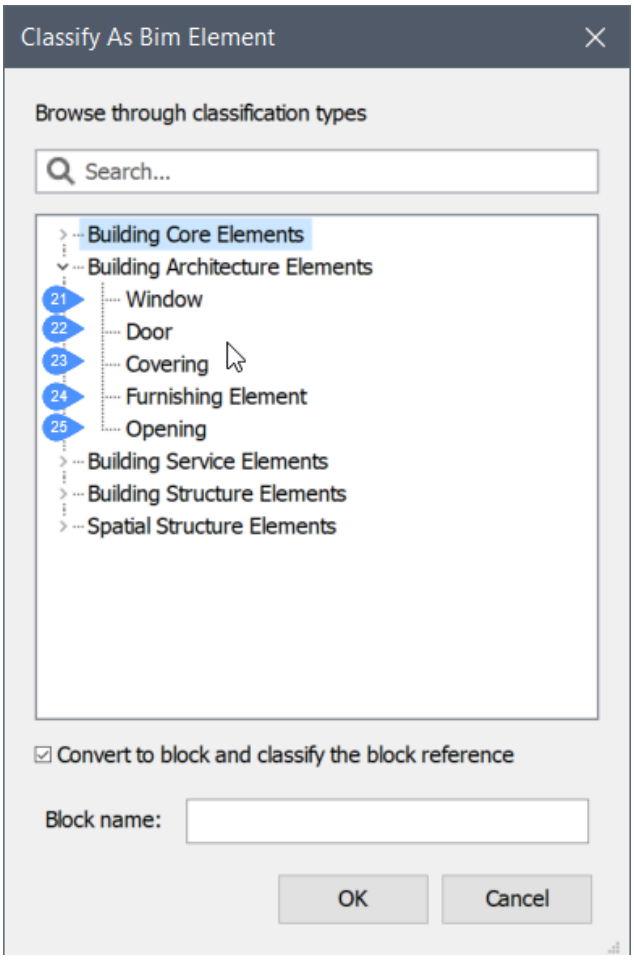
Example: rectangular grid, radial grid, triangular grid.

Annotation

A graphical representation within the geometric context of a project, that adds a note or meaning to the objects of the project.

Example: additional line drawings, text, dimensioning, hatching, ...

4.12.30.2 Building Architecture Elements



21. Window

22. Door

23. Covering

24. Furnishing Element

25. Opening

Window

Construction for closing a (near) vertical opening in a wall or pitched roof that will admit light and may admit fresh air. The window is a building element that is predominately used to provide natural light and fresh air. A window consists of a lining and one or several panels.

Example: skylights, light domes, swinging, pivoting, sliding, or revolving panels and fixed panels.

Door

Construction for closing an opening. The door is a building element that is predominately used to provide controlled access for people and goods. A door consists of a lining and one or several panels.

Example: hinged, pivoted, sliding, revolving, folding doors.

Covering

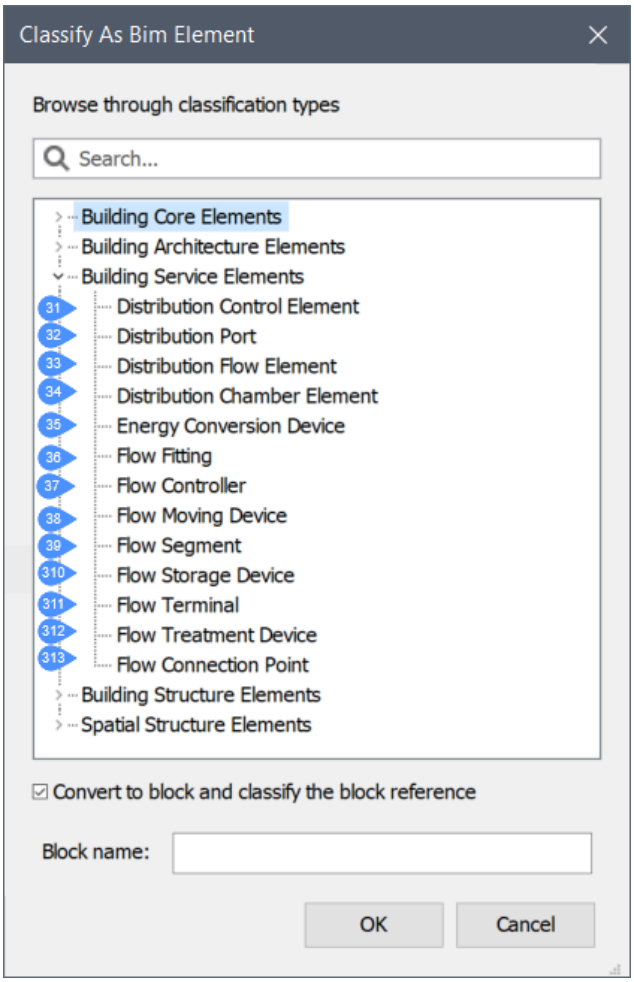
A covering is an element which covers some part of another element and is fully dependent on that other element.

Example: wall claddings, floorings, suspended ceilings, finish trim, base molding.

Furnishing Element
Generalization of all furniture related objects.
Example: closet, table, chair, bed.

Opening
Represents a void within any element that has physical manifestation.
Example: opening for a door, opening for a window.

4.12.30.3 Building Service Elements



31. Distribution Control Element

32. Distribution Port

33. Distribution Flow Element

34. Distribution Chamber Element

35. Energy Conversion Device

36. Flow Fitting

37. Flow Controller

38. Flow Moving Device

39. Flow Segment

310. Flow Storage Device

311. Flow Terminal

312. Flow Treatment Device

313. Flow Connection Point

Distribution Control Element

The occurrence elements of a building automation control system that are used to impart control over elements of a distribution system. It senses elements and measures changes in the controlled variable such as temperature, humidity, pressure, or flow.

Example: thermostat

Distribution Port

The occurrence of a specialized port for use within the context of distribution elements.

Example: A gas-powered hot water heater may have 3 ports: GAS, DOMESTICCOLDWATER, and DOMESTICHOTWATER. The heater is a member of 2 systems (GAS and DOMESTICCOLDWATER) and hosts 1 system (DOMESTICHOTWATER) at the corresponding port.

Distribution Flow Element

The occurrence elements of a distribution system that facilitate the distribution of energy or matter, such as air, water or power.

Example: ducts, pipes, wires, fittings

Distribution Chamber Element

A place at which distribution systems and their constituent elements may be inspected or through which they may travel.

Example: sump, trench

Energy Conversion Device

The occurrence of a device used to perform energy conversion or heat transfer and typically participates in a flow distribution system.

Example: boiler, chiller, or a cooling coil

Flow Fitting

The occurrence of a junction or transition in a flow distribution system.

Example: elbow, tee, a junction box in an electrical distribution system

Flow Controller

The occurrence of elements of a distribution system that are used to regulate flow through a distribution system.

Example: damper, valve, switch, relay

Flow Moving Device

The occurrence of an apparatus used to distribute, circulate or perform conveyance of fluids, including liquids and gases, and typically participates in a flow distribution system.

Example: pump, fan

Flow Segment

A flow segment is a section of a distribution system that typically has only 2 ports.

Example: section of a duct, pipe, conduit, etc.

Flow Storage Device

Device used for the temporary storage of a fluid such as a liquid or a gas, the voltage potential induced by the induced electron flow.

Example: tank, battery

Flow Terminal

The occurrence of a permanently attached element that acts as a terminus or beginning of a distribution system. A terminal is typically a point at which a system interfaces with an external environment.

Example: air outlet, drain, water closet, sink

Flow Treatment Device

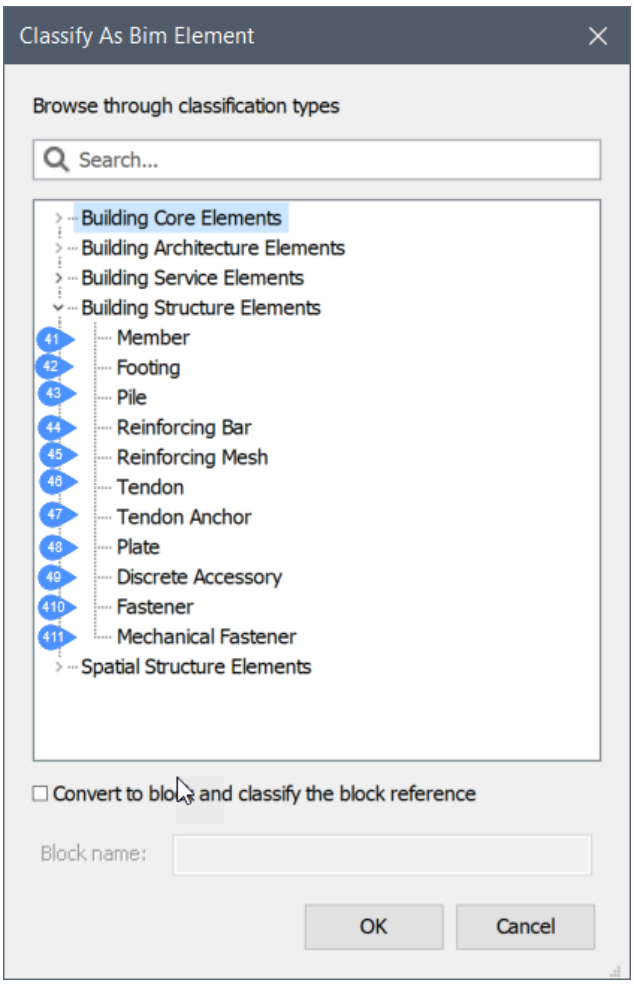
Device used to change the physical properties of the medium, flow treatment types (or the instantiable subtypes) may be exchanged without being already assigned to occurrences.

Example: air, oil or water filter, duct silencer

Flow Connection Point

The connection point is used to describe the geometric constraints that facilitate the physical connection of 2 objects at a point.

4.12.30.4 Building Structure Elements



41. Member

42. Footing

43. Pile

44. Reinforcing Bar

45. Reinforcing Mesh

46. Tendon

47. Tendon Anchor

48. Plate

49. Discrete Accessory

410. Fastener

411. Mechanical Fastener

Member

Structural member designed to carry loads between or beyond points of support. It is not required to be load bearing. The orientation of the member (being horizontal, vertical or sloped) is not relevant to its definition.

Example: diagonal element of a truss construction

Footing

A part of the foundation of a structure that spreads and transmits the load directly to the soil.

Example: wall footing, stepped footing, sloped footing, raft footing

Pile

A slender timber, concrete or steel structural element, driven, jetted or otherwise embedded on end in the ground for the purpose of supporting a load. A pile is also characterized as deep foundation, where the loads are transferred to deeper subsurface layers.

Example: pile foundation

Reinforcing Bar

A steel bar, usually with manufactured deformations in the surface, used in concrete and masonry construction to provide additional strength.

Example: reinforced concrete

Reinforcing Mesh

A series of longitudinal and transverse wires or bars of various gauges, arranged at right angles to each other and welded at all points of intersection.

Example: reinforced concrete floor slab

Tendon

A steel element used to impart pre-stress to concrete when the element is tensioned.

Example: wire, cable, bar, rod, strand

Tendon Anchor

A tendon anchor is the end connection for tendons in pre-stressed or post-tensioned concrete.

Plate

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

Discrete Accessory

A discrete accessory is a representation of different kinds of accessories included in or added to elements.

Example: visible steel corbel (corbel system made from steel components protruding from the master element), corner fixing plate (fixing plate attached to the corner of an element), steel plate (steel plate used as an accessory in a joint)

Fastener

Representations of fixing parts which are used as fasteners to connect or join elements with other elements.

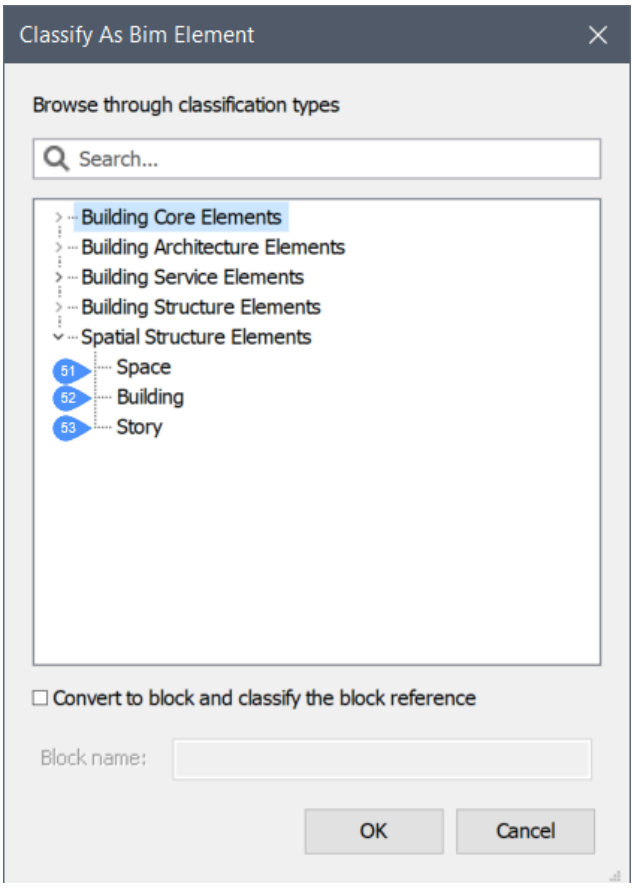
Example: jointing mortar, glue, weld

Mechanical Fastener

Fasteners connecting building elements mechanically.

Example: bolt, screw, nail, nut, washer, rivet

4.12.30.5 Spatial Structure Elements



51. Space

52. Building

53. Story

Space

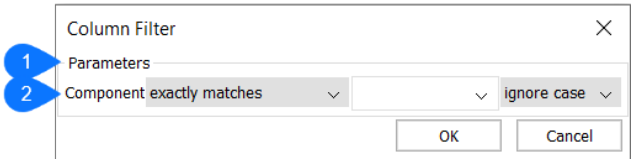
BIM Space entities from enclosed boundaries (3D solids or linear 2D entities).

Building
Buildings in the model.

Story
Story of the building / spatial elements.

4.12.31 Column filter dialog box

The Column Filter dialog box configures the column filter.



1 Parameters

2 Column

4.12.31.1 Parameters

Defines the parameters to use in the column filter.

4.12.31.2 Column type

Displays the name of the column you put a filter on. Selects the type of filter that should be applied to this column and configures it.

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

Nota: For all text filters, you can also specify whether they should be case-sensitive or case-insensitive.

Always accepted
All column values are accepted.

Equal to
Value should be equal to the target one.

Nota: Appears with numerical columns.

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

Nota: Appears with numerical columns.

Less than
Value should be less than the target one.

Nota: Appears with numerical columns.

Greater than
Value should be greater than the target one.

Nota: Appears with numerical columns.

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

Nota: Appears with numerical columns.

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

Nota: Appears with numerical columns.

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

Nota: Appears with numerical columns.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

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

Nota: Appears with columns containing text.

Included in

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

Nota: Appears with columns containing text and numerical columns.

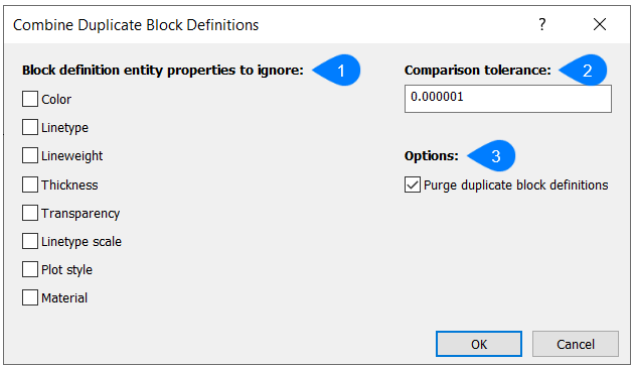
Not included in

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

Nota: Appears with columns containing text and numerical columns.

4.12.32 Combine Duplicate Block Definitions dialog box

The Combine Duplicate Block Definitions dialog box allows you to search the drawing for duplicate block definitions and replace all such block instances by the most recent of these duplicates.



1 Block definition entity properties to ignore

2 Comparison tolerance

3 Options

4.12.32.1 Block definition entity properties to ignore

Specifies the block properties to ignore when searching for duplicate block definitions.

4.12.32.2 Comparison tolerance

Sets the tolerance for overlapping blocks. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

4.12.32.3 Options

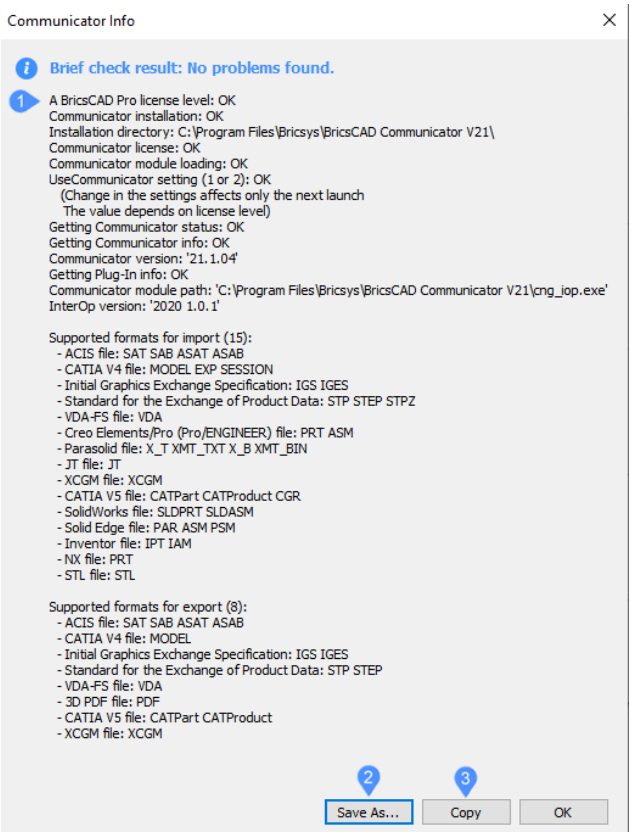
Purge duplicate block definitions

Deletes duplicate block definitions.

4.12.33 Communicator info dialog box

The Communicator Info dialog box checks whether the Communicator add-on is correctly installed and provides a diagnostic report.

The command is available on Windows only.



1 Diagnostic

2 Save As…

3 Copy

4.12.33.1 Diagnostic

Displays a diagnostic report for the Communicator add-on installation, supported formats to import and to export.

4.12.33.2 Save As…

Opens the Save Communicator Info As dialog box.

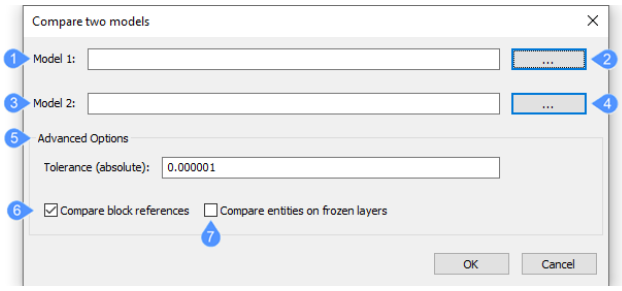
4.12.33.3 Copy

Copies the text displayed in this dialog box to clipboard.

4.12.34 Compare two models dialog box

The Compare Two Models dialog box allows you to perform a geometric comparison of solids and surfaces between two drawing files.

The result is shown in a new drawing and reported in the Mechanical Browser panel.



1 Model 1

2 Select 1st drawing

3 Model 2

4 Select 2nd drawing

5 Advanced Options

6 Compare block reference

7 Compare entities on frozen layers

4.12.34.1 Model 1

Specifies the file name of the first drawing to compare.

4.12.34.2 Select 1st drawing

Opens the Select first drawing for comparison dialog box to choose the drawing you want.

4.12.34.3 Model 2

Specifies the file name of the second drawing to compare.

4.12.34.4 Select 2nd drawing

Opens the Select second drawing for comparison dialog box to choose the drawing you want.

4.12.34.5 Advanced Options

The tolerance (absolute) expresses the threshold value for geometry comparison: determines how close two models must be to be considered unchanged.

4.12.34.6 Compare block reference

Determines if the content of blocks should be compared:

- **Yes** - compares solids and surfaces in blocks.
- **No** - ignores solids and surfaces in blocks.

This command works with regular blocks and inserts of arbitrary depth, which contain solids or surfaces, including assemblies that use block structures.

4.12.34.7 Compare entities on frozen layers

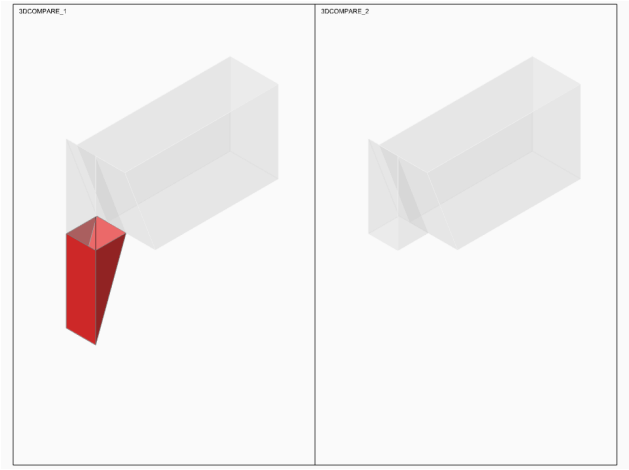
Determines if entities on frozen layers should be compared:

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

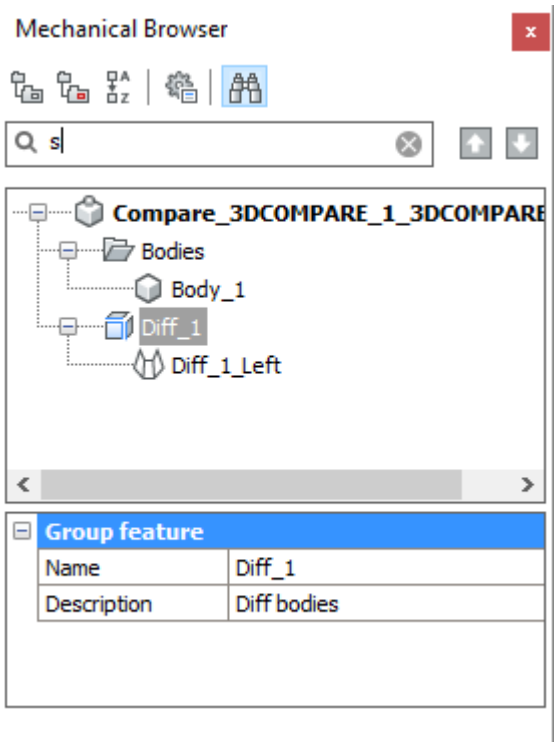
The value of the *3DCOMPAREMODE* system variable controls the display in the viewports of the Comparison layout; in the Settings dialog box, search for "3dcomparemode".

The selected files are attached to a new drawing that is named Compare_<File_1>_<File_2>.dwg

In this drawing, a layout named “Comparison” is created automatically with two viewports, one for each drawing, illustrating the differences between the drawing using colors.

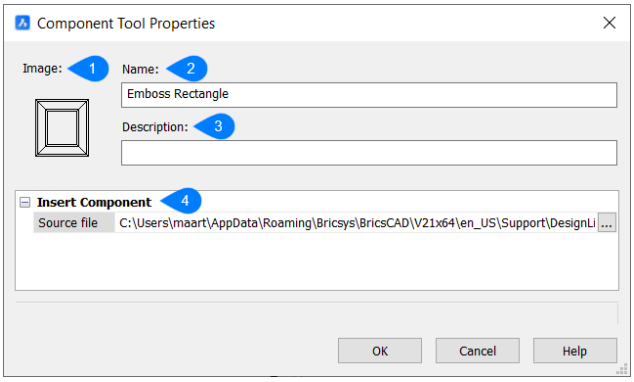


At the same time, the Mechanical Browser panel opens automatically and lists the differences between the 2 drawings.



4.12.35 Component Tool Properties

The Component Tool Properties dialog box allows you to view and change the basic properties of components in the tool palettes panel.



1 Image

2 Name

3 Description

4 Insert Component

4.12.35.1 Image

Displays the image that is used as the icon for the selected component in the tool palettes panel.

4.12.35.2 Name

Change the name of the selected component.

4.12.35.3 Description

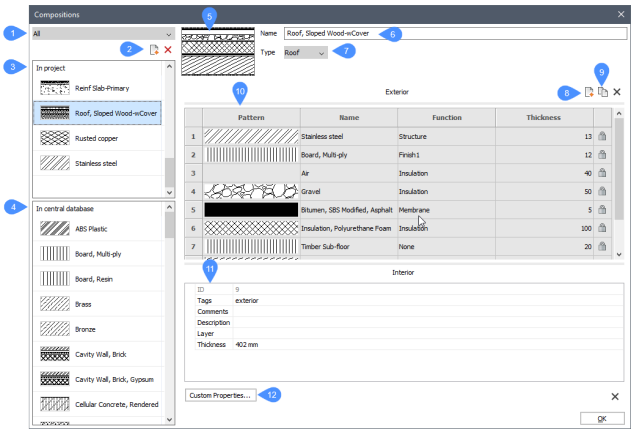
Provide an optional description of the component.

4.12.35.4 Insert Component

The browse icon opens the Choose a File dialog box to allow you to select the source file of the component.

4.12.36 Compositions dialog box

The Compositions dialog box allows you to create, modify and delete compositions.



1 Composition category selection

2 New composition

3 In project

4 In library

5 Preview pane

6 Name

7 Type

8 Add ply

9 Duplicate ply

10 Structure grid

11 Properties grid

12 Custom properties

4.12.36.1 Composition category selection

Composition categories are used to filter the list of the compositions.

Click the drop-down button and choose one of the available categories: All, Generic, Roof, Slab, and Wall.

4.12.36.2 New composition

Click this button to create a new composition.

4.12.36.3 In project

Displays the compositions of the selected category in the Project database.

Right-click on a composition to add a new composition, duplicate, delete or copy to library.

Nota: Compositions that are used in the project cannot be deleted from the project database.

4.12.36.4 In library

Displays the compositions of the selected category in the Library database.

Right-click on a composition to add a new composition, duplicate, delete or copy to project.

4.12.36.5 Preview pane

Displays the preview of the selected composition.

4.12.36.6 Name

Displays the selected composition name. You can change the default name.

4.12.36.7 Type

Displays the category for the selected composition. You can click the drop-down button and select a new category. The available categories are: Generic, Roof, Slab, and Wall.

4.12.36.8 Add ply

Opens the Physical Materials dialog box which allows you to choose a material from the library or project database.

4.12.36.9 Duplicate ply

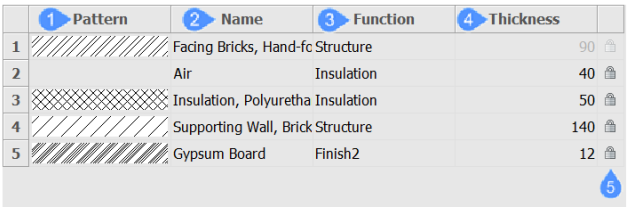
Inserts a copy of the selected ply.

4.12.36.1 Structure grid

0

Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.

The plies of a composition are ordered from Exterior (top) to Interior (bottom). You can drag the ply number on the desired position to change the ply order.



1 Pattern

2 Name

3 Function

4 Thickness

5 Lock/Unlock thickness

Pattern

Displays a preview of the ply. You can double click on the Pattern field and change its appearance from the Physical Materials dialog box.

Name

Displays the name of the ply. You can double click on the Name field and change it from the Physical Materials dialog box.

Function

You can select a function from the drop-down list. The available functions are: None, Structure, Substrate, Insulation, Finish1, Finish2, and Membrane.

Thickness

Click on the Thickness field to modify the thickness of a ply.

Nota: You can only modify the ply thickness if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

Lock/Unlock thickness

You can click the Lock/Unlock button to change its status.

Nota: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

- **Fixed**: all plies have locked thicknesses.
- **Minimal**: the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
- **Free**: the composition contains a single ply that has an unlocked thickness.

Nota: The thickness of a ply can only be unlocked if the Variable Thickness property in the Physical Materials dialog box is set to Yes.

4.12.36.1 Properties grid

1

Displays the properties of the selected composition like tags, comments, description, layer, thickness and custom.

Tags

Select the Tags field and type new tags, separated by commas. Tags are case sensitive.
Click the Browse button next to the tags field to open the Tags dialog box.

Comments

Optionally, you can add comments to the selected composition.

Description

Optionally, you can add a description to the selected composition.

Layer
Optionally, you can assign a layer from the drop-down list to the selected composition.

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

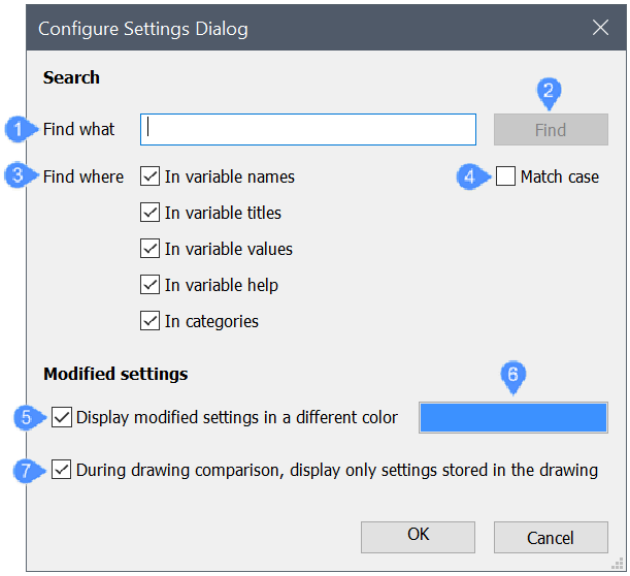
4.12.36.1 Custom properties

2

Opens the Select/Create a Custom Property dialog box. You can add custom properties for compositions.

4.12.37 Configure Settings dialog box

The Configure Settings dialog box allows you to control how BricsCAD searches for variables and how the differences are handled.



1 Find what

2 Find

3 Find where

4 Match case

5 Display modified settings in a different color

6 Select color

7 During drawing comparison, display only settings stored in the drawing

4.12.37.1 Find what

Type the name of the setting you are looking for.

4.12.37.2 Find

Searches in the Settings dialog box for the word you have typed.

4.12.37.3 Find where

Determines which part of the Settings dialog box should be searched.

In variable names
Thick this checkbox to search in variable names.

In variable titles
Thick this checkbox to search in variable titles.

In variable values
Thick this checkbox to search in variable values.

In variable help
Thick this checkbox to search in variable help.

In categories
Thick this checkbox to search in categories.

4.12.37.4 Match case

Thick this checkbox to specify that the capitalization of the searched phrase must match that of the words in the Settings dialog box.

4.12.37.5 Display modified settings in a different color

Thick this checkbox to see the modified settings in a different color.

4.12.37.6 Select color

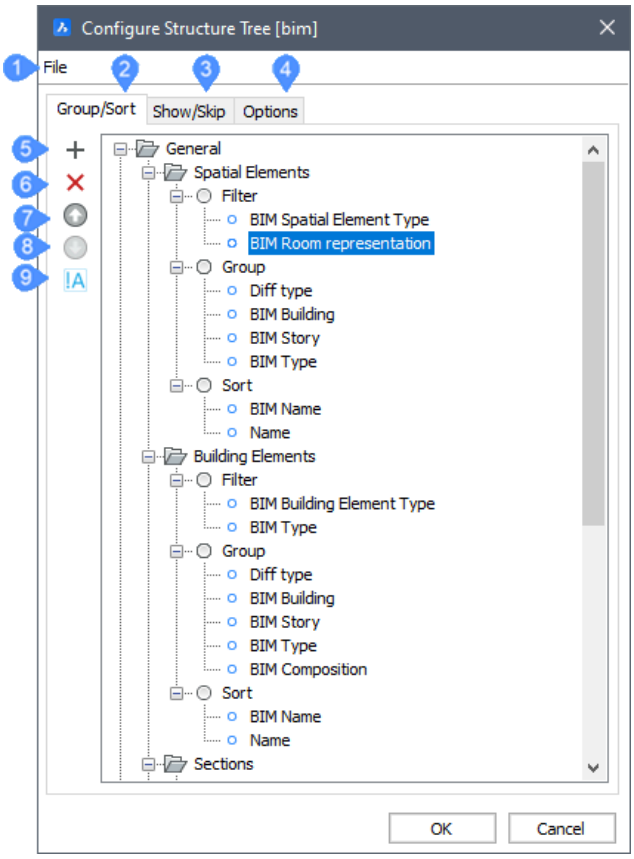
Click this button to select another color.

4.12.37.7 During drawing comparison, display only settings stored in the drawing

Thick this checkbox to see the differences for settings stored in the current drawing and another drawing that you open by running the *DWGCOMPARE* command.

4.12.38 Configure structure tree dialog box

The Configure structure tree dialog box allows you to define the tree structure of the Structural panel.



1 File

2 Group/Sort Tab

3 Show/Skip

4 Options

5 Add

6 Delete

7 Move Up

8 Move Down

9 Toggle negation

4.12.38.1 File

Click the File menu in the dialog box and choose one of the available options on the list:

- Open...: opens the Select Structure Tree Configuration File dialog box to select another *.cst configuration file.
- Save: saves the current configuration and keep the file dialog open.
- Save as: saves the current configuration under a different name and keep the file dialog open.

4.12.38.2 Group/Sort Tab

Add (+) (5)
Adds a new rule.

- Select an existing rule, then click the Add icon (+) or right-click and choose Add Rule in the context menu.
- The new rule is added below the selected rule.
- Select the newly created Rule, then click the name or right-click and choose Rename in the context menu and type a new name for the rule.
- Select the Filter node of the rule, then click the Add icon (+) or right-click and choose Add filter property in the context menu.
- Select the Group node of the rule, then click the Add icon (+) or right-click and choose Add Grouping Property in the context menu.
- Select the Sort node of the rule, then click the Add icon (+) or right-click and choose Add sorting property in the context menu.

The Select Property dialog box displays.

- Select a property, then double-click or click the OK button.
- Optionally, type a search string in the Search field, then click the Up ( ) or Down ( ) arrow buttons to look up the properties containing the search string.
- Repeat the previous step to add more filter / grouping / sorting properties.

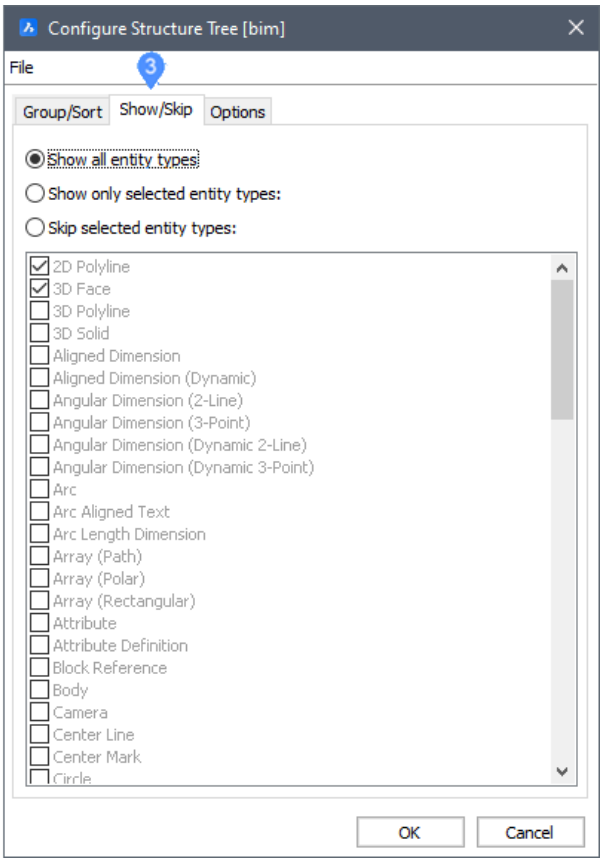
Delete (✕) (6)
Deletes a selected rule or property.

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

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

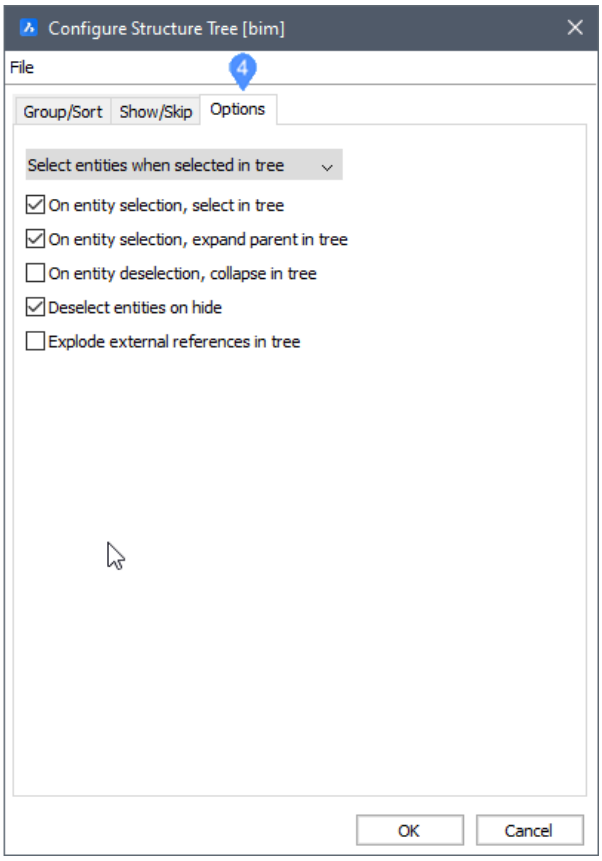
Toggle negation (9)
Toggle negation only for filter properties.

4.12.38.3 Show/Skip Tab



Select an option and then select the entity types to show or to exclude.

4.12.38.4 Options Tab

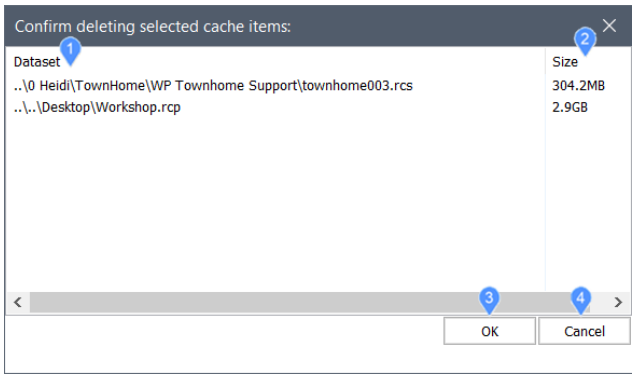


- Choose what happens in the model when entities are selected in the structure tree.
- Select what happens in the structure tree when an entity is selected/deselected in the model.

4.12.39 Confirm deleting selected cache items

TheConfirm deleting selected cache items dialog box enables you to review a list of cached point cloud files that were selected to be deleted.

This gives you a final opportunity to cancel the operation before permanently removing them from the cache.



1 Dataset

2 Size

3 OK

4 Cancel

4.12.39.1 Dataset

Displays a list of cached point cloud files that you selected to delete from the cache. It includes their names and locations.

4.12.39.2 Size

Displays the amount of cache used for each of the selected point cloud files.

4.12.39.3 OK

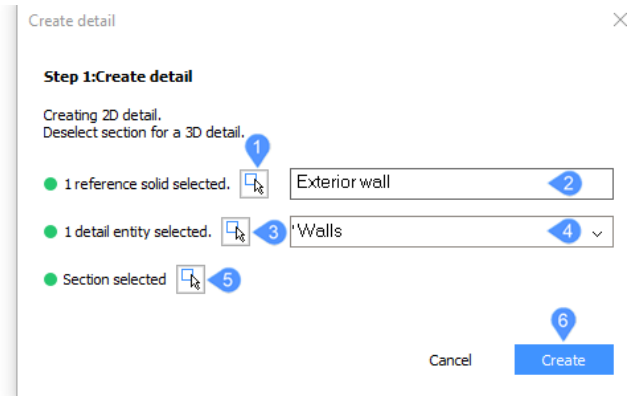
Deletes the point cloud files from the cache. Any drawings that reference these files will be unable to find them.

4.12.39.4 Cancel

Closes the dialog box without deleting point cloud files from the cache.

4.12.40 Create detail dialog box

The Create Detail dialog box allows you to create details and save them in the library.



1 Pick reference solids

2 Detail name

3 Pick detail entities

4 Create or choose category

5 Pick section

6 Create

4.12.40.1 Pick reference solids

Select reference solids, press Enter to display the dialog box and continue the procedure. These will not be copied but used to match other locations.

4.12.40.2 Detail name

Type the name you want to give to the detail.

4.12.40.3 Pick detail entities

Select detail objects (such as solids, block references, faces and edges) to be copied.

4.12.40.4 Create or choose category

Select a category in the drop-down list or type a name to create a new category.

4.12.40.5 Pick section

Select a BIM section.

Detail categories are subfolders of the Details folder, which is defined by the DETAILSPATH system variable. By default, this is *C:\ProgramData\Bricsys\Details*.

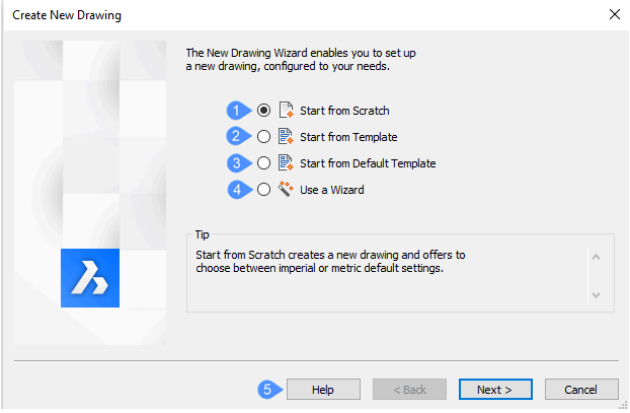
4.12.40.6 Create

Accept/Reject the changes or inflate first.

Displays the Save Detail dialog box, where you will add tags to the details in order to find it easier in the Details panel.

4.12.41 Create new drawing dialog box

The Create New Drawing dialog box allows you to create new drawings by several methods with the New Drawing Wizard.



1 Start from Scratch

2 Start from Template

3 Start from Default Template

4 Use a Wizard

5 Help

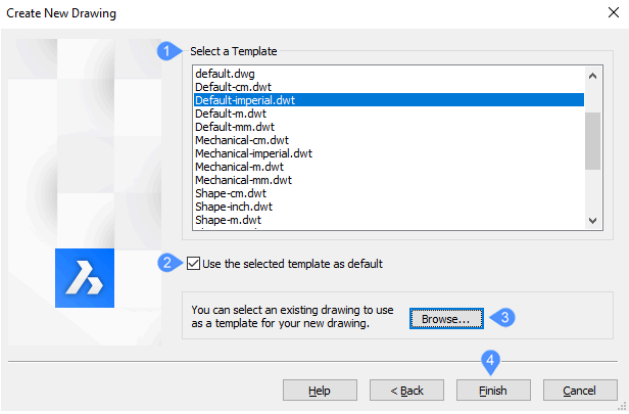
4.12.41.1 Start from Scratch

Starts the new drawing using the default template file specified by the BASEFILE system variable. Click the **Next** button and choose a measurement system:

- **Imperial (feet and inches)** - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- **Metric** - new drawing is based on the default-mm.dwt file and so uses metric measurements.

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

4.12.41.2 Start from Template



1 Select a Template

2 Use default template

3 Browse…

4 Finish

Select a Template

Allows you to choose a file to act as the template. From the Template list, choose one of the DWT file names. These files are stored in the folder specified by the TemplatePath variable; the default location is *C:\Users\useid\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates*.

Use default template

Turn on **Use the selected template as default** option when you want to use this file as the template from now on. If so, then the program updates the value of the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

Browse…

Opens the Select a template file dialog box from where you can select the following kind of file type for a template: DWG - drawing files, DWT - template files, DXF - drawing interchange files.

Finish

Click the Finish button after selecting a Template from the list or from the dialog box.

4.12.41.3 Start from Default Template

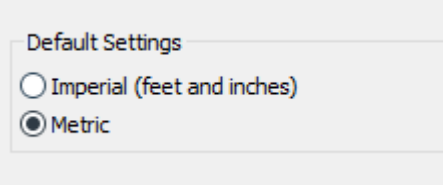
Starts new drawings with the default template defined by the BaseFile variable. After you click the **Finish** button, the program opens a new drawing whose content depends on the template file you have selected.

4.12.41.4 Use a Wizard

Starts the new drawing by stepping through a wizard that guides you along the way:

1 Click the **Next** button.

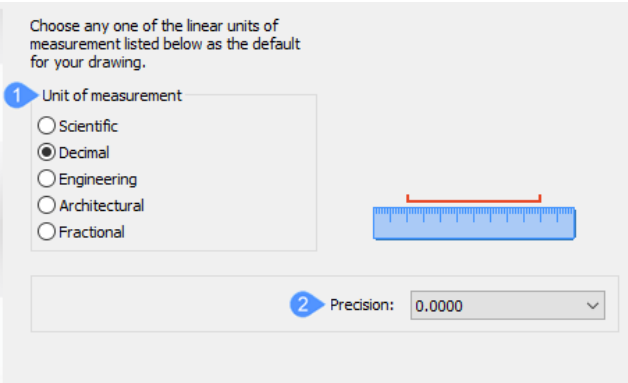
2 Choose a measurement system:



- Imperial (feet and inches) - new drawing is based on the default-imperial.dwt file and so uses Imperial measurements.
- Metric - new drawing is based on the default-mm.dwt file and so uses metric measurements.

3 Click the **Next** button.

4 Select a linear unit of measurement and display precision.



a Unit of Measurement

b Precision

Unit of Measurement

Choose one from the list. If you are unsure, choose Decimal; you can always change the units any time later with the UNITS command.

Unit Name	Measures	Example Display
Architectural	Feet, inches, fractional inches	4'-6 1/16"
Engineering	Feet, inches, decimal inches	4'-6.0625"
Fractional	Inches, fractional inches	54 1/6"
Decimal	(default) Units and decimal units	2128.4449
Scientific	Units, decimal units, exponents	2.1284E+03

Precision

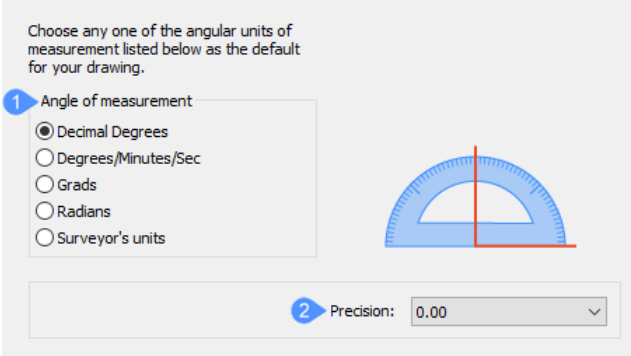
Click the drop-down list and choose a level of precision. If you are unsure, choose the defaults, as you can always change the precision any time later with the Units command.

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

Nota:

1 Click the **Next** button.

2 Select an angular unit of measurement and display precision.



a Angle of measurement

b Precision

Angle of Measurement

Choose one form the list. If you are unsure, choose Decimal/Degrees; you can always change the units any time later with the Units command.

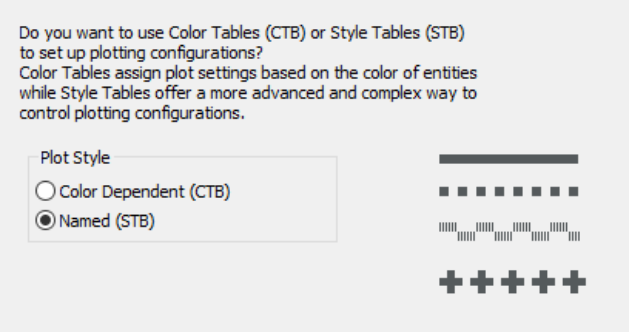
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds	Degrees,minutes,seconds,decimal secs	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units	North/South degrees towards East/West	N12d34'56"E

Precision

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

1 Click the **Next** button.

2 Determine which kinds of plot style the drawing should employ:

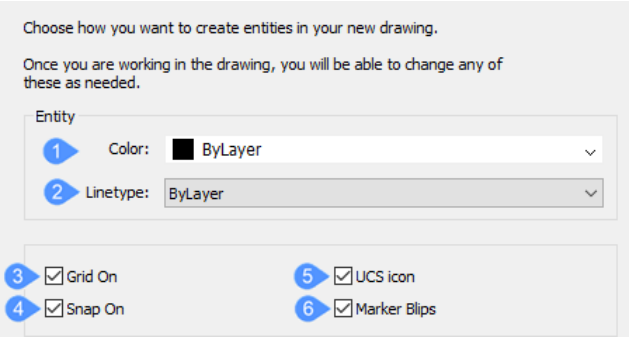


- **Color Dependent (CTB)** - these plot styles are simpler to employ, but less flexible.
- **Named (STB)** - plot styles are more flexible, but complex, because they control every aspect of every entity property during plotting.

If you are unsure, choose Color-dependent.

3 Click the **Next** button.

4 Choose the default working color, the linetype and the visual aids you want to be displayed by the drawing:



a Color
b Linetype
c Grid On
d Snap On
e UCS icon
f Marker Blips

Color

You can choose any color from the drop-down list. However, this is not recommended. Choose ByLayer so that layers control the colors of entities. See COLOR and LAYER commands.

Linetype

You can choose any linetype from the drop-down list, this again is not recommended. Choose ByLayer, for the same reason. See LINETYPE command.

Grid On

When on, displays a grid of dots (or lines) that helps you see the unit spacing, as well as drawing extents, such as every 1m; see the GRID command. It is recommended you turn this on.

Snap On

When on, sets the movement increment of the cursor, such as every 0.5m; see SNAP command. It is recommended you turn this on.

UCS icon

When on, displays the UCS icon for showing the orientation of the x, y, and z axes; see the UCSICON command. It is recommended that you turn this on for 3D drawings, but turn it off for 2D ones.

Marker Blips

When on, draws a tiny + each time you click in the drawing; see the BLIPMODE command. It is recommended you leave this one off, as it can turn the screen messy.

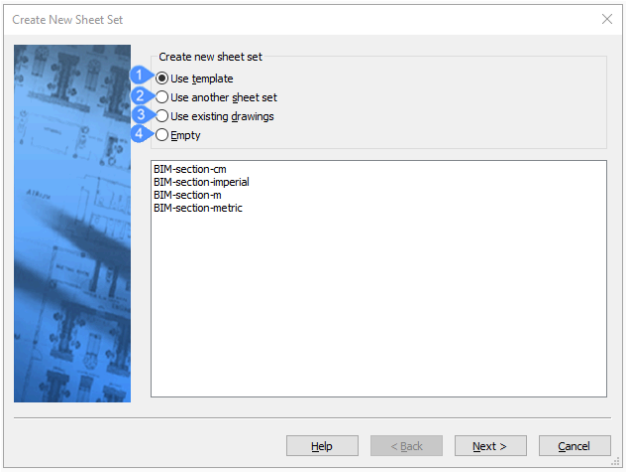
After you click the **Finish** button, the program opens a new drawing whose content depends on the default template file and the options you choose during the wizard.

4.12.41.5 Help

Opens the Bricsys Help article about the NEWWIZ command.

4.12.42 Create new sheet set dialog box

The Create New Sheet Set dialog box provides 4 methods to create a new sheet set.



1 Use template

2 Use another sheet set

3 Use existing drawings

4 Empty

4.12.42.1 Use template

Use template

Create a new sheet set from a template.

Sheet set templates are stored as DST files in the folder set by the SheetSetTemplatePath preference (default path is *C:\Users\<UserName>\AppData\Local\Bricsys\BricsCAD\V[Version]\en_US\Templates*)

1 Select a template sheet set from the list shown in the dialog box.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Description

1 Optional, type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

4.12.42.2 Use another sheet set

Creates a new sheet set from the settings in an existing sheet set.

Select sheetset to copy

1 Click **Browse** and select an existing sheetset file from Select a Sheet Set file dialog box.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder that stores the new file.

Description

1 Optional, type a description for the sheet set file.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

4.12.42.3 Use existing drawings

Create a new sheet set from an existing set of drawings.

1 Select **Use existing drawings**.

2 Click **Next**.

Select Folder

1 Select a folder from the Choose a folder dialog box where the new sheet set will be stored.

Nota: Notice that the dialog box fills with names of drawings and their layouts.

2 Choose among the options (it is recommended to keep both options turned on).

Create subsets from folder structure

Uses the folders in the main folder to determine how the sheet set is structured.

Prefix sheet titles with drawing file names

1 Add the drawing file's name to the start of the sheet set titles, which makes it easier to track the source of the files.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder.

Description

Optional, type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the Select Drawing Template dialog box.

2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

4.12.42.4 Empty

Creates a new sheet set from scratch.

1 Select **Empty**.

2 Click **Next**.

Save sheet set file as

1 Type a name for the sheet set file.

2 Click **Browse** to choose a destination folder in which to save the sheet set and save it.

Importante: You can not progress until you select a folder.

Description

Optionally, type a description for the sheet set file.

Default drawing template for new sheets

1 Select a template sheet set from the *Select Drawing Template* dialog box.

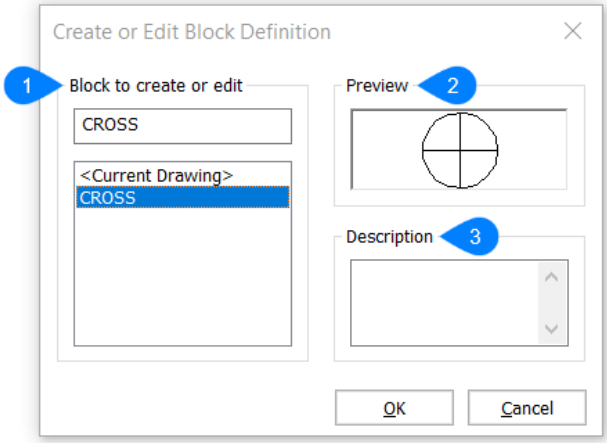
2 Click **Next**.

Sheet Set Preview

Review the summary of the sheet set that will be created, and then click Finish to create the new sheet set file. The program opens the Sheet Set panel with the new sheet set.

4.12.43 Create or edit block definition dialog box

The Create or Edit Block Definition dialog box allows you to select an existing block definition to edit, or to enter a new name to create a new block definition.



1 Block to create or edit

2 Preview

3 Description

4.12.43.1 Block to create or edit

Specifies the name of the block you want to create or edit. To edit a block, you can select a block from the list below with the blocks available in the drawing.

4.12.43.2 Preview

Provides a preview of the selected block.

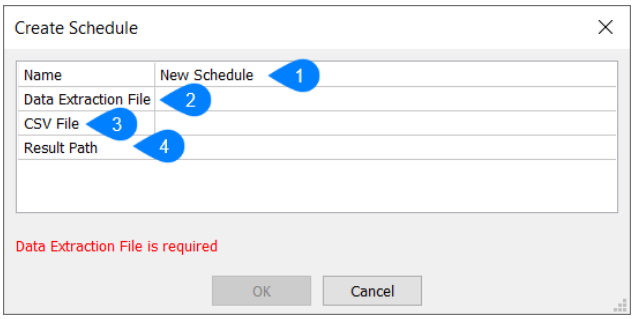
4.12.43.3 Description

Displays the description of the selected block.

Click **OK**, to open the Block Editor.

4.12.44 Create Schedule dialog box

Create a new schedule in the project browser.



1 Name

2 Data Extraction File

3 CSV File

4 Result Path

4.12.44.1 Name

Specifies the name of the new schedule.

4.12.44.2 Data Extraction File

Selects the data extraction file that will be used. Opens the Select Data Extraction Definition file dialog box where you can select a file.

4.12.44.3 CSV File

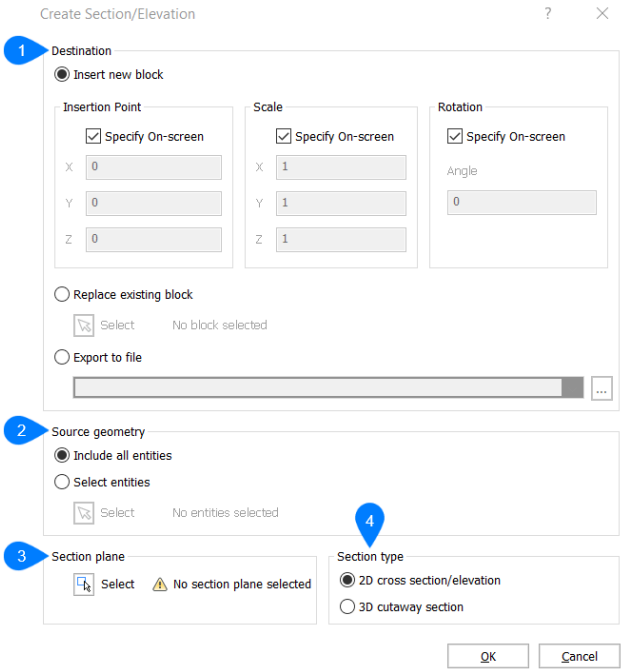
Defines the CSV file that will be used. Opens the Select Delimiter-separated format file dialog box where you can select a file.

4.12.44.4 Result Path

Defines the path where the resulting drawing will be saved. Opens the Select Drawing file dialog box where you can select a drawing to save the result.

4.12.45 Create section/elevation dialog box

The Create Section/Elevation dialog box saves section planes as blocks inserted in the current drawing or saved to disk as DWG files.



1 Destination

2 Source geometry

3 Section plane

4 Section type

4.12.45.1 Destination

Defines where the block will be placed in the drawing.

Insert new block
Inserts the created section as a new block in the drawing. The insertion point, scale and rotation can be specified.

Insertion point
Allows to specify the insertion point in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z
Defines the coordinates where the block should be inserted, only available if the Specify on-screen button is checked off.

Scale
Allows to specify the scale factor in the model space after you click the OK button, if the Specify on-screen button is checked.

X/Y/Z
Defines the individual X,Y,Z direction scale factors, only available if the Specify on-screen button is checked off.

Rotation
Allows to specify the rotation angle in the model space after you click the OK button, if the Specify on-screen button is checked.

Angle
Defines the rotation angle of the inserted section/elevation.

Replace existing block
Replaces an existing block in the drawing. This block will be replaced by the created section

Select
Allows to select the block to replace in the model space.

Export to file
Opens the Select export file dialog box. Here you can select a file to export the created section to.

4.12.45.2 Source geometry

Specifies the entities that will be included from the drawing.

Include all entities
Selects all 3D geometry in the drawing (includes 3D solids, 3D surfaces, polyface meshes, and 3D faces).

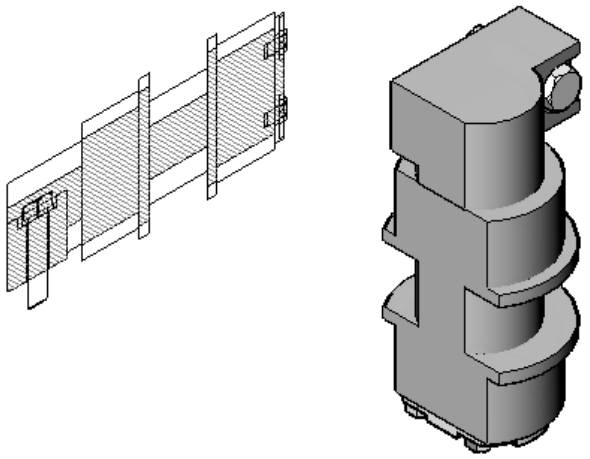
Select entities
Allows to select the entities to be included in the model space.

4.12.45.3 Section plane

Selects the section plane to be converted to a block.

4.12.45.4 Section type

Determines the kind of section plane block to create.

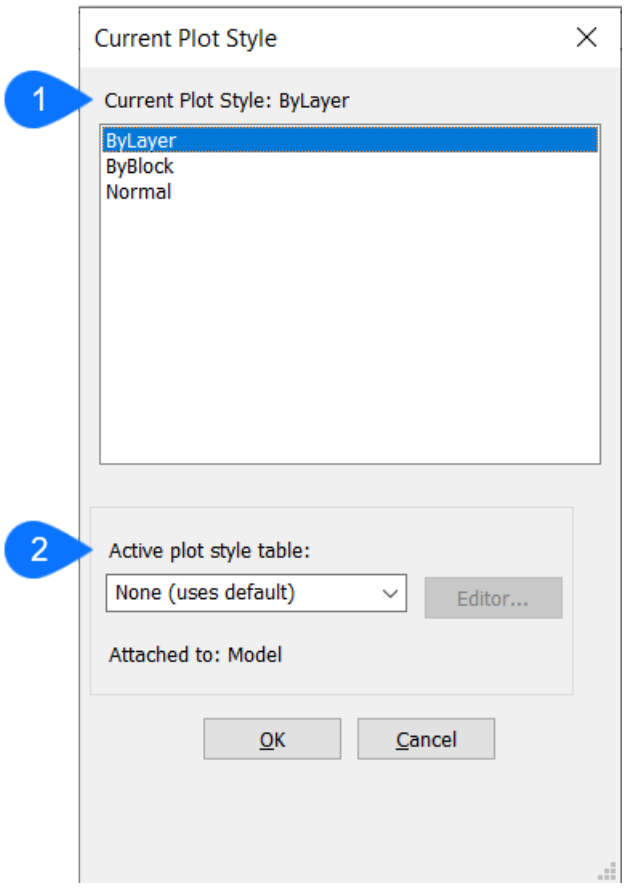


2D cross section/elevation
Uses the 2D section settings defined by the SectionPlaneSettings dialog box to create the block.

3D cutaway section
Uses the 3D section settings defined by the SectionPlaneSettings dialog box to create the block.

4.12.46 Current Plot Style dialog box

Sets the current plot style.



1 Current Plot Style

2 Active plot style table

4.12.46.1 Current Plot Style

Sets the current plot style for the drawing. You can choose between:

- **ByLayer**: uses the plot style of the layer of the entity.
- **ByBlock**: uses the plot style of the block the entity is assigned to.
- **Normal**: uses the normal plot style. The default properties of the entity are used.

4.12.46.2 Active plot style table

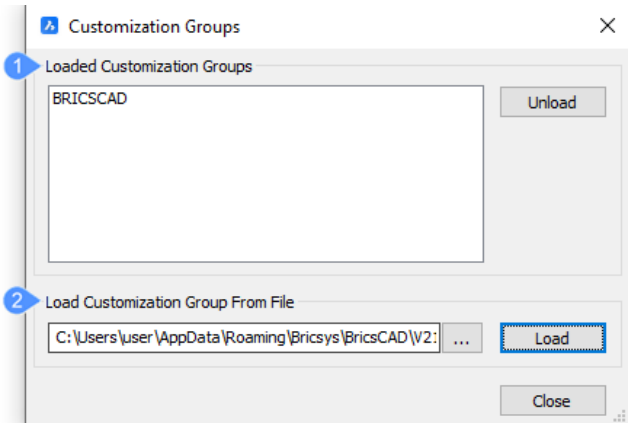
Lists the names of STB files (short for "style table"). To edit the chosen plot style table, click 'Editor...'. The Plot Style Table Editor dialog box will be displayed.

4.12.47 Customization groups dialog box

The **Customizations Groups** Dialog Box allows you to load or to unload a partial CUI file.

When you install BricsCAD the default CUI files are copied to the **Support** folder of your Roamable root folder. (e.g. C:\Users\<user_name>\AppData\Roaming\Bricsys\Bricscad\Vxx.x\en_US\Support).

When you launch BricsCAD the first time the default.cui file is loaded automatically.



1 Loaded Customization Groups

2 Load Customization Group from File

4.12.47.1 Loaded Customization Groups

A list with the names of the customization files already loaded is displayed.

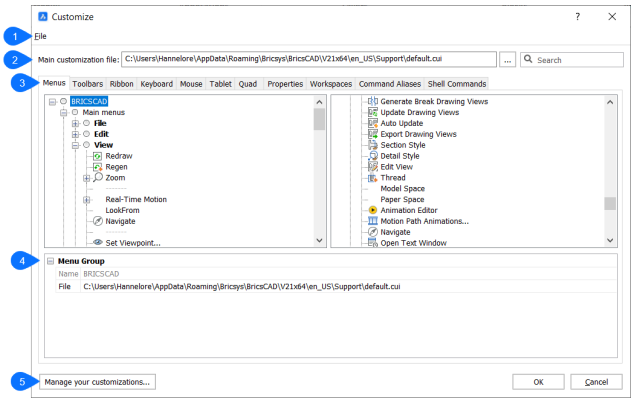
A CUI file can be unloaded by selecting it from the list and clicking the **Unload** button.

4.12.47.2 Load Customization Group from File

Click the browse button to select a partial CUI file and click the **Load** button to load it.

4.12.48 Customize dialog box

The Customize dialog box allows you to customize the user interface of BricsCAD.



1 File

2 Main customization file

3 Tab options

4 Menu Group

5 Manage your customizations

4.12.48.1 File

Shows a menu to select customization files or import workspaces.

Load main CUI file...
Opens the Select main CUI file dialog box to select a customization file. See the CUILOAD command.

Save main CUI file as...
Opens the Save main CUI file as dialog box to save the current main CUI file.

Nota: You can save the main CUI file under a different name to copy the adapted main CUI file.

Load Partial CUI file...
Opens the Choose a customization file dialog box to open a partial CUI file.

Nota: See chapter **Loading a partial CUI file** in **Loading and unloading CUI files** for a detailed explanation about the partial CUI files.

Create new partial CUI file...
Opens the Create a Customization File dialog box to create an empty CUI file.

Import workspace(s)…
Opens the Choose a Customization File dialog box to load one or more workspaces from another CUI file.

4.12.48.2 Main customization file

Specifies the CUI file that defines customization of the menu, toolbar, and other elements of the user interface.

Nota: Aliases and shell commands are defined in a PGP file.

Browse
Opens the Select main CUI file dialog box to load a different main CUI file.

Search
Searches the Available Tools list for command names.

4.12.48.3 Tab options

Menus
Adds, edits, moves and deletes menu items.
When right clicking a menu, menu item or submenu in the Menus tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append item or Insert item from the context menu, the Add menu item dialog box displays.

Menu Group/Menu/Menu Item/Submenu
Displays the options of the selected menu, submenu, menu item or separator.

Title
Specifies the name that is displayed by the menu, menu item or submenu.

Nota: You can prefix a letter with '&' to create an Alt-key shortcut. For example: &Line.

ID
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the menu, menu item or submenu. The browse icon opens the 'Edit Aliases' dialog box where you can edit, create or delete aliases.

Diesel
Specifies the Diesel code to be employed by the menu, menu item or submenu.

Tool ID
Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

Image
Specifies the image to display for the menu, menu item or submenu. The browse icon opens the Tool Image dialog box. There are five options to select an image.

Toolbars
Adds, Moves and deletes toolbar items.
When right clicking a toolbar, tool, control, flyout or separator in the Toolbars tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append tool or Insert tool from the context menu, the Add tool dialog box displays. For options description see the Add menu item dialog box.

Menu Group/Toolbar (Button)
Displays the options of the selected toolbar, tool, control, flyout or separator.

Title
Specifies the name that is displayed by the tooltip.

ID
Unique identifier for each Menu item. (The ID is assigned by BricsCAD).

Alias
Defines the aliases of the selected item. The browse icon opens the 'Edit Aliases' dialog box where you can edit, create or delete aliases.

Position

Default Display
Determines whether the selected item will be added to the workspace.

Rows
Specifies the number of rows for an undocked toolbar.

Xval Yval
Specifies the distance in pixels measured from the top left corner of the screen to the toolbar. It applies to undocked toolbars.
The values are taken from the X,Y options of the toolbar as they are set in the Workspace tab.

Diesel
Specifies the Diesel code to be employed by the selected item.

Tool ID
Identifies the toolbar item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

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

Ribbon
Manages the ribbons and/or adds panels to a ribbon tab.
When right clicking a ribbon tab or panel in the Ribbon tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add launcher, Insert toggle button or Insert command button from the context menu, the Add ribbon dialog launcher dialog box, Add ribbon toggle button or add ribbon command button displays.

- Select available tool: assigns an existing command to the new item. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new item. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop down list
- Title : specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

Menu Group/Ribbon Tab/Ribbon Panel Reference
Displays the options of the selected ribbon tab or panel.

ID
Identifies the item in the CUI file. (The ID is assigned by BricsCAD).

Collapse
Allows to choose if the panel will collapse automatically or if it will never collapse.

Label
Specifies the name of the ribbon tab or panel.

Title
Specifies the name that is displayed by the selected item.

Key Tip

Keyboard

Assigns different keyboard shortcuts to different commands.

When right clicking a keyboard shortcut in the Keyboard tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append shortcut or Insert shortcut from the context menu, the Add keyboard shortcut dialog box displays.

- Select available tool: assigns an existing command to the new keyboard shortcut. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the shortcut. If this option is selected, Available tools is grayed out.
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list.
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar
- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool.

Menu Group/Keyboard Shortcut

Displays the options of the selected toolbar, tool, control, flyout or separator.

Key

Specifies the used keyboard shortcut.

Tool ID

Identifies the keyboard item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Help

Specifies the help string displayed on the status bar, even though help strings are not displayed by keyboard shortcuts.

Command

Specifies the command(s) to associate with the keyboard shortcut.

Image

Specifies the image to associate with the command, even though images are not used with keyboard shortcuts.

Mouse

Modifies the different actions that are connected to the mouse buttons.

When right clicking a button or button group in the Mouse tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append button or Insert button from the context menu, the Add button item dialog box displays.

Alias
Defines the aliases of the selected item. The browse icon opens the 'Edit Aliases' dialog box where you can edit, create or delete aliases.

Button
Specifies the used mouse button.

Tool ID
Identifies the menu item to other elements in the CUI file. (The Tool ID is assigned by BricsCAD).

Title
Specifies the name that is displayed by the mouse button.

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros.

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

Tablet
Customizes the tablet options.
When right clicking a button or button group in the Tablet tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: When you choose Append button or Insert button from the context menu, the Add button item dialog box displays.

Menu Group/Button Group

Alias
Defines the aliases of the selected item. The browse icon opens the 'Edit Aliases' dialog box where you can edit, create or delete aliases.

Quad
Adds commands to the quad tab or move commands to other tabs.
When right clicking a quad tab or quad button in the Quad tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Append quad button or Insert from the context menu, the Add quad button dialog box displays.

- Select available tool: assigns an existing command to the new quad button. If this option is selected, the other options are grayed out, except for Available tools
- Create new tool: assigns a new command/macro to the new quad button. If this option is selected, Available tools is greyed out
- Toolbox: specifies the toolbox to which to add the new command. You can choose a toolbox in the drop-down list
- Title: specifies the name of the new tool
- Help: specifies the help string displayed on the status bar

- Command: specifies the command(s) or macros
- Image: specifies the image to display for the new tool. The browse icon opens the Tool Image dialog box. There are five options to select an image
- Available tools: selects an existing tool

ID
Identifies the Quad item in the CUI file. (The ID is assigned by BricsCAD).

Title
Specifies the name that is displayed by the quad button.

Help
Specifies the help string displayed on the status bar.

Command
Specifies the command(s) or macros related to the quad button.

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

Entity Filter
Specifies which items will be displayed in the Quad, depending on the type of the entity.

Properties
Edits which properties are shown for different kinds of entities.

Nota: These properties will only be shown when the ROLLOVERTIPS system variable is ON.

Rollover
Specifies which properties should be displayed.

Workspaces
Controls which menu tabs are visible in different workspaces.
When right clicking an item in the Workspace tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Command Aliases
Customize aliases and add them to different commands.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add alias or Edit alias from the context menu, the Add alias or edit alias dialog box displays where you can add or edit an alias.

Alias
Specifies the name of the alias, a command abbreviation.

Nota: The list of Aliases and commands will be ordered alphabetically according to the aliases when you click on **Alias**.

Command
Specifies the command to which the alias is assigned.

Nota: The list of Aliases and commands will be ordered alphabetically according to the commands when you click on **Command**.

Add
Adds an alias.

Edit
Edits the selected alias.

Delete
Deletes the selected alias.

Commands
Lists all the available commands to add an alias to.

Shell Commands
Create and edit shell commands. Shell commands run programs external to BricsCAD.
When right clicking an item in the Command Aliases tab, a context menu opens. See the paragraph Options Within the Context Menu for further explanation.

Nota: If you select Add shell command or Edit shell command from the context menu, the Add shell command or edit shell command dialog box displays where you can add or edit a shell command.

Alias
Specifies the name of the shell command; this is the command you enter at the ' : ' prompt.

Shell Command
Specifies the operating system command to execute, or program to run.
To run a program, use the Start prefix, as in Start Notepad.exe.

Flags
Determines how the shell command operates:

- Don't wait: BricsCAD immediately returns to the command prompt without waiting for the application to finish
- Minimized: the application runs minimized
- Hidden: the application does not appear on the screen
- Quotes: the shell command uses quotations when the command string uses spaces, such as *C:\cad programs\file name.exe*

Prompt
Specifies the prompt to display in the Command line, for example: "Enter file to edit:"

Add
Adds a shell command through a dialog box.

Edit
Edits the shell command through a dialog box.

Delete
Deletes the selected shell command. Gives a warning where you can confirm you to delete the selected item.

Options Within the Context Menu

Append …
Adds the mentioned item below the last item.

Insert ...
Adds the mentioned item above the selected item.

Add …
Adds the mentioned item in alphabetical order.

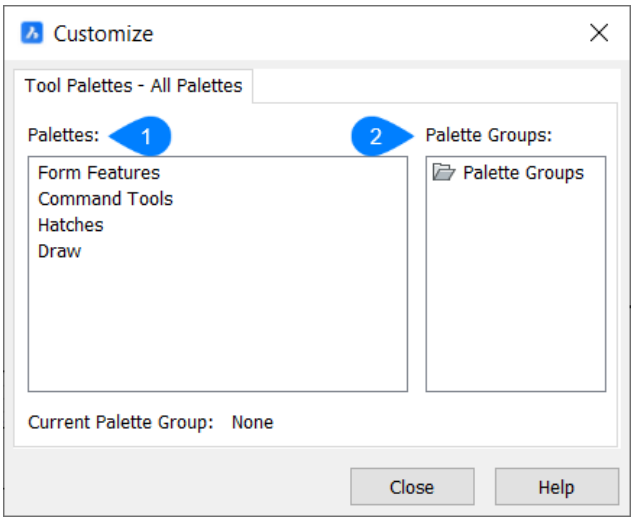
Delete ...
Deletes the menu, submenu, item or separator selected item. Gives a warning where you can confirm you to delete the selected item.

4.12.48.4 Manage your customizations

Displays the Manage Customizations dialog box for confirming and reversing changes made to the user interface.

4.12.49 Customize (Tool Palettes) dialog box

The Customize (Tool Palettes) dialog box allows you to customize your own tabs and tools to use in the tool palettes panel.



1 Palettes

2 Palette Groups

4.12.49.1 Palettes

Displays a list of all palettes available in the tool palettes panel. By right clicking on one of the palettes, several options may be available in the context menu.

Rename
Renames the Palette by entering the new name.

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

Delete
Deletes the selected palette

Export
Exports a palette through the Export Palette dialog box.

Import
Imports a palette through the Import Palette dialog box.

4.12.49.2 Palette Groups

Displays the loaded palette groups. By right clicking on one of the groups, several options may be available in the context menu.

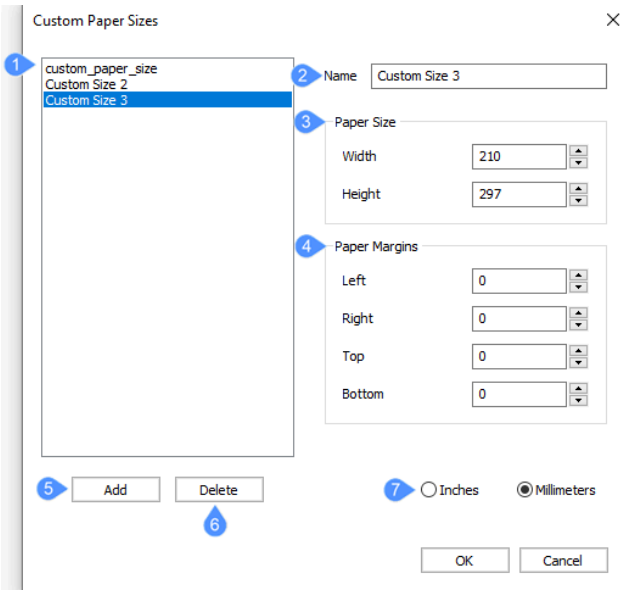
New group
Adds a new sub-group to the list with a generic name that can be changed right away.

Nota: For the additional sub group, the available options in the context menu are the same as those for palettes.

Import
Imports a palette through the Import Palette Group dialog box.

4.12.50 Custom paper sizes dialog box

The Custom Paper Sizes Dialog Box allows you to create custom paper sizes when Print as PDF.pc3 is selected in the Printer/Plotter Configuration list.



1 Custom Paper Sizes list

2 Name

3 Paper Size

4 Paper Margins

5 Add

6 Delete

7 Units

4.12.50.1 Custom Paper Sizes list

Displays the list of all custom paper sizes created by the user.

4.12.50.2 Name

Allows you to name the custom paper size.

4.12.50.3 Paper Size

You can insert the width and the height of the page in the appropriate fields.

4.12.50.4 Paper Margins

You can set the paper margins by entering the desired values in the fields. Only positive values are allowed.

4.12.50.5 Add

Allows you to create the new custom paper size and adds it to the list.

4.12.50.6 Delete

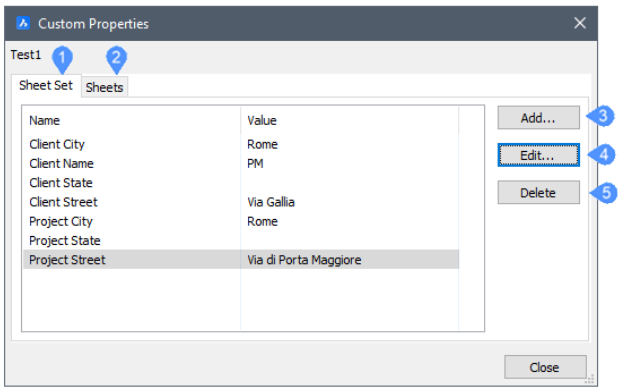
Allows you to select a custom paper size from the list and delete it. You can delete one item at a time.

4.12.50.7 Units

You can choose whether to use inches or millimeters for your custom paper size.

4.12.51 Custom properties dialog box

The **Custom Properties** dialog box allows you add custom properties for sheet set and sheets.



1 Sheet Set Tab

2 Sheets Tab

3 Add

4 Edit

5 Delete

4.12.51.1 Sheet Set Tab

Manages the custom sheet set properties.

4.12.51.2 Sheets Tab

Manages the custom sheets properties.

4.12.51.3 Add

Adds a custom property. Displays the **Add Custom Property** dialog box in which you add a property name and its initial value.

4.12.51.4 Edit

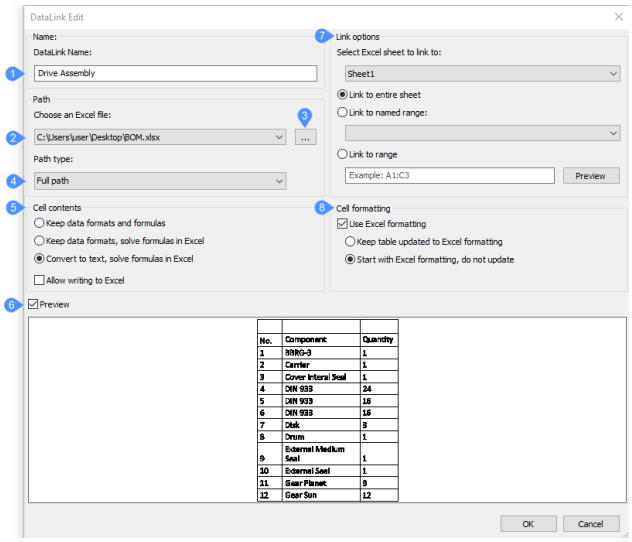
Allows to edit the value of a selected custom property. Displays the **Custom Sheet Property** or **Custom Sheet Set Property** dialog box in which you edit the name of the property.

4.12.51.5 Delete

Deletes a selected custom property.

4.12.52 DataLink edit dialog box

The **DataLink Edit** dialog box allows you to link data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Name

2 Path

3 Browse

4 Path type

5 Cell contents

6 Preview

7 Link options

8 Cell formatting

4.12.52.1 Name

You can type a name in the **DataLink Name** field.

4.12.52.2 Path

Displays you the path where your Excel file is located.

4.12.52.3 Browse

Allows you to select a data file. *.xls, *.xlsx and *.csv files are accepted.

4.12.52.4 Path type

Allows you to choose a path type option:

- **Full Path**: BricsCAD searches for the data file in the specified folder only.
- **Relative Path**: the data file is searched for in a subfolder of the drawing file folder.
- **No Path**: the data file is searched for in the drawing file folder only.

4.12.52.5 Cell contents

- **Keep data formats and formulas**: imports data with formulas and supported data formats.

- **Keep data formats, solve formulas in Excel**: imports data formats. Data is calculated from formulas in Excel.
- **Convert to text, solve formulas in Excel**: imports data as text with data calculated from formulas in Excel.
- **Allow writing to Excel**: if checked, the DATALINKUPDATE command uploads changes to linked data in your drawing to the original external spreadsheet.

4.12.52.6 Preview

If this button is selected, a preview of the data file is displayed.

4.12.52.7 Link options

Allows you to select Excel sheet to link to:

- **Link to the entire sheet**: all cells that contain data are linked.
- **Linked to named range**: links to a named range of cells. Click the button to select a named range.
- **Link to range**: specify a range of cells. Click the **Preview** button to display the selected range.

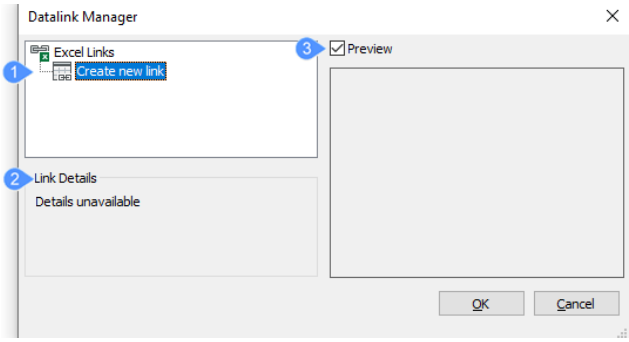
4.12.52.8 Cell formatting

You must select the **Use Excel formatting** option to be able to select one of the following options:

- **Keep table updated to Excel formatting**: updates any changed formatting when the DATALINKUPDATE command is executed.
- **Start with Excel formatting. Do not update**: imports the formatting specified in the original data file. The DATALINKUPDATE command does not affect any changes made to the cell formatting.

4.12.53 Datalink manager dialog box

The **Datalink Manager** dialog box links data from an Excel spreadsheet directly into drawing tables for bi-directional updating.



1 Create new link

2 Link details

3 Preview

4.12.53.1 Create new link

Opens the **DataLink Edit** dialog box.

4.12.53.2 Link details

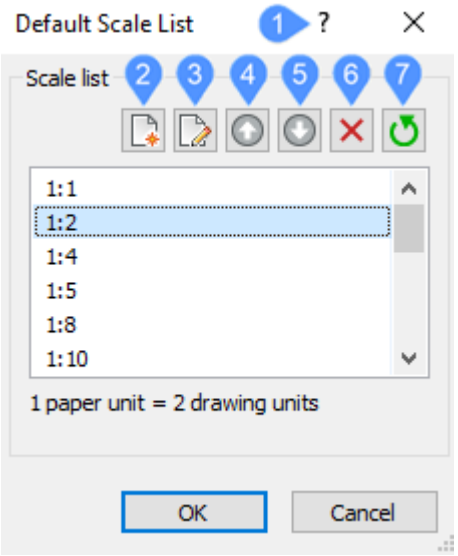
Displays link details.

4.12.53.3 Preview

Display a preview of the datalink.

4.12.54 Default scale list dialog box

The **Default Scale List** dialog box allows you to edit the list of scale factors displayed by commands, such as PRINT and PAGESETUP and by annotative scale factors.



1 Command reference

2 Add new scale

3 Edit scale

4 Move up

5 Move down

6 Delete

7 Reset

4.12.54.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

4.12.54.2 Add new scale

Allows you to add a new scale factor to the list. Opens the **Add Scale** dialog box.

4.12.54.3 Edit scale

Allows you to edit the factors of an existing scale. Opens the **Edit Scale** dialog box.

4.12.54.4 Move up

Moves the selected scale factor up the list.

4.12.54.5 Move down

Moves the selected scale factor down the list.

4.12.54.6 Delete

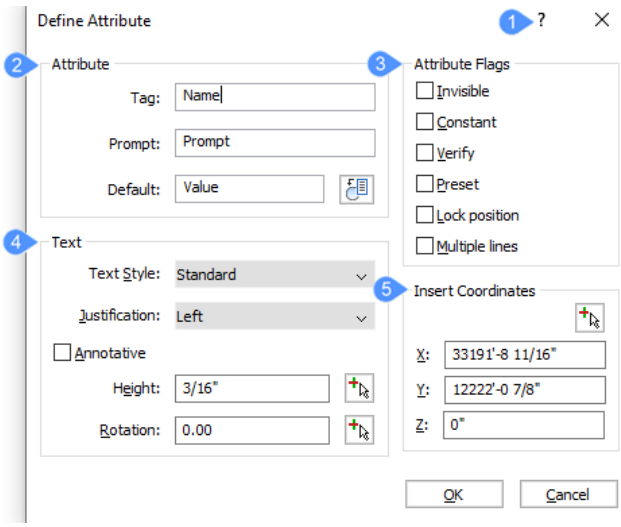
Deletes the selected scale factor(s). Press the Ctrl key to select and delete more than one factor at a time. Scale 1:1 cannot be deleted.

4.12.54.7 Reset

Returns the list to its original condition, adding back in the deleted factors and removing those that were added, but are not being used.

4.12.55 Define attribute dialog box

The **Define Attribute** Dialog Box defines the options for the attribute values.



1 Command reference

2 Attribute options

3 Attribute Flags options

4 Text options

5 Insert Coordinates options

4.12.55.1 Command reference

Opens the Bricsys Help article about the DEFAULTSCALELIST command.

4.12.55.2 Attribute options

- **Tag**

  Specify the name of the attribute. This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers and punctuation.

- **Prompt**

  Specify the user prompt. This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank. BricsCAD then uses the tag as the prompt during attribute insertion.

- **Default**

  Specify the default value. This is displayed in angle brackets, like <360>. As with other defaults, the user only needs to press Enter to accept this value.

  Click the **Insert Field** icon to assign a field value to the attribute (see the FIELD command).

4.12.55.3 Attribute Flags options

- **Invisible**

  Hide the attribute from view; they are not displayed and not printed. Hidden attributes can, however, be displayed with the ATTDISP command.

- **Constant**

  Specify a default value that the user cannot change.

- **Verify**

  Forces to enter the value a second time; this helps to ensure that the value is entered correctly.

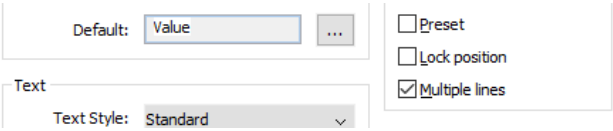
- **Preset**

  Inserts attributes without prompting the user; the attributes can be changes later with the ATTEDIT command.

- **Lock position**
  - On: locks the position of the attribute within the block reference.
  - Off: unlocked attributes can be moved relative to the rest of the block using grip editing.

- **Multiple lines**
  - On: allows the use of multiline text.
  - Off: limits to a single line.

  When the **Multiple lines** box is checked, the button icon displays next to the dimmed **Default** text field.



When you click the button, the **Define Attribute** dialog box temporarily closes to let you define a multiline text entity.

The MTEXT command launches. Type the text for the attribute, then click the **OK** button on the **Text Formatting** toolbar to return to the **Attribute Definition** dialog box.

4.12.55.4 Text options

- **Text style**

  Specify the text style for the attribute text. Choose from one of the styles defined in the drawing. (You can create additional text styles with the STYLE command.)
- **Justification**

  Specify the justification of the attribute text, such as **Center** or **Fit**.
- **Height**

  Specify the height of the attribute text, unless overridden by the text style. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.
- **Rotation**

  Specify the rotation angle of the rotation text. Enter a value, or specify two points in the drawing by clicking the button. The last option dismisses the dialog box temporarily.
- **Annotative**

  Specify the **Annotative** property of the attribute text.

4.12.55.5 Insert Coordinates options

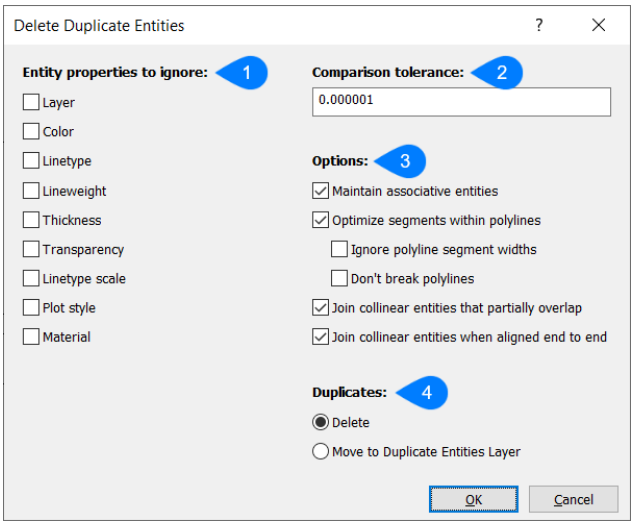
- **Select insertion point**

  Specify the insertion point directly in model.
- **X/Y/Z**

  Specify the coordinates of the attribute's insertion point.

4.12.56 Delete duplicate entities dialog box

The Delete Duplicate Entities dialog box allows you to delete duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones.



1 Entity properties to ignore

2 Comparison tolerance

3 Options

4 Duplicates

4.12.56.1 Entity properties to ignore

Selected properties are ignored when comparing duplicate or overlapping entities.

4.12.56.2 Comparison tolerance

Sets the tolerance for the comparison process. If set to 0 (zero), entities must match completely before being evaluated in the overkill process.

4.12.56.3 Options

Determines how special cases are handled.

Maintain associative entities
Associative entities are not processed.

Optimize segments within polylines
Evaluates segments of selected polylines individually, duplicate vertices and segments are removed.

- **Ignore polyline segment widths**: ignores the Width property of overlapping polyline segments.
- **Don't break polylines**: polylines entities are kept intact.

Join collinear entities that partially overlap
Unifies partially overlapping entities into a single entity.

Join collinear entities when aligned end to end
Unifies collinear lines and polylines having coinciding endpoints into a single entity or segment.

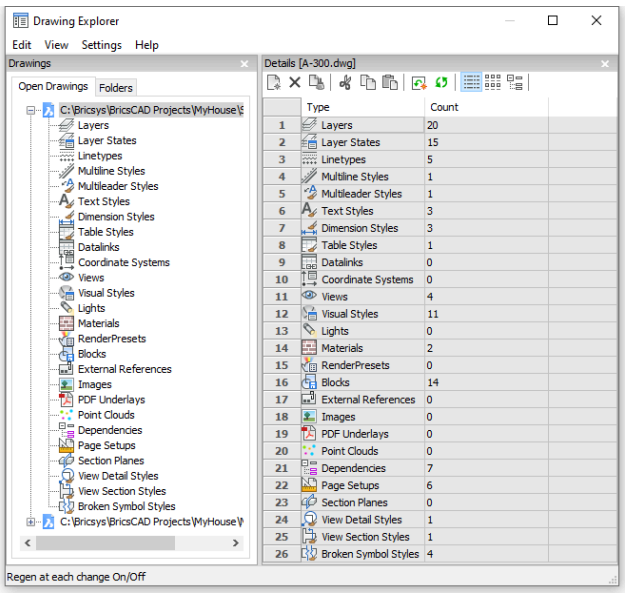
4.12.56.4 Duplicates

Determines what to do with duplicates: either delete them or move them to a Duplicate Entities layer.

4.12.57 Drawing explorer dialog box

4.12.57.1 Opening the Drawing Explorer dialog box

- Launch the EXPLORER command.
- Launch a definition category command, such as LAYERS or EXPBLOCKS.
- Choose a definition category in the **Tools** > **Drawing Explorer** menu.
- Choose **Drawing Explorer** in the **General** tab of the No Selection Quad.



Categories	Description
Layers	Reports the number of layers defined in the selected drawing. Click to explore the layers; see the LAYER command.
Layer States	Reports the number of layer states defined in the selected drawing. Click to explore the layer states; see the LAYERSTATE command.
Linetypes	Reports the number of linetypes loaded into the selected drawing. Click to explore the linetypes; see the LINETYPE command.
Multiline Styles	Reports the number of multiline styles defined in the selected drawing. Click to explore multiline styles; see the MLSTYLE command.
Multileader Styles	Reports the number of multileader styles defined in the selected drawing. Click to explore multileader styles; see the MLEADERSTYLE command.
Text Styles	Reports the number of text styles defined in the selected drawing. Click to explore text styles; see the STYLE command.

Categories	Description
Dimension Styles	Reports the number of dimension styles defined in the selected drawing. Click to explore dimension styles; see the DIMSTYLE command.
Table Styles	Reports the number of table styles defined in the selected drawing. Click to explore table styles; see the TABLESTYLE command.
Data Links	Reports the number of data in the drawing linked to external spreadsheet files. Click to explore data links; see the DATALINK command.
Coordinate Systems	Reports the number of named user coordinate systems (UCS) defined in the selected drawing. Click to explore coordinate systems; see the EXPUCS command.
Views	Reports the number of named views in the selected drawing. Click to explore saved views; see the VIEW command.
Visual Styles	Reports the number of visual styles defined in the selected drawing. Click to explore visual styles; see the VISUALSTYLES command.
Lights	Reports the number of lights placed in the selected drawing. Click to explore lights; see the LIGHTLIST command.
Materials	Reports the number of materials defined in the selected drawing. Click to explore materials; see the MATERIALS command.
Render Presets	Reports the number of render presets defined in the drawing. Click to explore render presets; see the RENDERPRESETS command.
Blocks	Reports the number of blocks defined in the selected drawing. Click to explore blocks; see the EXPBLOCKS command.
External References	Reports the number of external references (xrefs) attached to the selected drawing. Click to explore xrefs; see the XREF command.
Images	Reports the number of images attached to the selected drawing. Click to explore the images; see the IMAGE command.
PDF Underlays	Reports the number of PDF underlays attached to the selected drawing. Click to explore PDF underlays; see the PDF command.
Point Clouds	Reports the number of point clouds attached to the selected drawing. Click to explore point cloud; see the POINTCLOUD command.

Categories	Description
Dependencies	Reports the number of external files that the selected drawing depends on. Click to explore dependencies; see the ETRANSMIT command.
Page Setups	Reports the number of the page setups attached to the selected drawing. Click to explore page setups; see the PAGESETUP command.
Section Planes	Reports the number of section planes defined in the selected drawing. Click to explore the section planes; see the SECTIONPLANESETTINGS command.
View Detail Styles	Reports the number of view detail styles defined in the selected drawing. Click to explore the view detail styles; see the VIEWDETAILSTYLE command.
View Section Styles	Reports the number of view section styles defined in the selected drawing. Click to explore the view section styles; see the VIEWSECTIONSTYLE command.
Broken Symbol Styles	Reports the number of available styles for the VIEWBREAK command. Click to explore the broken symbol styles.

4.12.57.2 The Drawing Explorer layout

The **Drawing Explorer** window consists of a number of sub-windows or panes.

You can select which panes you want to have open or closed in the **View** menu. BricsCAD remembers the display settings for each category.

Item	Description
Drawing	Displays the **Drawing** pane. • **Open Drawings**: a list of all drawings that are currently open. • **Folders**: - **Local Folders**: your favorite local drawing folders. - **Bricsys 24/7**: your drawing folders on Bricsys 24/7.
Details	Displays the **Details** pane. Shows the details of the selected drawing or the details of the selected category.
Preview	Displays the **Preview** pane. Shows a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material.
Editor	Displays the **Editor** pane. Allows you to define or edit the properties of the selected category.

Item	Description
Status bar	Displays the Status bar at the bottom of the **Drawing Explorer** dialog box.

	View options for the Details pane
Details	Displays the details of definitions in a grid.
Icons	Displays definitions as an array of icons; applies only to blocks and external references.
Tree	Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies.

Regen	Updates the drawing display to reflect changes made in this dialog box.
Regen at each change	Toggles the automatic updating of the display at each change.

Hide Xref symbols	Do not display definitions in Xrefs.

Nota:

- If you click a definition category of an open drawing in the **Drawings** sub-window when the **Details** sub-window is closed, the **Details** window is re-opened, showing the content of the selected category.
- To close a sub-window, click the close button (✕) in its title bar.
- To restore the default layout, choose **Restore Default Explorer Layout** in the **Settings** menu.

4.12.57.3 The Settings menu

Item	Description
Restore Explorer Default Layout	Restores the layout of the Drawing Explorer to the default layout for the selected category. When you makes changes to the layout of the Drawing Explorer, the program remembers them. This option resets all your changes.

Item	Description
Options	Applies to Blocks and External References only. Specifies default settings for inserting blocks. See the EXPBLOCK command.

4.12.57.4 The Details toolbar

Icon	Name	Shortcut	Description
	New	Ctrl + N	Creates a new definition with a generic name, such as NewLayer 1 or NewStyle1. Use the Rename button to rename it.
	Delete	Ctrl + D	Erases the selected definition from the drawing. When the item is in use, such as a layer with entities, you have to option to retain the entities from the layer to be erased. Some items cannot be erased, such as layer 0, Global material, visual style 2D Wireframe, …
	Purge	Ctrl + P	Purges unused definitions from the drawing. Some definitions cannot be erased, such as layer 0 and text style Standard. See the PURGE command.
	Cut	Ctrl + X	Copies the selected definition(s) to the Clipboard, and then deletes it from the drawing following the same procedure as the Delete tool.
	Copy	Ctrl + C	Copies the definition(s) to the Clipboard for pasting elsewhere in the Drawing Explorer.
	Paste	Ctrl + V	Pastes the definition(s) from the Clipboard, into the Drawing Explorer. When the item on which the Clipboard content will be pasted is in use, such as a layer with entities, you have options: • **Copy and Replace** - replaces the existing item with the pasted one • **Don't Copy** - cancels the copy operation • **Copy, but keep both** - pastes the Clipboard content with an incremented name, usually suffixed by a 1 • **Cancel** - also cancels the operation
	Details View		Displays the details of definitions in a grid.
	Icon View		Displays definitions as an array of icons; applies only to blocks and external references.

Icon	Name	Shortcut	Description
	Tree View		Displays definitions in tree form; applies only to layer states, data links, dimension styles, external references, images, PDF underlays and dependencies.
	Regen		Updates the drawing display to reflect changes made in this dialog box.
	Toggle Regen		Toggles the automatic updating of the display at each change.

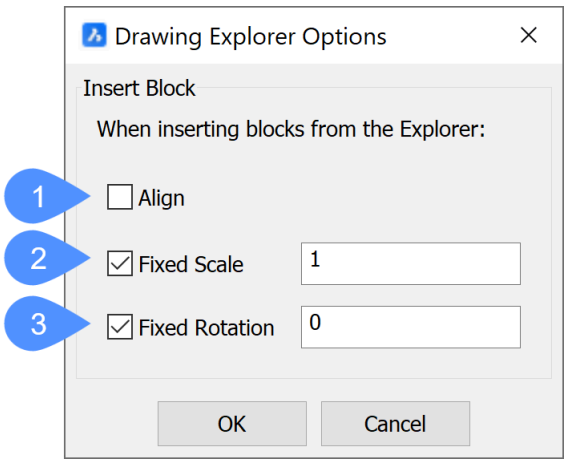
4.12.57.5 The Context menu

Select a definition, then right-click.

Item	Shortcut	Description
New	Ctrl + N	Creates a new definition as a copy of the selected definition.
Delete	Ctrl + D	Deletes the selected definitions. Definitions that are in use cannot be deleted.
Purge	Ctrl + P	Deletes unused definitions in the current selection.
Cut	Ctrl + X	Cuts the selected definitions to the clipboard.
Copy	Ctrl + C	Copies the selected definitions to the clipboard.
Paste	Ctrl + V	Pastes definitions from the clipboard.
Rename	F2	Renames the selected definition; alternatively, click on the name twice, slowly, to rename it.
Select All	Ctrl + A	Selects all definitions in the current category.
Invert Selection		Inverts the selected definition(s), e.g. select a few layers, then use this to select all other layers.
Current		Makes the definition current, such as making the selected layer the current (working) layer.

4.12.58 Drawing explorer options dialog box

The **Drawing Explorer Options** dialog box allows you to align or set the scale and the rotation of the inserted blocks.



1 Align

2 Fixed Scale

3 Fixed Rotation

4.12.58.1 Align

Tick the checkbox to align blocks with the snapped entity.

4.12.58.2 Fixed Scale

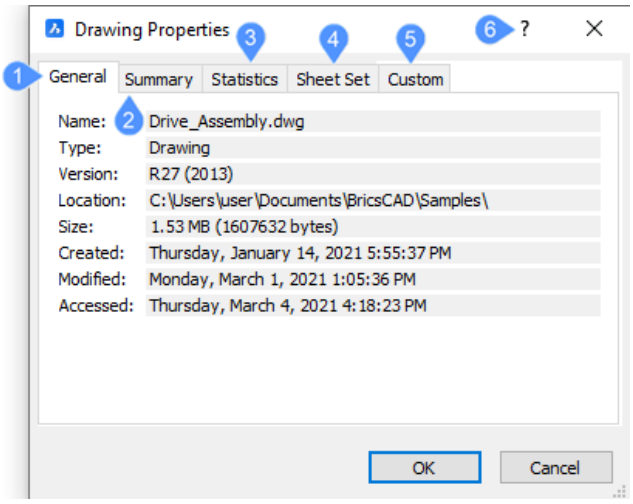
Tick the checkbox to specify the scale factor to be used for all block insertions.

4.12.58.3 Fixed Rotation

Tick the checkbox to specify the rotation angle to be used for all block insertions.

4.12.59 Drawing properties

The **Drawing Properties** dialog box allows you to see general information about the drawing and user-defined properties.



1 General

2 Summary

3 Statistics

4 Sheet Set

5 Custom

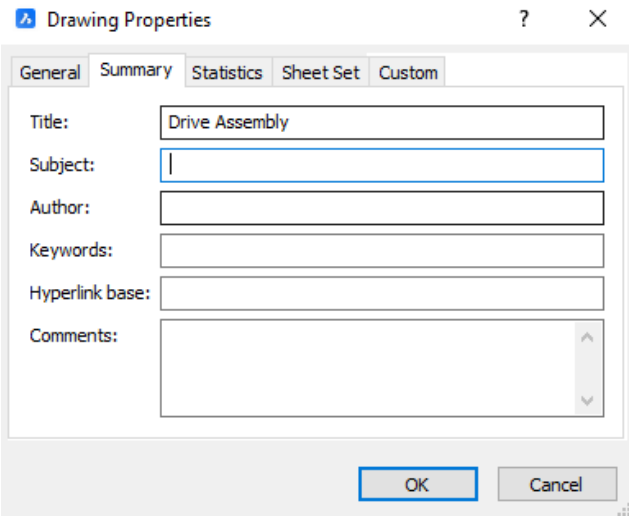
6 Command reference

4.12.59.1 General

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

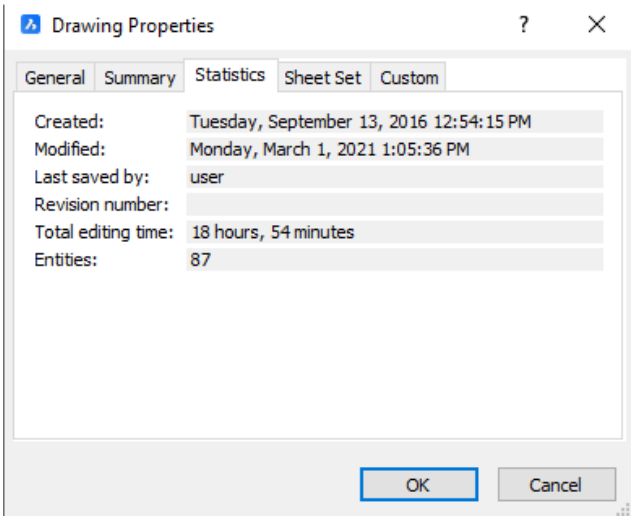
4.12.59.2 Summary

Lets you define the summary properties of the drawing, such as subject and keywords. This data can be searched for by EDMS (engineering data management software) to find drawings in large collections.



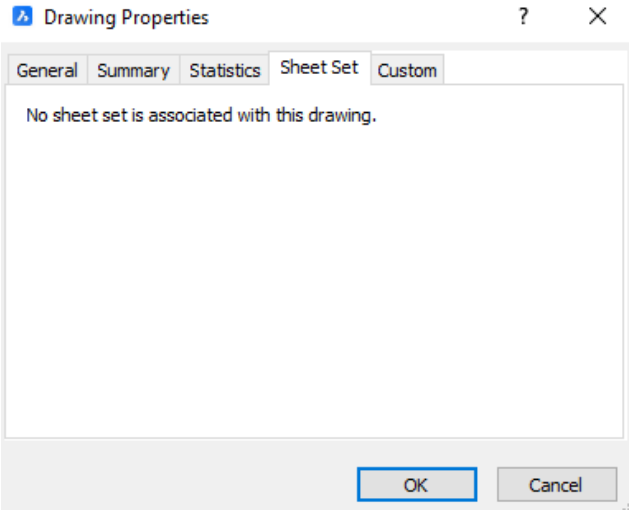
4.12.59.3 Statistics

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



4.12.59.4 Sheet Set

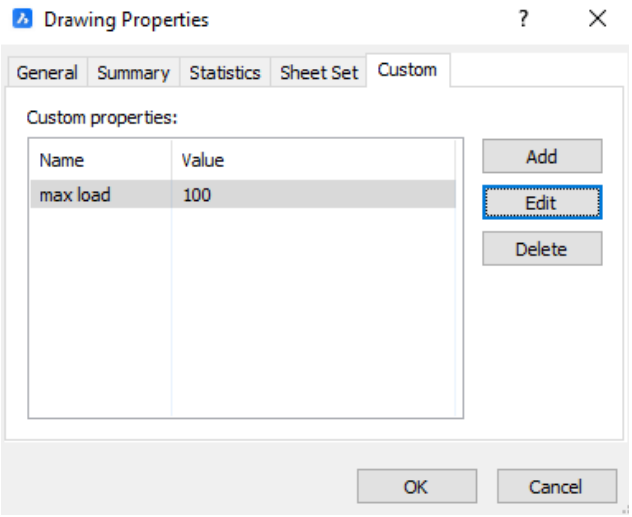
Displays sheet set data when the drawing is associated with a sheet set.



4.12.59.5 Custom

Displays the custom properties of the drawing. Click the **Add** button to add a new property or the **Edit** button to edit a property.

The Custom Property dialog box appears when clicking the **Add** and **Edit** buttons.

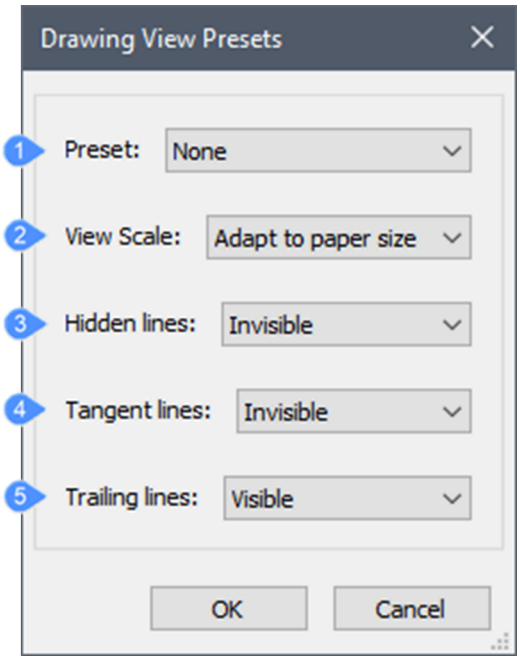


4.12.59.6 Command reference

Opens the Bricsys Help article about the DWGPROPS command.

4.12.60 Drawing view presets dialog box

Specifies the types of generated drawings and their placement in the layout.



1 Preset

2 View Scale

3 Hidden lines

4 Tangent lines

5 Trailing lines

4.12.60.1 Preset

Select a preset view arrangement:

- **None**: No preset affects the views generated.
- **Architectural Views**: Generates:
  - Five standard views: **Front**, **Right**, **Back**, **Left** (used for elevations) and **Top** views.
  - Three section views: two horizontal sections (for plans) and 1 vertical section.
- **Mechanical Views**: Generates:
  - Three standard views: **Front**, **Top** and **Left** (first angle projection) or **Right** (third angle projection).

The layout of the mechanical views is controlled by the projection type: first or third angle. The current projection type is saved in the PROJECTIONTYPE system variable.

4.12.60.2 View Scale

Sets the default scale factor for the drawing views from drop-down list. The scale list can be edited by the SCALELISTEDIT command.

4.12.60.3 Hidden lines

Sets the default setting for hidden lines in drawing views.

4.12.60.4 Tangent lines

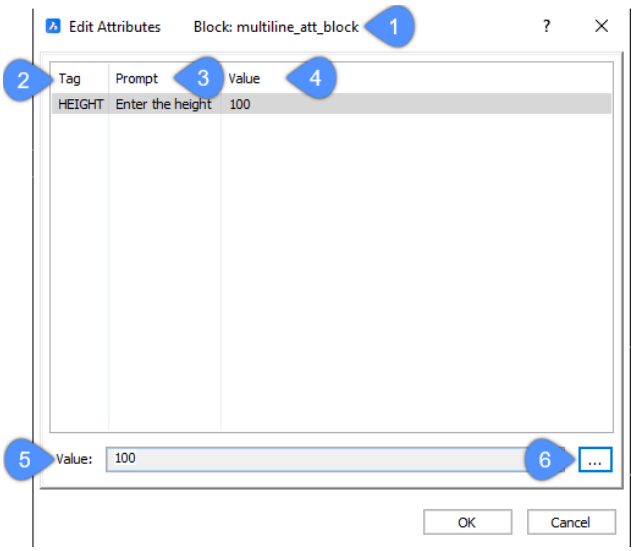
Sets the default for **Tangent Lines** in drawing views.

4.12.60.5 Trailing lines

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

4.12.61 Edit attributes dialog box

The Edit Attributes dialog box allows you to edit the values of attributes.



1 Block name

2 Tag

3 Prompt

4 Value

5 Value

6 Text Formatting

4.12.61.1 Block name

Displays the name of the selected block.

4.12.61.2 Tag

Specifies the name of the attribute.

Nota: This is the name by which BricsCAD identifies the attribute. You can use up to 255 letters, numbers, and punctuation.

4.12.61.3 Prompt

Specifies the user prompt.

Nota: This is displayed at the command prompt when the attribute is later inserted in the drawing. You can leave this field blank, BricsCAD uses the tag as the prompt during attribute insertion.

4.12.61.4 Value

Displays the value that belongs to a certain attribute.

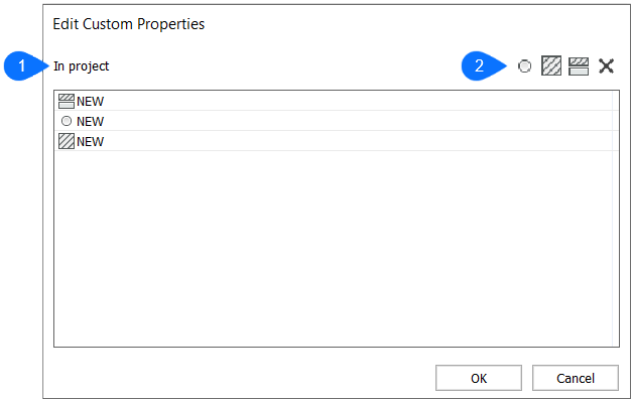
4.12.61.5 ... Text Formatting

Changes the value of the attribute text.

Nota: To be able to use this option, the Multiple lines attribute flag must be selected, in the Define Attribute dialog box, when defining an attribute.

4.12.62 Edit custom properties dialog box

The Edit Custom Properties dialog box allows you to add custom properties to materials and compositions.



1 In project

2 Add Generic/ Material/Composition/Delete properties

4.12.62.1 In project

Displays the custom properties available in the project.

4.12.62.2 Add/Delete Properties

Add Generic Custom Property
Creates a new generic custom property, which can be assigned to building elements in the project.

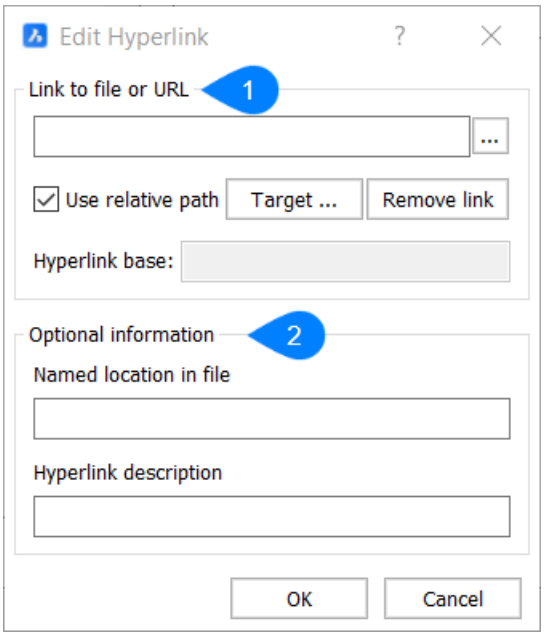
Add Material Custom Property
Creates a new material custom property, which can be assigned to materials in the project.

Add Composition Custom Property
Creates a new composition custom property, which can be assigned to compositions in the project.

Delete
Deletes the selected property.

4.12.63 Edit hyperlink dialog box

The Edit Hyperlink dialog box allows you to change the hyperlink to a file or URL.



1 Link to file or URL

2 Optional information

4.12.63.1 Link to file or URL

Specifies the URL of the file either by entering the URL or file path or through the Select File dialog box by pressing the Browse button.

Use relative path
Specifies a relative path by selecting the target file.

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

Hyperlink base
Displays the hyperlink base path.

4.12.63.2 Optional information

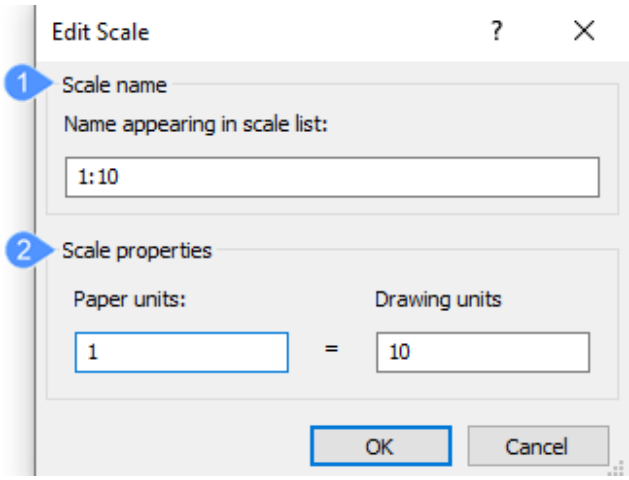
Specifies additional information

Named location file
Specifies a bookmark name. The # character specifies locations (bookmarks) within files or Web pages.

Hyperlink description
Adds an additional description of the hyperlink.

4.12.64 Edit scale dialog box

The **Edit Scale** dialog box allows you to edit the name and the scale properties of a scale factor.



1 Scale name

2 Scale properties

4.12.64.1 Scale name

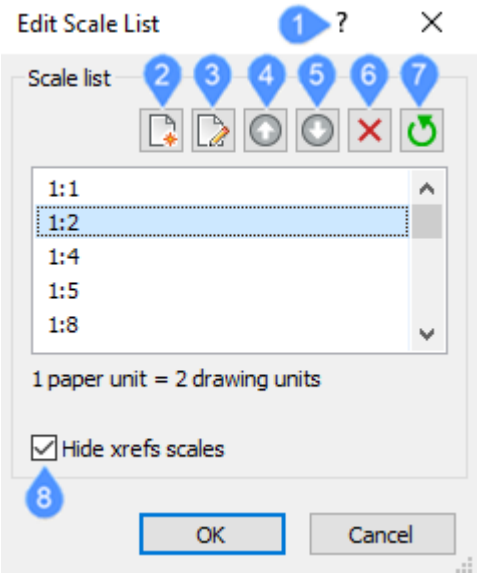
Edit the name by typing the new name.

4.12.64.2 Scale properties

Change the scale factor in paper (size when plotted) and/or drawing units.

4.12.65 Edit scale list dialog box

The **Edit Scale List** dialog box allows you to add, edit and delete scales.



1 Command reference

2 Add a new scale

3 Edit scale

4 Move up

5 Move down

6 Delete the selected scale

7 Reset

8 Hide xrefs scales

4.12.65.1 Command reference

Opens the Bricsys Help article about the SCALELISTEDIT command.

4.12.65.2 Add a new scale

Opens the **Add scale** dialog box.

4.12.65.3 Edit scale

Opens the **Edit Scale** dialog box.

4.12.65.4 Move up

Moves the selected scale up the list.

4.12.65.5 Move down

Moves the selected scale down the list.

4.12.65.6 Delete the selected scale

Deletes the selected scale. Press the CTRL key to select and delete more than one scale at a time.

Note that the scale 1:1 cannot be deleted.

4.12.65.7 Reset

Returns the list to its original condition, adding back in the deleted scales and removing all added but unused scales.

4.12.65.8 Hide xrefs scales

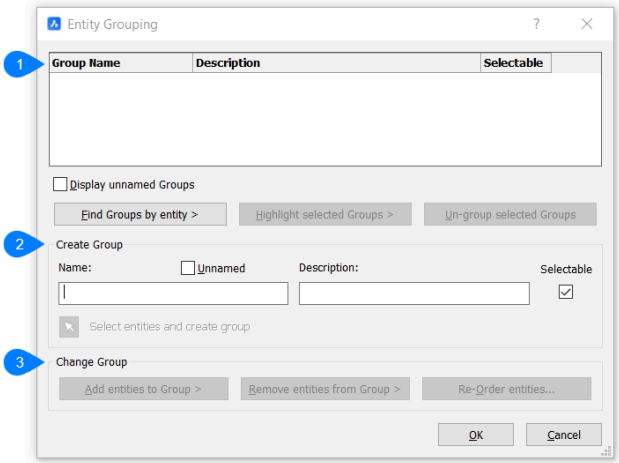
Determines if the scale factors from xrefs are displayed in the list:

- **Yes**: scale factors from externally referenced drawings are not shown in the list.
- **No**: scales in the attached xrefs not in the current drawing are listed here.

Toggles the HIDEXREFSCALES variable.

4.12.66 Entity grouping dialog box

The Entity grouping dialog box allows you to create, modify and delete named groups of entities.



1 Existing groups

2 Create Group

3 Change Group

4.12.66.1 Existing groups

Lists all existing groups with the Group Name and a short description.

Display unnamed Groups
Toggles the display of unnamed groups.

Find Groups by entity
Reports the name(s) of groups when entities in the drawing are selected.

Highlight selected Groups
Highlights all entities that belong to the selected group.

Ungroup selected Groups
Removes group status from selected entities.

4.12.66.2 Create Group

Name
Specifies the name of a group.

Unnamed
Toggles whether the group has a name.

Description
Adds an optional description of the group.

Selectable
Determines how groups are selected when the PICKSTYLE system variable is set to 1 or 3.

Select entities
Selects entities in the workspace which should be part of the group.

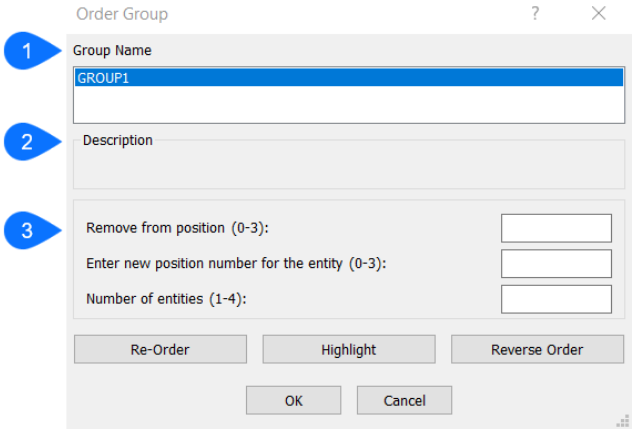
4.12.66.3 Change Group

Add entities to group
Adds entities to the selected group.

Remove entities from group
Removes entities from the selected group.

Re-Order entities
Changes the order of entities in groups trough the Order Group dialog box.

- Options in the Order Group dialog box

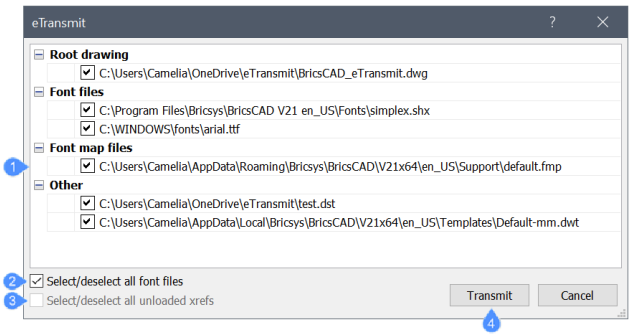


a Group Name
b Description
c Options

- **Group Name**: lists the names of named and unnamed groups.
- **Remove from position**: specifies the position number of the entity to be reordered.
- **Enter new position number for the entity**: specifies the new position number for the group.
- **Number of entities**: specifies the range of entities to reorder.
- **Re-Order**: applies the reordering dictated by the previous three fields.
- **Highlight**: highlights entities in the group; dismisses the Order Group dialog box.
- **Reverse Order**: reverses the order of entities in the group.

4.12.67 eTransmit dialog box

The eTransmit dialog box creates a package consisting of the drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables, and font map files.



1 Available files

2 Select/deselect all font files

3 Select/deselect all unloaded xrefs

4 Transmit

4.12.67.1 Available files

Select the files to be included in the eTransmit package (usually all of them).

4.12.67.2 Select/deselect all font files

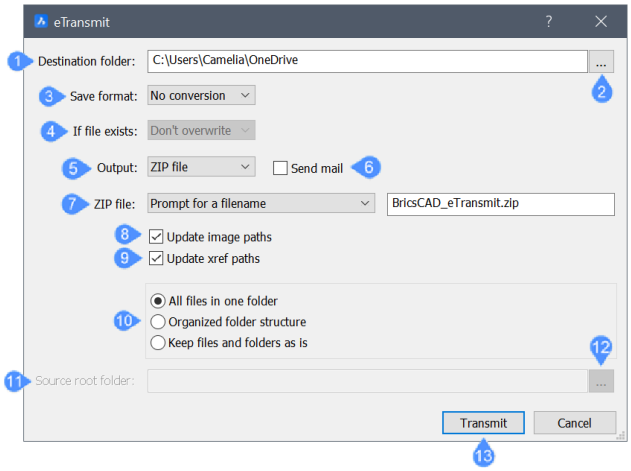
Tick the checkbox to automatically select all font files.

4.12.67.3 Select/deselect all unloaded xrefs

Tick the checkbox to automatically select all xref files.

4.12.67.4 Transmit

Opens the following dialog box:



1 Destination folder

2 Browse (destination folder)

3 Save format

4 If file exists

5 Output

6 Send mail

7 ZIP file

8 Update image paths

9 Update xref paths

10 Path options

11 Source root folder

12 Browse (root folder)

13 Transmit

Destination folder

Specifies the output folder for the eTransmit package. If the destination folder does not exist, you are prompted to create the folder.

Browse (destination folder)

Opens the Select the destination folder dialog box to allow you to choose the destination folder for the transmitted package.

Save format

Specifies the format of the drawing files in the eTransmit package.

If file exists

Specifies what to do if a file with the same name already exists in the output folder.

This option is available when Folder is chosen as output.

Select an option from the drop-down list:

- **Overwrite**: replaces the existing files with new ones.

- **Don't overwrite**: all files are transmitted, except files with the same file name as an existing file in the folder.
- **Abort**: the operation is aborted and no files are saved if a file with the same name already exists in the destination folder.

Output
Specifies whether to create a ZIP file or copy the files to the output folder.

Send mail
Tick the checkbox to send a mail with the zipped eTransmit package in attachment using your default mail application.
This option is available when ZIP is chosen as output.

ZIP file
Specifies what to do when a ZIP file with the same name already exists in the output folder.
Choose an option from the drop-down list:

- **Prompt for a file name**: opens a dialog box where you can choose a name and a path.
- **Overwrite if necessary**: replaces the existing file with the new one if a file with the same name already exists.
- **Increment name if necessary**: adds a suffix to the file name.

Update image paths
Tick the checkbox to update the saved paths for image files according to the new location of the transmitted files.

Update xref paths
Tick the checkbox to update the saved paths to externally-referenced files according to the new location of the transmitted files.

Path options
All files in one folder
All files in the eTransmit package are copied to the destination folder.

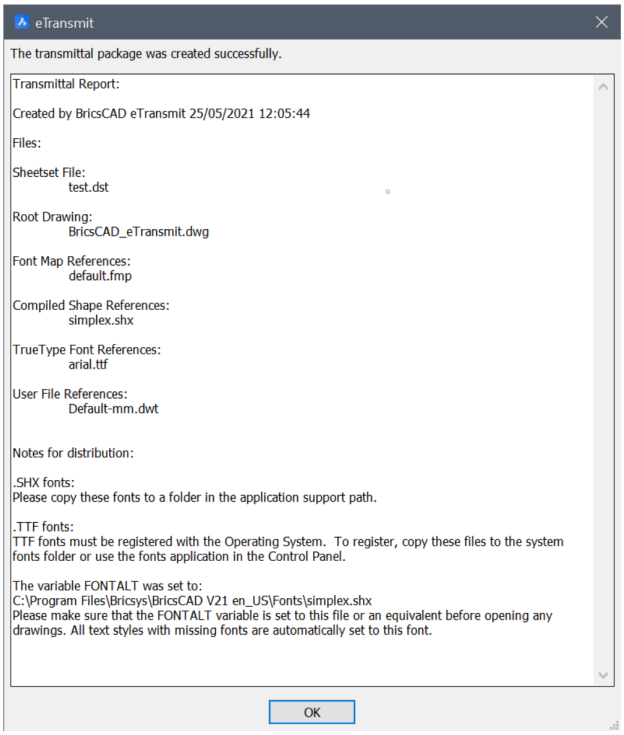
Organized folder structure
Creates a hierarchical folder tree with subfolders for all drawing file dependencies.

Keep files and folders as is
Keeps the folder structure of all files in the eTransmit package. This procedure allows you to easily install the package on another computer using the folder structure of the source system.

Source root folder
Specifies the source root folder for relative paths of drawing-dependent files, such as xrefs or image files.
This option is available when **Organized folder structure** is checked.

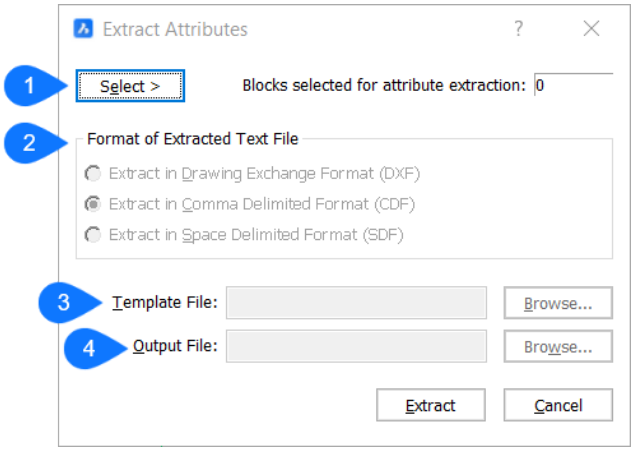
Browse (root folder)
Opens the Select the source root folder dialog box to allow you to choose a root folder.

Transmit
Transmits the files according to the selected options and displays a report of the transmission. The report contains a complete list of all files included and notes for distribution.



4.12.68 Extract attributes dialog box

The Extract Attributes dialog box allows you to copy data from attributes to a text file.



1 Select

2 Format of Extracted Text File

3 Template File

4 Output File

4.12.68.1 Select

Specifies the set of blocks with attributes to copy data from.

Nota: If blocks without attributes are selected, BricsCAD ignores them.

4.12.68.2 Format of Extracted Text File

Specifies the format of the Extracted text file. Several formats are available:

- DXF: drawing Exchange Format
- CDF: extract in Comma Delimited Format
- SDF: space Delimited Format

4.12.68.3 Template File

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

4.12.68.4 Output File

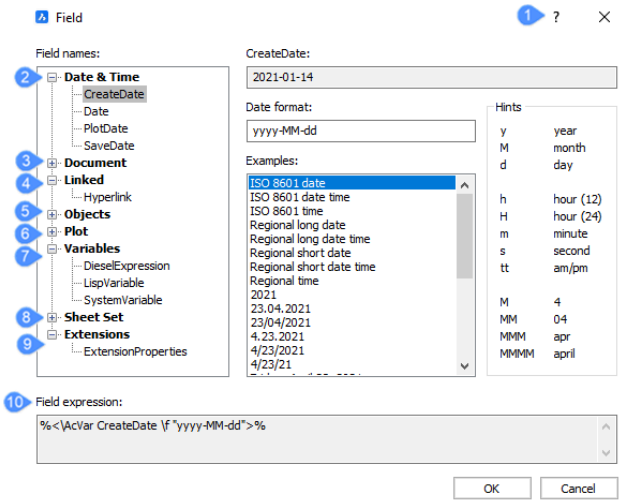
Specifies the location path of the output file by entering the path or through the Attribute Extract Output file dialog box after pressing the Browse button.

4.12.68.5 Extract

Exports the attribute values to the output file.

4.12.69 Field dialog box

The **Field** dialog box allows you to insert a field in a multiline text, table cells, attributes, and so on.



1 Command reference

2 Date & Time

3 Document

4 Linked

5 Objects

6 Plot

7 Variables

8 Sheet Set

9 Extensions

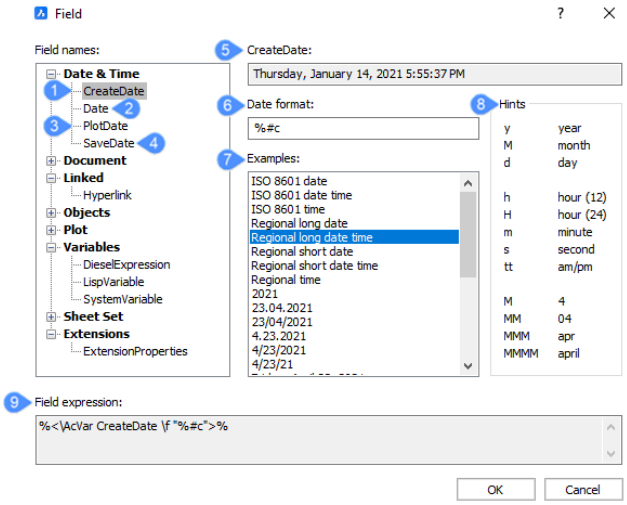
10 Field expression

4.12.69.1 Command reference

Opens the Bricsys Help article about the FIELD command.

4.12.69.2 Date & Time

For the **Date & Time** category, only the field number 1 changes based on the selected item.



1 CreateDate

2 Date

3 PlotDate

4 SaveDate

5 CreateDate

6 Date format

7 Examples

8 Hints

9 Field expression

CreateDate
Displays a random date.

Date
Displays the current date.

PlotDate
Displays the date the drawing was last plotted.

SaveDate
Displays the date the drawing was last saved.

Date format
Select a date format in the list or type a format in this field.

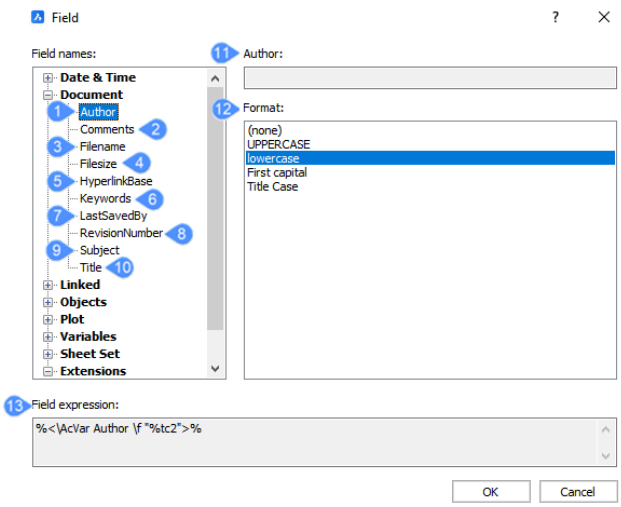
Examples
Displays date formats examples.

Hints
Explains the meaning of digits used in the date format acronym/abbreviation definition.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.3 Document

It includes 10 categories and 3 areas, excepting the Filename (no. 3) category that will be documented separately.



1 Author

2 Comments

3 Filename

4 Filesize

5 HyperlinkBase

6 Keywords

7 LastSavedBy

8 RevisionNumber

9 Subject

10 Title

11 Author

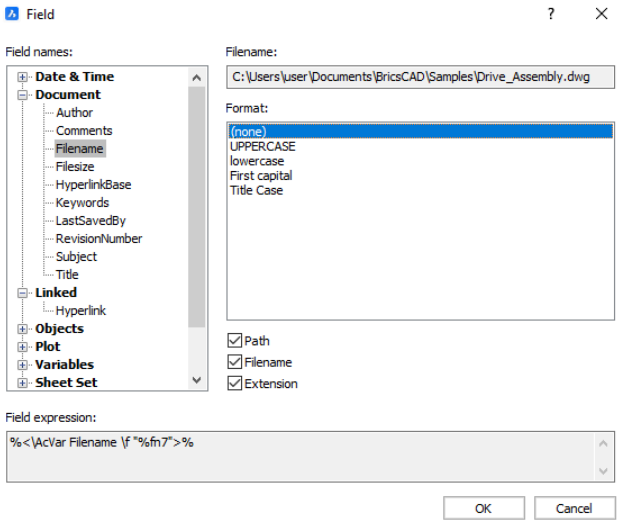
12 Format

13 Field expression

Author
Displays the author of the document.

Comments
Displays the comments of the document.

Filename
Shows the filename. It can include the path and the extension of the file.



Filesize
Displays the file size expressed in Bytes, Kilobytes and Megabytes.

HyperLinkBase
Displays the default path for relative hyperlink in drawings.

Keywords
Displays the keywords for the opened file.

LastSavedBy
Displays the name of the person who have last saved the file.

RevisionNumber
Displays the drawing revision number.

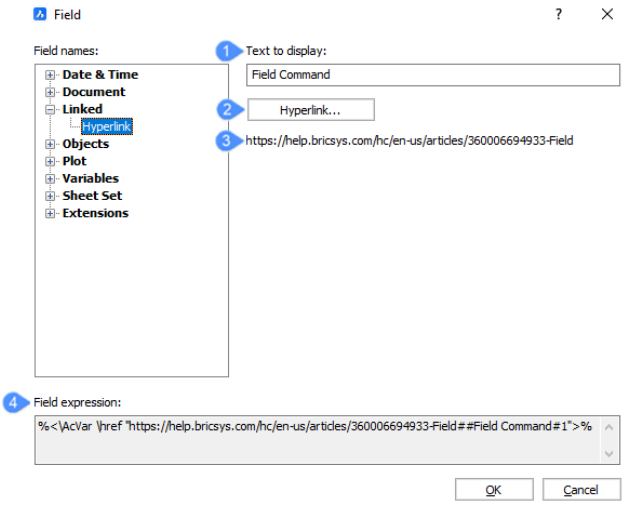
Subject
Displays the subject of the drawing.

Title
Displays the title of the drawing.

Format
Displays the format of the selected category.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.4 Linked



1 Text to display

2 Hyperlink

3 URL

4 Field expression

Text to display
You can insert in this field the text you want to be displayed for this hyperlink.

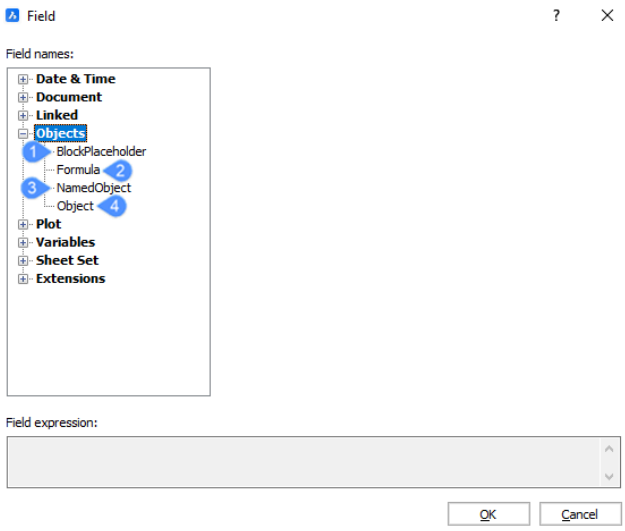
Hyperlink
Opens the **Edit Hyperlink** dialog box.

URL
Displays the URL.

Field expression
Displays the field expression of the selected date. An operand in an expression can be a DisplayProperty-Name enclosed in arrow brackets (<>).

4.12.69.5 Objects

It includes 4 categories.



1 BlockPlaceHolder

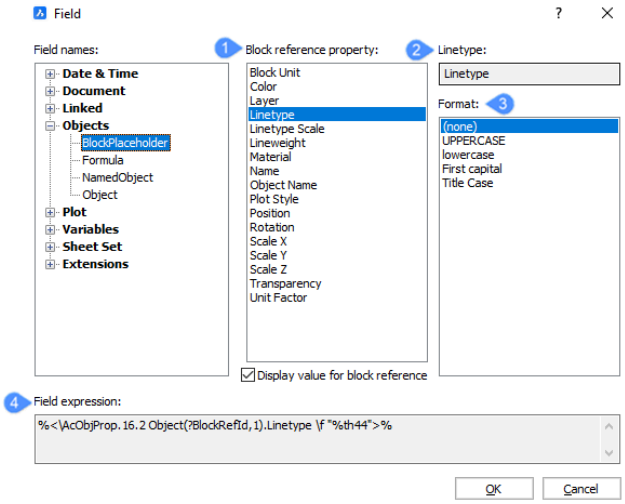
2 Formula

3 NamedObject

4 Object

BlockPlaceHolder

See the ATTDEF command to create block property fields in an attribute. When such attribute is included in a block definition, the field displays the current value of the block property.



1 Block reference property

2 Block name

3 Format

4 Field expression

- **Block reference property**: displays the block reference property.
- **Block name**: displays the name of the block.
- **Format**: displays the name of the block in different letter cases.

- **Field expression**: displays the field expression of the selected date. An operand in an expression can be a DisplayPropertyName enclosed in arrow brackets (<>).

Formula
Creates a formula field. You can use the values of table cells in a formula field. Click the **Average**, **Sum**, **Count** or **Cell** button. The **Field** dialog closes to let you select cells in a table in the drawing.

NamedObject
Creates a field that shows the current name of a named entity.

- Click the **Named object type** field, then select the object type in the list.

  All named objects of the selected type are listed in the **Name list** box.
- Select a name in the list.
- Choose a format.
- Click the **OK** button to place the field.

Object
Creates a field that shows a property of a selected entity.

- Click the select button, next to the **Object type** field.

  The **Field** dialog temporarily closes.
- Select an entity in the drawing.
- The properties of the selected entity are listed in the Property list.
- Select a property in the list.
- Choose a format.
- Click the **OK** button to place the field.

4.12.69.6 Plot

Plot fields apply to paper space layouts.

Allows to create fields which show the current print settings of a layout.

4.12.69.7 Variables

Allows to create fields which show the current value of Diesel Expressions, Lisp Variables and System Variables.

4.12.69.8 Sheet Set

Allows to create fields which show default and custom sheet set properties.

4.12.69.9 Extensions

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

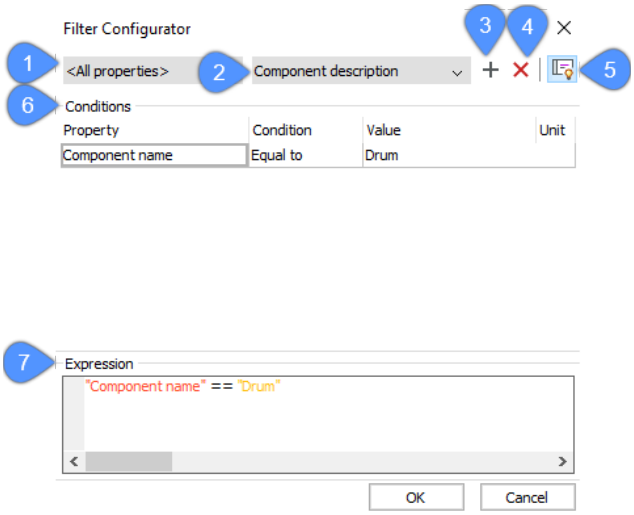
4.12.69.1 Field expression

0

Displays the expression of the field.

4.12.70 Filter Configurator dialog box

The Filter Configurator dialog box allows you to specify the conditions to filter components in the BOM table.



1 Columns

2 Properties

3 Add

4 Delete

5 Expression

6 Conditions

7 Expression Editor

4.12.70.1 Columns

Selects columns to apply the filter to.

4.12.70.2 Properties

Specifies the available properties that can be used to apply the filter.

Nota: When specified the options above, the condition can be added by pressing the plus sign.

Filter conditions

Displays the conditions were added. This includes the properties, it's condition, value and unit.

4.12.70.3 Add

Adds a condition for the selected property.

4.12.70.4 Delete

Deletes the selected condition(s) from the filter.

4.12.70.5 Expression

Shows or hides the expression editor.

4.12.70.6 Conditions

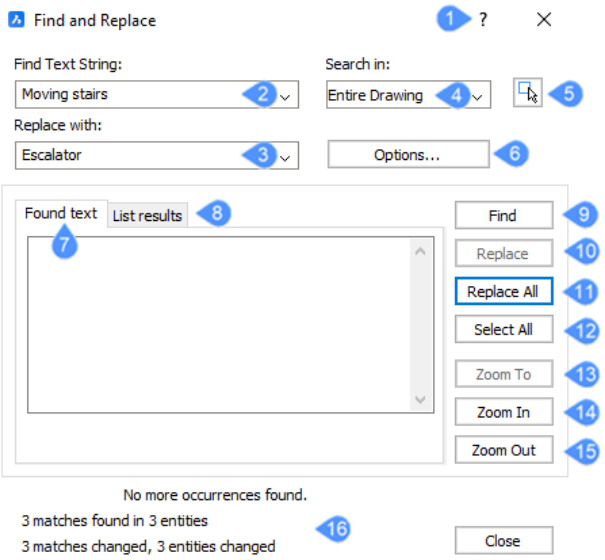
Displays all the added conditions that will be used to filter the content of the BOM table.

4.12.70.7 Expression Editor

Displays the Expression Editor where you can configure and preview the filtering expression.

4.12.71 Find and replace dialog box

The **Find and Replace** dialog box allows you to search for text in blocks, attributes, dimensions and hyperlinks, and optionally to replace it with another text string.



1 Command reference

2 Find Text

3 Replace With

4 Search In

5 Select Entities

6 Options

7 Found text

8 List results

9 Find

10 Replace

11 Replace All

12 Select All

13 Zoom To

14 Zoom In

15 Zoom Out

16 Search report

4.12.71.1 Command reference

Opens the Bricsys Help article about the FIND command.

4.12.71.2 Find Text

Specifies the text to be located.

4.12.71.3 Replace With

Specifies the text with which to replace the located text. Do not fill in this field when only searching for text.

4.12.71.4 Search In

Specifies the range of search:

- **Entire Drawing**: searches all text in the drawing.
- **Current Layout**: searches only among the selected entities.
- **Current Selection**: searches only in the current selection that can be made using button no. 5.

4.12.71.5 Select Entities

Dismisses the dialog box so that you can select the entities to search. After choosing one or more entities, press ENTER to return to the dialog box.

4.12.71.6 Options

Opens the **Find and Replace Options** dialog box that allows you to specify the kinds of text entities to search.

4.12.71.7 Found text

Displays the text string found.

4.12.71.8 List results

Displays the text string found, its location and the object type.

4.12.71.9 Find

Finds the next occurrence of the text string.

Shortcut Keystroke: CTRL+F.

4.12.71.1 Replace
0

Replaces the found text string with the replacement string.

Shortcut Keystroke: CTRL+R.

4.12.71.1 Replace All

1

Replaces all occurrences of the text string with the replacement string.

Shortcut Keystroke: CTRL+A.

4.12.71.1 Select All

2

Selects all occurrences of the text string.

4.12.71.1 Zoom To

3

Zooms to the part of the drawing containing the found text string.

4.12.71.1 Zoom In

4

Zooms in on the center of the window by a factor of 2.

4.12.71.1 Zoom Out

5

Zooms out from the center of the window by a factor of ½.

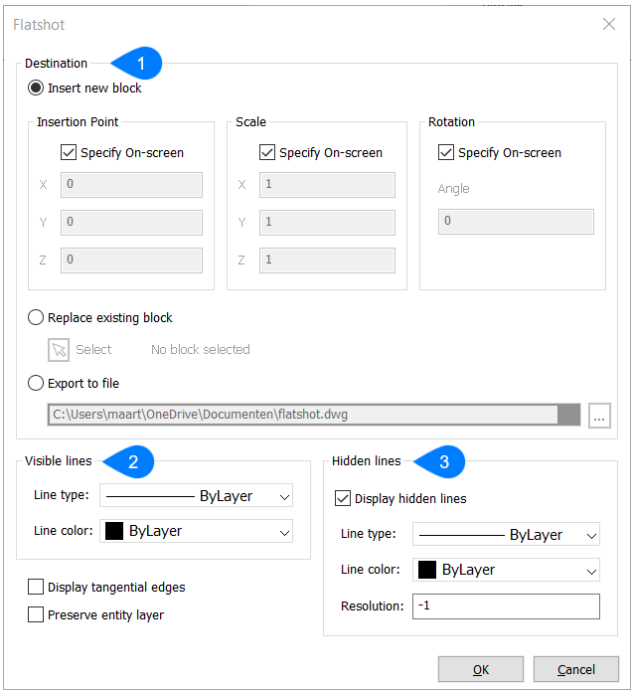
4.12.71.1 Search report

6

Displays the number of matches found in the selected entities, but also the number of matches and entities changed after replacement.

4.12.72 Flatshot dialog box

The Flatshot dialog box allows you to create a flattened, hidden line representation of 3D solids, 3D surfaces, and polyface meshes as a block or a new drawing.



1 Destination

2 Visible lines

3 Hidden lines

4.12.72.1 Destination

Determines where the block should be placed. Several options are available

Insert new block
Inserts the flatshot block as a new block in the current drawing by specifying some characteristics

Replace existing block
Replaces a block already inserted in the drawing.

Export to file
Saves the flatshot block as a dwg-file trough the Select Export File dialog box.

4.12.72.2 Visible lines

Sets the linetype and color for the visible lines. Choose the linetype and color from the drop lists.

Nota: A new linetype can be loaded into the drawing trough the Load Linetypes dialog box by selecting Load. A new line color can be selected trough the Select Color dialog box by selecting Select color.

4.12.72.3 Hidden lines

Toggles the display of hidden lines and sets their properties.

Display hidden lines
Toggles the display of hidden lines.

Resolution
Resolution specifies the short entity that will be represented by a hidden line. Enter a negative value so that the program will determine this automatically. The range is 0.01 to 0.000000000001.

4.12.72.4 Display tangential edges

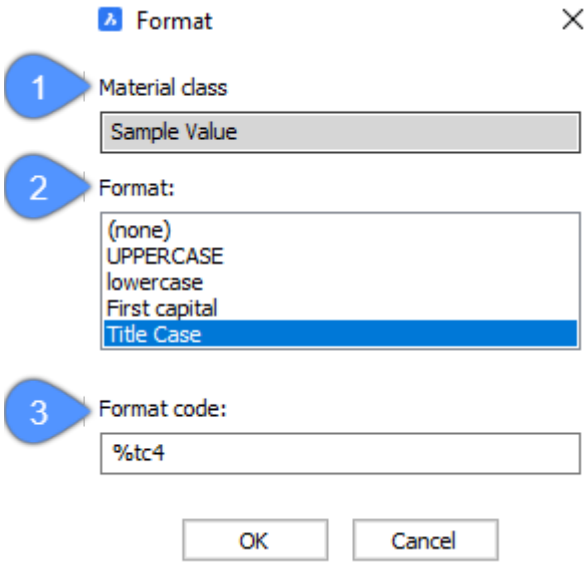
Toggles the display of tangential edges. A tangential edge is the imaginary transition line between two tangent faces.

4.12.72.5 Preserve entity layer

When on, maintains the layers of entities in the block.

4.12.73 Format dialog box

The Format dialog box allows you to specify the format of text strings in the specified column of a BOM table.



1 Material class

2 Format

3 Format code

4.12.73.1 Component

Specifies a component.

4.12.73.2 Format

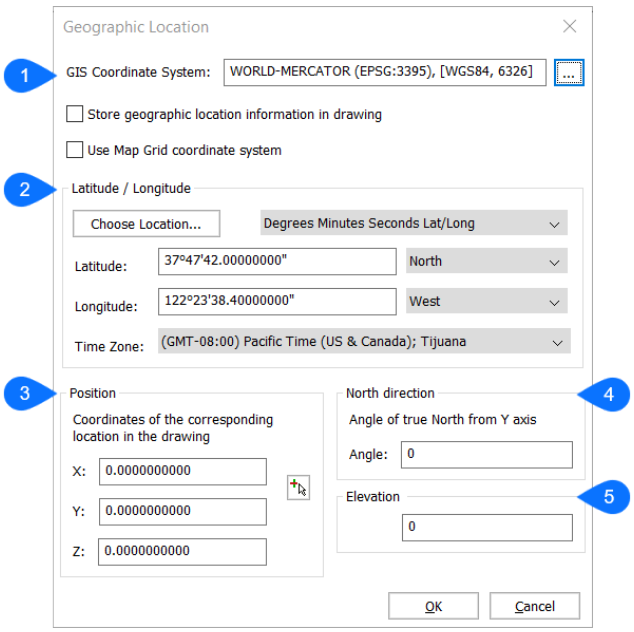
Specifies the desired format for the selected component from the list.

4.12.73.3 Format code

Changes depending on the format type you choose for all the materials in the BOM table.

4.12.74 Geographic location dialog box

The Geographic Location dialog box allows you to set the geographic location of the drawing by applying the longitude and latitude to a point in the drawing.



1 GIS Coordinate System

2 Latitude / Longitude

3 Position

4 North direction

5 Elevation

4.12.74.1 GIS Coordinate System

Reports the GIS system being used by the drawing.

Nota: Click the browse icon to change the system.

Store geographic location information in drawing
When checked, a red dot indicates the position in the drawing.

Use Map Grid coordinate system
Uses the location data from the GIS Coordinate System.

4.12.74.2 Latitude / Longitude

Choose Location
Specifies the graphic location trough the Choose Geographic Location dialog box.

Degrees Minutes Seconds Lat/Long
Displays the location in a Degrees, Minutes and Seconds format.

Decimal Lat/Long
Displays the location in a decimal format.

Latitude
Sets the altitude. The valid range is 0 to 90.

Longitude
Sets the longitude. The valid range is 0 to 180.

Time zone
Specifies the time zone from the Time Zone drop list.

Nota: This setting is saved to the TIMEZONE system variable.

4.12.74.3 Position

Specifies the position of the location in the drawing by entering coordinate values.

Nota: You can also select a specific point in the workspace to set the position.

4.12.74.4 North direction

Specifies the angle of the sun from north, in the context of the world coordinate system.

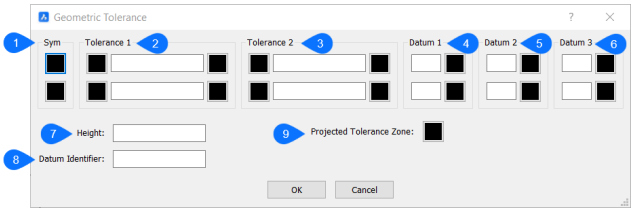
Nota: The setting is saved to the NORTHDIRECTION system variable.

4.12.74.5 Elevation

Specifies an elevation for the long-lat point. The value can be positive or negative.

4.12.75 Geometric tolerance dialog box

The Geometric Tolerance dialog box allows you to place tolerance symbols in drawings.



1 Symbol

2 Tolerance 1

3 Tolerance 2

4 Datum 1

5 Datum 2

6 Datum 3

4.12.75.1 Symbol

Specifies a tolerance symbol trough the Symbol dialog box.

4.12.75.2 Tolerance 1 & 2

Specifies the tolerance specifications (diameter, value and material condition).

Diameter
Toggles the diameter symbol.

Value
Specifies the tolerance value.

Material condition
Specifies the material condition trough the Material condition dialog box.

4.12.75.3 Datum 1, 2 & 3

Specifies the datum reference (value and material condition).

Value
Specifies the datum value.

Material condition
Specifies the material condition trough the Material condition dialog box.

Height
Specifies the height of the tolerance symbols.

Datum Identifier
Specifies datum identifier, such as Datum A.

Projected Tolerance Zone
Toggles the projected tolerance zone symbol.

Tolerance Symbols

Symbol	Characteristic	Type
	Position	Location
	Concentricity or Coaxiality	Location
	Symmetry	Location
	Parallelism	Orientation
	Perpendicularity	Orientation
	Angularity	Orientation
	Cylindricity	Form

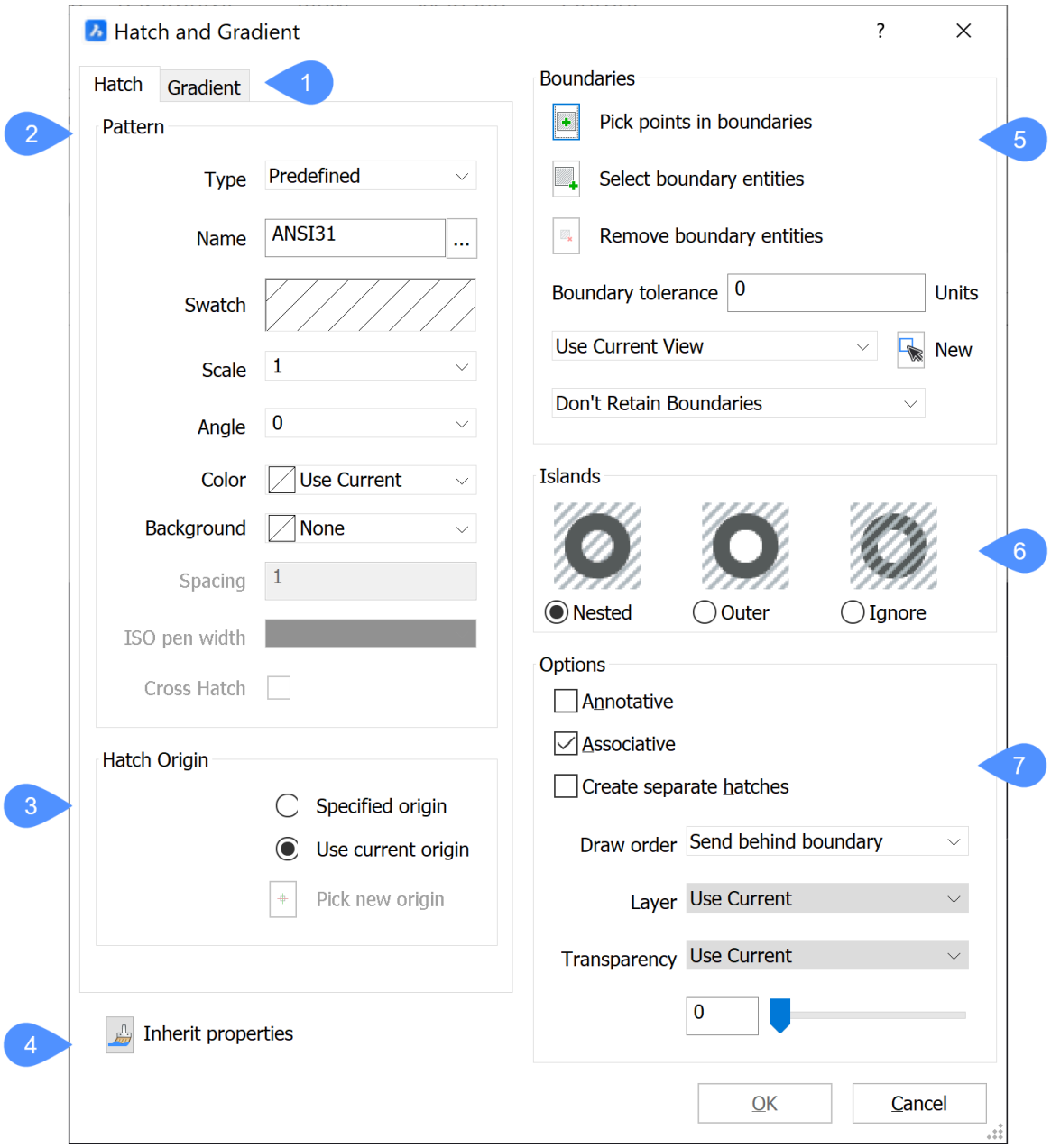
Symbol	Characteristic	Type
	Flatness	Form
	Circularity or Roundness	Form
	Straightness	Form
	Profile of a surface	Profile
	Profile of a line	Profile
	Circular Run-out	Run-out
	Total Run-out	Run-out

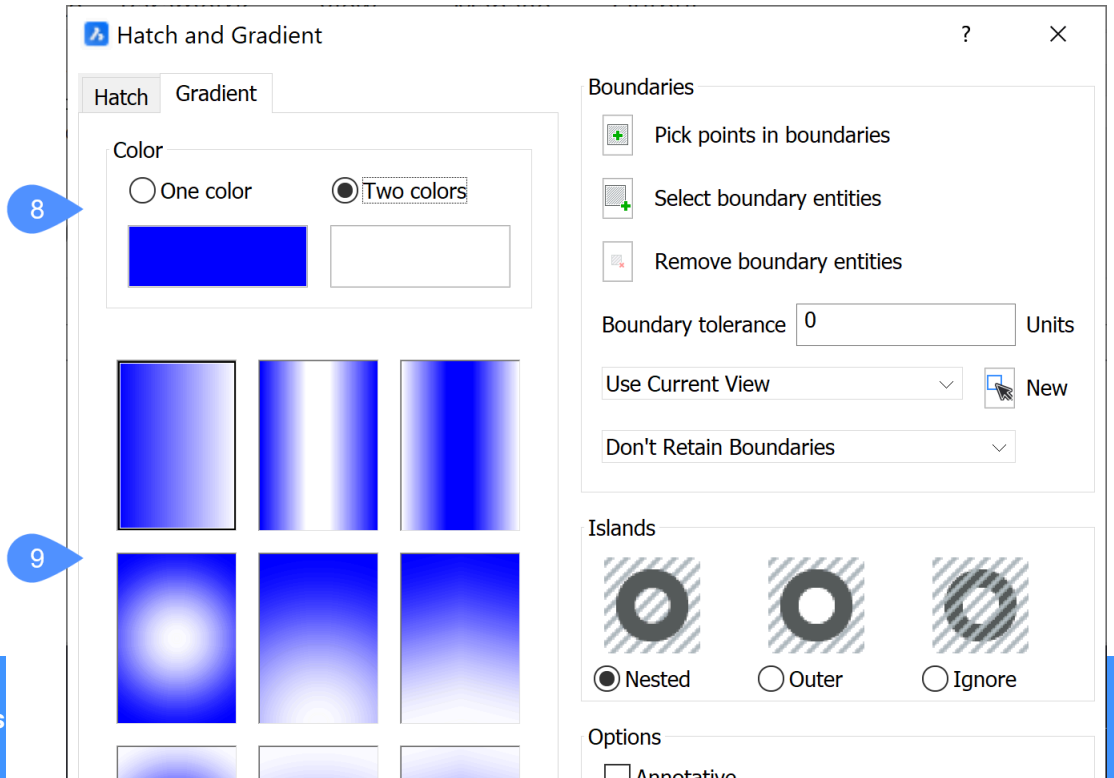
Material Condition Symbols

Symbol	Definition
Ⓜ	At maximum material condition (MMC), a feature contains the maximum amount of material stated in the limits.
Ⓛ	At least material condition (LMC), a feature contains the minimum amount of material stated in the limits.
Ⓢ	Regardless of feature size (RFS) indicates that the feature can be any size within the stated limits.

4.12.76 Hatch and gradient dialog box

The Hatch and gradient dialog box allows you to fill closed 2D areas with repeating patterns or solid colors.


Hatch and Gradient
Hatch
Gradient
1
2
Pattern
Type
Predefined
Name
ANSI31
Swatch
Scale
1
Angle
0
Color
Use Current
Background
None
Spacing
1
ISO pen width
Cross Hatch
Hatch Origin
3
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
Annotative

2 Hatch Pattern

3 Hatch origin

4 Inherit properties

5 Boundaries

6 Islands

7 Options

8 Gradient Color

9 Gradient pattern

10 Gradient Orientation

4.12.76.1 Pattern

Specifies the hatch pattern properties.

Type
Specifies the type of hatch.

User defined
Constructs the pattern from the user-defined parameters of Angle, Spacing, Color, Background and Cross Hatch.

Predefined
Uses the patterns defined in the iso.pat (metric units) or default.pat (imperial units) pattern definition files provided with the program.

Custom
Uses a pattern defined by *.pat files (limited to one pattern definition per file) created by users. The program searches for *.pat files in the folders that are defined in the SRCHPATH system variable.

Name
Specifies the name of the predefined hatch pattern by one of these methods. Click the Browse button to display the Hatch Pattern Palette dialog box, to choose a pattern.

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

Swatch
Displays the Hatch Pattern Palette dialog box.

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

Scale
Sets the scale factor of the pattern. To use an enlarged hatch pattern, enter a value bigger than 1.

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

Angle
Sets the angle of the pattern.

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

Color
Sets the color of the hatch pattern lines.

Nota: For the Use Color option, the current color specified by the CECOLOR system variable is used.

Background
Sets the color of the background of the hatch.

Spacing
Specifies the distance between hatch lines.

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

ISO pen width
Specifies the lineweight.

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

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

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

4.12.76.2 Hatch origin

Specifies the origin of the hatch. Either the current or a new specified origin can be chosen.

4.12.76.3 Inherit properties

Copies the properties of another existing hatch pattern to use with this pattern.

4.12.76.4 Boundaries

Specifies the boundary conditions of the hatch.

Pick points in boundaries
Specifies the closed areas (boundaries) in which to place the pattern.

Nota: You cannot pick an area that already contains a hatch pattern, or areas that are not closed (whose gap is larger than the value specified by Boundary Tolerance).

Select boundary entities
Selects the entities that make up the hatching boundary to constrain the extent of the pattern.

Nota: This option allows you to hatch closed areas, including ones that already contain a hatch. It hatches open areas whose gap is less than the value specified by Boundary Tolerance.

Remove boundary entities
Remove entities from the detected boundary set.

Boundary tolerance
Specifies the largest gap the program ignores when hatching a boundary that is not fully closed.

- 0 - (Default): the tolerance is set by the application, based on the current view size. When zoomed in closely, boundary detection will fail; when zooming out further so the contour 'looks' closed, the boundary will become detected
- Any Value : defines the maximum gap in drawing units.

Nota: The value is saved in the HPGAPTOL system variable.

Use current view / boundary set
Specifies where the program should search for entities that make up the boundary.

New
Creates a new selection set of entities that make up the pattern boundary.

Nota: The next step is to click the Pick Points in Boundaries button to choose the hatch area.

Retain boundaries
Determines what happens to boundaries.

Don't Retain Boundaries
Removes the boundary after the hatch is created.

Retain Boundaries as Polylines
Retains boundaries and turns them into polylines.

Retain Boundaries as Regions
Retains boundaries and turns them into regions (Pro or Platinum versions only).

4.12.76.5 Islands

Specifies how BricsCAD will respond when other boundaries are present within the closed hatch boundary.

Nested
When a closed hatch boundary contains other boundaries, BricsCAD hatches alternating areas.

Outer
When a closed hatch boundary contains other boundaries, BricsCAD hatches only the outermost area.

Ignore
When a closed hatch boundary contains other boundaries, BricsCAD hatches all interior areas, as if they were not present.

4.12.76.6 Options

Annotative
Toggles the Annotative property of the hatch. When turned on, BricsCAD applies the current annotative scale.

Nota: When this option is turned on, the Associative option is unavailable.

Associative
Toggles the associativity of hatch patterns: when the boundary is changed, the pattern automatically updates itself.

Nota: This option is not available for annotative hatches.

Create separate hatches
Toggles how multiple hatches are treated. If turned on, BricsCAD creates a separate hatch entity for each boundary in the selection set.

Draw order
Specifies where the pattern should be placed relative to overlapping entities.

Layer
Specifies the layer on which the hatch should be placed.

Transparency
Set the transparency property of the hatch.

Nota: For the Use current option, the transparency value, defined by the CETRANSPARENCY system variable, is applied.

4.12.76.7 Gradient Color

Specifies the gradient color. The gradient can either exist of one color to create a solid fill hatch or exist out of 2 colors.

4.12.76.8 Gradient pattern

Specifies the pattern of the gradient.

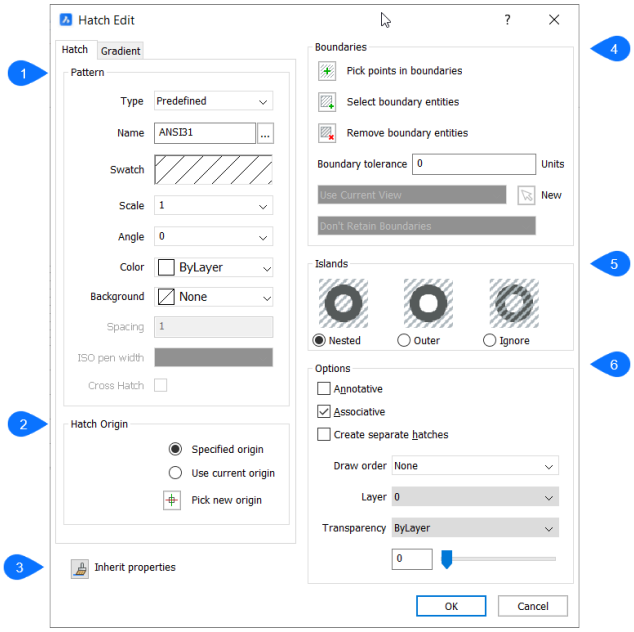
4.12.76.9 Gradient Orientation

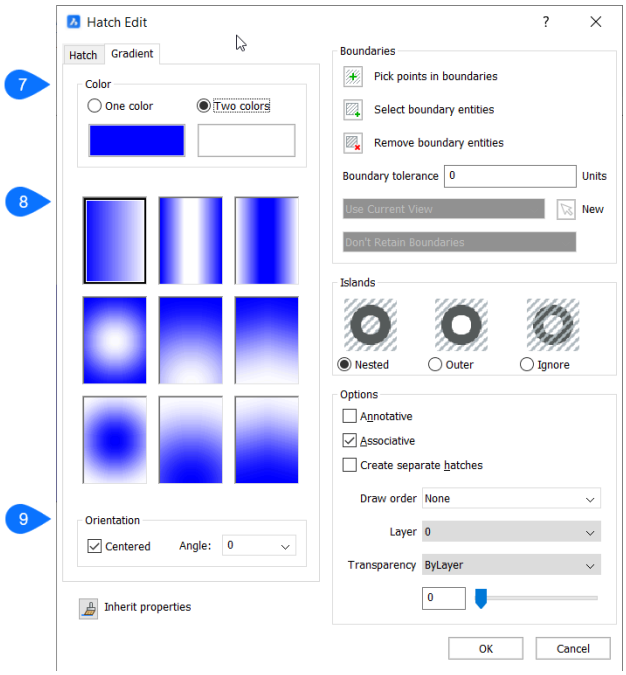
Specifies how the fill is placed in the boundary.

- Centered: centers gradient fill in the area to be filled; when off, starts the fill from the upper left of the boundary.
- Angle: select an angle of the gradient fill, relative to the current UCS; choose from increments on 15 degrees, or else enter any other value for the angle.

4.12.77 Hatch Edit dialog box

The Hatch Edit dialog box allows you to edit an already existing hatch in the drawing. All options resemble the options of the Hatch and Gradient dialog box.





1 Pattern

2 Hatch origin

3 Inherit properties

4 Boundaries

5 Islands

6 Options

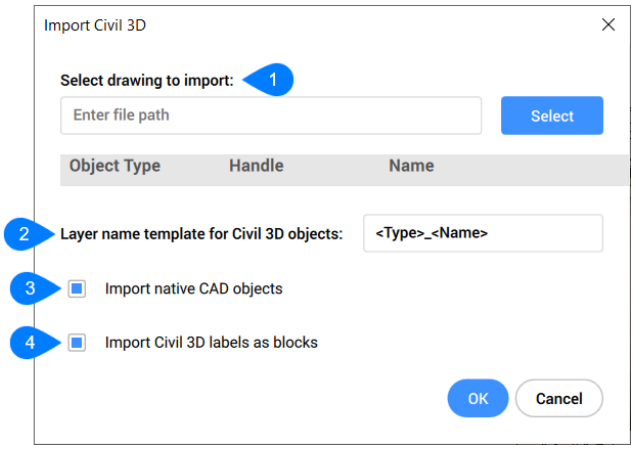
7 Color

8 Pattern

9 Orientation

4.12.78 Import Civil 3D dialog box

The Import Civil 3D dialog box allows you to create BricsCAD civil entities from an Autodesk® Civil 3D drawing.



1 Select drawing to import

2 Layer name template for Civil 3D objects

3 Import native CAD objects

4 Import Civil 3D labels as blocks

4.12.78.1 Select drawing to import

Specifies the drawing to import by clicking the Select button, which opens the Open File dialog box.

4.12.78.2 Layer name template for Civil 3D objects

Edits the template that is used for the layer name of all Civil 3D objects.

4.12.78.3 Import native CAD objects

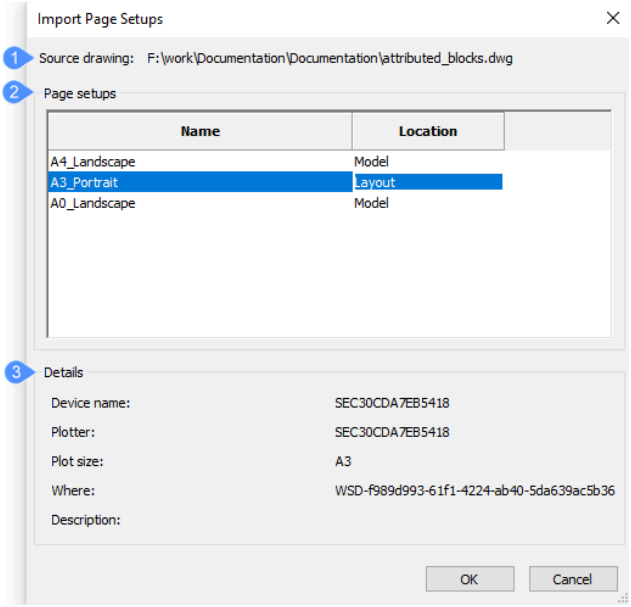
Toggles the option to also import native CAD objects.

4.12.78.4 Import Civil 3D labels as blocks

Toggles the option to import all Civil 3D labels as blocks.

4.12.79 Import page setups dialog box

The **Import Page Setups** dialog box allows you to select one or more page setups (using the CTRL key) from the list and import them into the current drawing.



1 Source Drawing

2 Page setups list

3 Details

4.12.79.1 Source Drawing

Displays the source of the drawing used to import the page setup(s).

4.12.79.2 Page setups list

Specifies the name and the location of each page setup from the list:

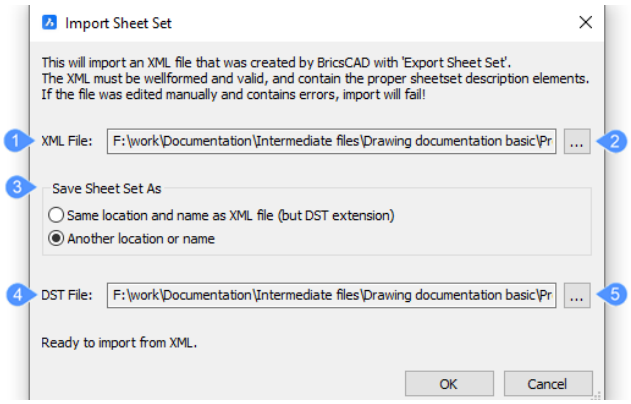
- **Name**: specifies the names of page setups in the drawing. When the drawing contains no page setups, then the dialog box is blank.
- **Location**: specifies the layout and model locations of page setups.

4.12.79.3 Details

Includes information about the device name, plotter, plot size, location and a description field.

4.12.80 Import sheet set dialog box

The **Import Sheet Set** dialog box allows you to import a sheet set file that has been created using the **Export Sheet Set As** dialog box.



1 XML File path

2 Select XML File

3 Save Sheet Set As

4 DST File path

5 Save Sheet Set As

4.12.80.1 XML File path

Displays the path of the *.xml file after selecting it using the **Select XML File** button.

4.12.80.2 Select XML File

Opens the **Select XML File** dialog box.

4.12.80.3 Save Sheet Set As

Choose whether to save the imported XML file as DST file using:

- The same location and name as XML file. Sections named 4 and 5 are unavailable.
- Another location or name. For the second option, click the button number 5.

4.12.80.4 DST File path

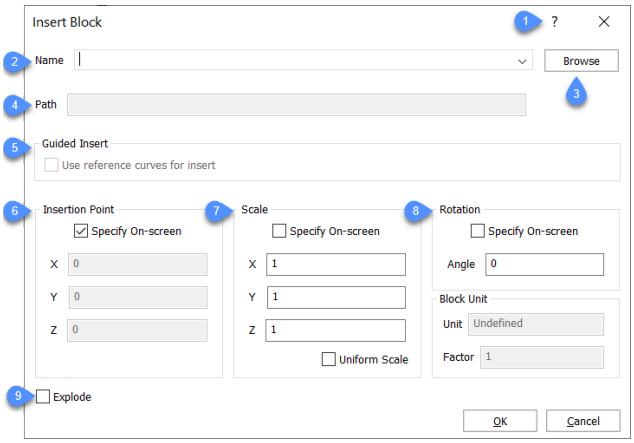
When you choose to save the imported XML file as DST file to a new location or name is displayed the path of this file.

4.12.80.5 Save Sheet Set As

Opens the **Save Sheet Set As** dialog box.

4.12.81 Insert block dialog box

The **Insert Block** dialog box allows you to insert blocks into the current drawing. You can also insert entire DWG and DXF files as blocks into the drawing.



Nota: If the block contains attributes, then you are prompted to enter values for the attributes in the Command line or through **Edit Attributes** dialog box, depending on the value of the value of the ATTDIA (attribute dialog) variable.

1 Command reference

2 Name

3 Browse

4 Path

5 Guided Insert

6 Insertion Point

7 Scale

8 Rotation

9 Explode

4.12.81.1 Command reference

Opens the Bricsys Help article about the INSERT command.

4.12.81.2 Name

Specifies the name of the block, whose definition exists in the drawing, or is a DWG or DFX file on the computer or network.

4.12.81.3 Browse

Selects a DWG or DXF file from your computer or network. Opens the **Select Drawing File** dialog box.

4.12.81.4 Path

Displays the path to the block, if the block was opened from a DWG or DXF file.

4.12.81.5 Guided Insert

When a dwg is selected that contains guide curves, the **Guided Insert** option becomes selectable. It is used for the insertion of parametric blocks.

- **Use reference curves for insert**: Specifies if a reference should be used for insert or not.

4.12.81.6 Insertion Point

Specifies the insertion point of the block in the drawing.

Nota: Insertion points are typically chosen at the lower-left corner of the block, in the center of the block, or at entity in the block that represents a connection point.

4.12.81.7 Scale

Scales the block:

- Values larger than 1 make the block larger.
- 1 inserts the block at actual size.
- Values smaller than 1 make the block smaller.
- Values less than zero flip the block, like mirroring it.
- **X, Y, Z**: Defines the scaling of the block:
  - X specifies the scale factor along the x axis. Enter a negative value to mirror the block about the y axis.
  - Y specifies the scale factor along the y axis. Enter a negative value to mirror the block about the x axis.
  - Z specifies the scale factor along the z axis.
- **Uniform Scale**: Toggles whether the same scale factor is used for all axes.

4.12.81.8 Rotation

Specifies the rotation angle of the block about its insertion point, starting with the x-axis as 0 degrees.

Nota: Positive angles rotate the block counter-clockwise and negative angles rotate the block clockwise.

4.12.81.9 Block unit

Controls the automatic scaling of the block with respect to the INSUNITS system variable of the current drawing.

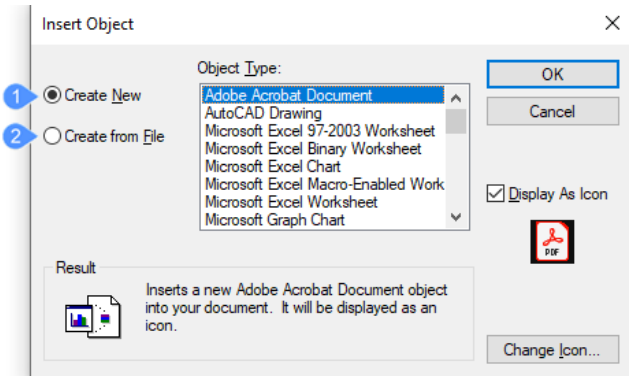
4.12.81.1 Explode

0

Toggles whether the block is inserted exploded.

4.12.82 Insert object dialog box

The Insert Object dialog box allows you to insert a linked or embedded document in the current drawing.

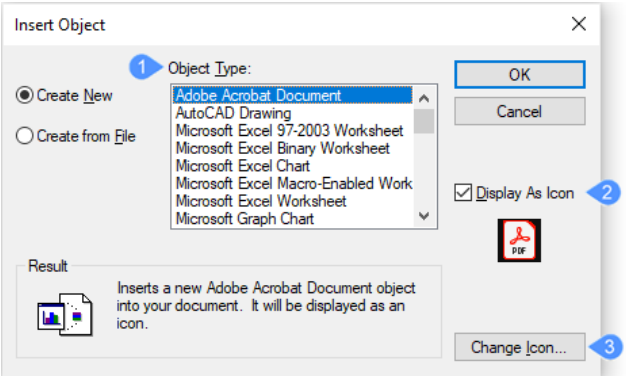


1 Create new Object

2 Create from File

Nota: The object types listed varies according the software installed on your computer.

4.12.82.1 Create new Object



1 Display as icon

2 Change icon

Display as icon
Displays the object as an icon. The icon is related to the document type and is defined by source application.

Change icon
Opens the Change Icon dialog box.

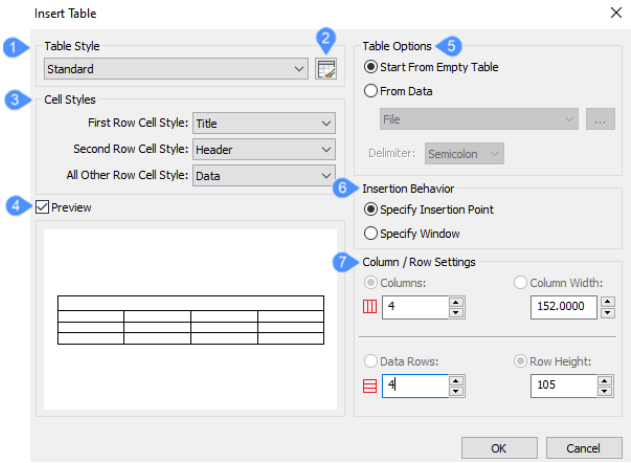
4.12.82.2 Create from File

Link
Toggles linkage to the source document:

- **On**: links the object in the drawing to the source file, so that when the source file is changed, the linked object also changes.
- **Off**: does link not the object.

4.12.83 Insert table dialog box

The **Insert table** dialog box allows you to create a table entity in the drawing either from scratch or from data in an external file or a data link.



1 Table Style

2 Table Style Explorer

3 Cell Styles

4 Preview

5 Table Options

6 Insertion Behavior

7 Column / Row settings

4.12.83.1 Table Style

Chooses an existing table style from the drop-down list.

4.12.83.2 Table Style Explorer

Allows you to create or edit table styles. Opens the **Table Styles** from the **Drawing Explorer** dialog box.

4.12.83.3 Cell Styles

A table style can have different sub-styles for the first two rows named **Title**, **Header** and **Data**:

- **First Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Title**. A **Title** row is the topmost row of the table.
- **Second Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Header**. A **Header** row is the first row of a column.
- **All Other Row Cell Style**: chooses a sub-style from the drop-down list, which by default is **Data**. A **Data** row is all other rows in the table.

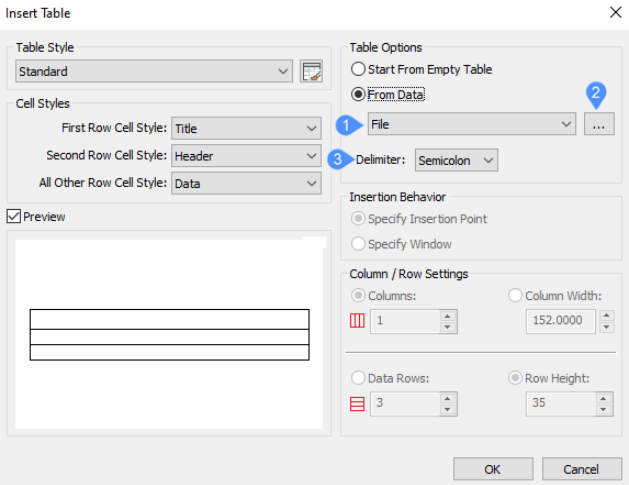
4.12.83.4 Preview

Displays a preview of the table depending on the options and settings you set in this dialog box.

4.12.83.5 Table Options

Determines the source of the new table:

- **Start from Empty Table**: creates a new, empty table with the parameters listed below.
- **From Data**: reads data from a file or a datalink.



a File

b Open

c Delimeter

- **File**

  Read data for the new table from a file.

- **Open**

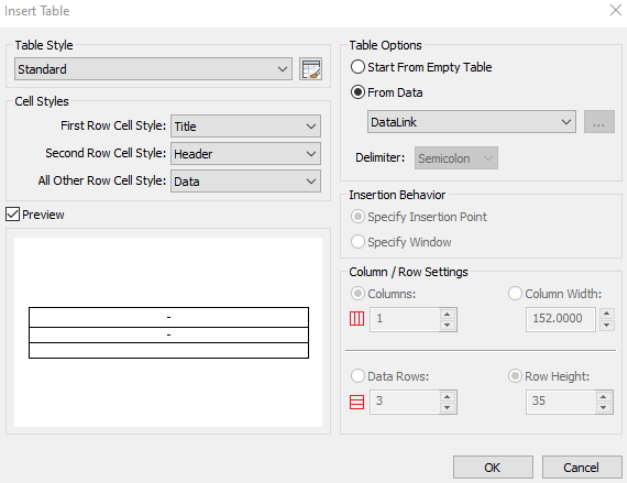
  Displays the **Open** dialog box from where you will select the file.

- **Delimeter**

  Specifies the delimiter character used to separate fields in CSV data files. Select from the drop-down list the delimiter you want to use.

  Nota: Available only for the **File** option.

- **DataLink**



When you select to read data from a datalink the **Datalink Manager** dialog box opens.

4.12.83.6 Insertion Behavior

(This option is available only when you create a new empty table.)

- **Specify Insertion Point**: specifies where the upper-left corner of the table will be inserted in the drawing.
- **Specify Window**: specifies the area that will be used by the table, this is useful when you need the table to fit a specific are, such as in a revision table or title block.

Nota: As you move the mouse, a fictive table dynamically resizes itself.

4.12.83.7 Column / Row settings

(This option is available only when you create a new empty table.)

Specifies the number of columns or rows and the cell width or height.

Choose one of:

- **Columns**: specifies the number of columns in the table
- **Column Width**: specifies the initial width of all columns

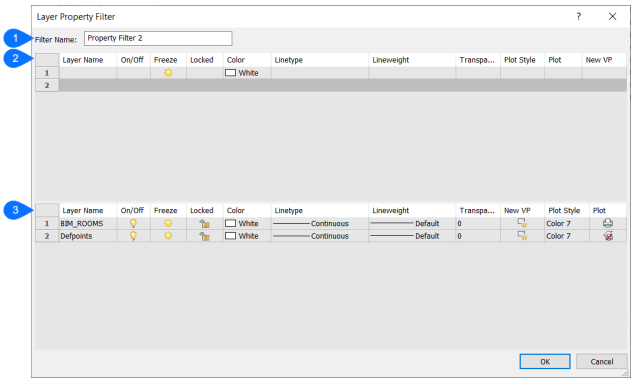
And choose one of:

- **Data Rows**: specifies the number of data rows in the table; the title and header rows are added automatically by the program
- **Row Height**: specifies the initial height of all rows.

The number of columns and rows and size of cells can be adjusted later, after the table is inserted.

4.12.84 Layer Property Filter dialog box

The Layer Property Filter Dialog Box allows you to create a new layer filter definition, based on the properties of the layer.



1 Filter Name

2 Filter Properties

3 Filtered Layers

4.12.84.1 Filter Name

Sets the name of the filter.

4.12.84.2 Filter Properties

Sets the properties to filter the layers. Layers that fulfill all indicated properties of the filter will be added to the layer filter.

Layer Name

Filters the layers based on their name. You can use wild cards to specify more than one layer, for example Doors-* for Doors-Exterior and Doors-Interior.

- # - matches any digit. For example, ## matches 12 and 34 but not 5 or 123 (ASCII code: 35).
- @ - matches any alphabetic character, so not digits or symbols. For example, @@@@@ matches all 5-letter layer names. (ASCII code: 64).
- . - matches non-alphanumeric characters, so symbols but not letters or numbers. For example, .* matches all layer names starting with a non-alphanumeric character such as _SITEPLAN and +1_LEVEL (ASCII code: 46).
- * - matches any string, including letters, numbers, and characters. For example, A* matches all layer names starting with the letter A (ASCII code: 42).
- ? - matches any single character. For example, ?AB* matches L**AB**ORATORY and C**AB**ARET but not ST**AB**LE, because STABLE begins with two letters and ? specifies a single letter (ASCII code: 63).
- ~ - negative match, in which any string but the pattern is returned. For example, ~*A* matches all layer names not containing the letter A (ASCII code: 126).
- [ ] - Matches any one of the characters enclosed inside the square brackets. For example, LA[YB] matches LA**Y**and LA**B**(ASCII code: 91 93).
- [~] - Specifies any one of the characters not found in the square brackets. For example, LA[~YB] matches L**AD**but not L**AY**or L**AB** (ASCII code: 91 126 93).

- [-] - Specifies a range for a single character in the layer name. For example, LAY-[A-E] matches any combination of LAY- and A through E, such as LAY-**A**... LAY-**E**but not LAY-F (ASCII code: 91 45 93).
- ‘ - Matches the next string exactly to find a layer name that contains a wild-card character when it is prefixed by the reverse quote character. Use ` to search for layer names that include characters like ? and ~. For example, `~LAY* matches ~LAY-1,~LAYER-1, and ~LAY-B (ASCII code: 96).

Nota: ASCII code: if a character is not available on the keyboard, press and hold the Alt key and then type the ASCII code on the numeric keypad.

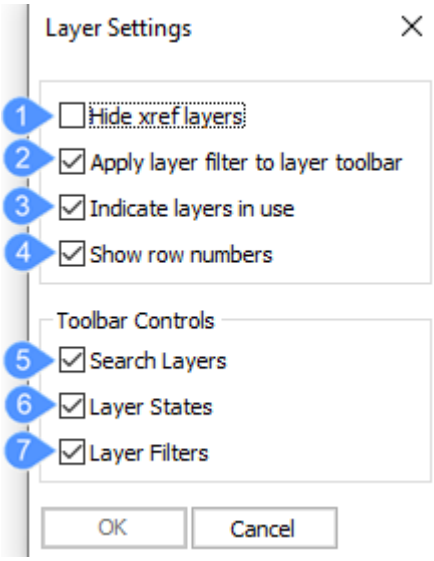
Properties
Specifies which layers to include according to their properties, such as freeze and color.

4.12.84.3 Filtered Layers

Displays the layers that fulfill the requirements of the filter.

4.12.85 Layer settings dialog box

The **Layer Settings** dialog box allows you to apply settings on the **Layers** Panel.



1 Hide xref layers

2 Apply layer filter to layer toolbar

3 Indicate layers in use

4 Show row numbers

5 Search Layers

6 Layer States

7 Layer Filters

4.12.85.1 Hide xref layers

When on, hide layers that came from externally referenced drawings.

4.12.85.2 Apply layer filter to layer toolbar

When on, shortens the list of layers displayed, according to the conditions of the filter. When off, displays all layer names.

4.12.85.3 Indicate layers in use

When on, displays icons in the **Current** column that indicate whether the layer has entities assigned to it. Unused layers can be purged.

4.12.85.4 Show row numbers

Toggles the display of the number at the start of each row.

4.12.85.5 Search Layers

Toggles the display of the **Search** field on and off.

4.12.85.6 Layer States

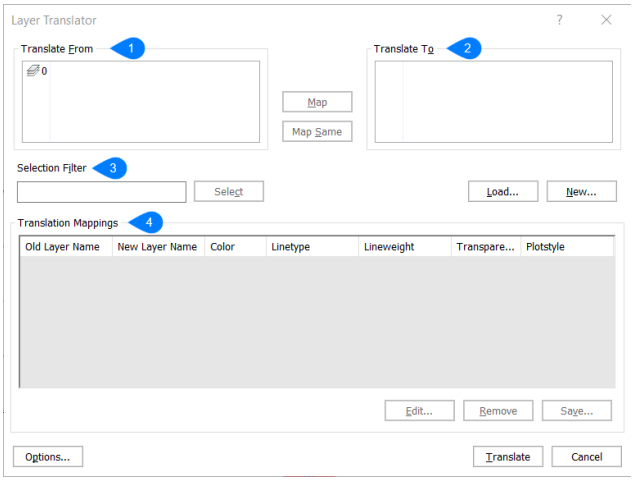
Toggles the display of the **Layer States** drop-down list.

4.12.85.7 Layer Filters

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

4.12.86 Layer translator dialog box

The Layer Translator dialog box allows you to apply layer names and properties from another drawing to the current drawing.



1 Translate from

2 Translate to

3 Selection filter

4 Translation mappings

4.12.86.1 Translate from

Lists all the layer names in the current drawing.

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

Nota: New layers can be created by simply entering the layer name and properties to which you want to map existing layer.

4.12.86.2 Translate to

Specifies which layer you want the chosen layer(s) to map to.

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

4.12.86.3 Translation mappings

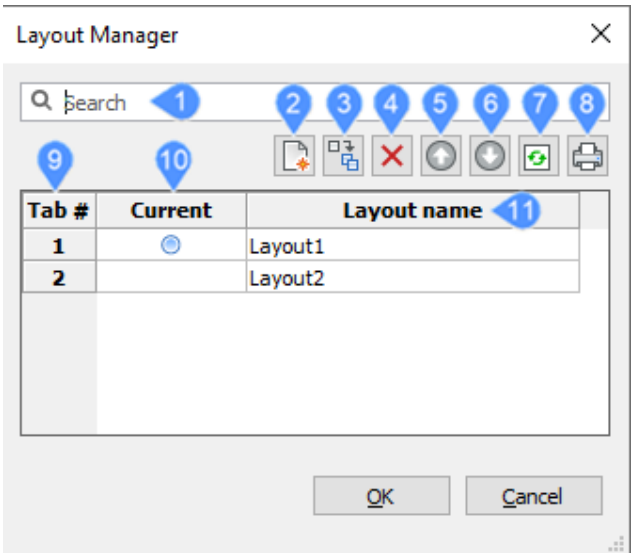
Displays an overview of which layer and its properties are translated to another layer.

4.12.86.4 Options

Displays the Options dialog box, which offers additional controls for layer mapping.

4.12.87 Layout manager dialog box

The Layout Manager dialog box allows you to create, copy and delete layouts.



1 Search

2 Add new layout

3 Copy selected layout

4 Delete

5 Move up

6 Move down

7 Clear selection

8 Publish

9 Tab

10 Current

11 Layout name

4.12.87.1 Search

Searches for a layout name and displays only the searched layout. This is useful when a drawing contains many layouts.

4.12.87.2 Add new layout

Creates a new layout.

4.12.87.3 Copy selected layout

Makes a copy of the selected layouts and adds them to the end of the list.

4.12.87.4 Delete

Deletes the selected layouts.

4.12.87.5 Move up

Moves the selected layouts up the list.

4.12.87.6 Move down

Moves the selected layouts down the list.

4.12.87.7 Clear selection

Removes highlighting from layout names, thereby unselecting them.

4.12.87.8 Publish

Opens the **Publish** dialog box.

4.12.87.9 Tab

Displays the number of the layout.

4.12.87.1 Current

0

Click in the **Current** column to make the layout current.

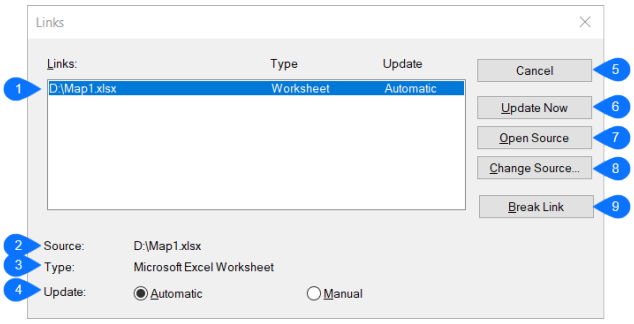
4.12.87.1 Layout name

1

Displays the name of the layout.

4.12.88 Links dialog box

The Links dialog box allows you to edit the links of OLE entities that were embedded or linked to drawings.



1 Links

2 Source

3 Type

4 Update

5 Cancel

6 Update Now

7 Open Source

8 Change Source

9 Break Link

4.12.88.1 Links

Provides a list of all links loaded in the file.

4.12.88.2 Source

Reports the path to the source file name.

4.12.88.3 Type

Reports the file type.

4.12.88.4 Update

Determines how the OLE entity in the drawing is updated when the source changes.

4.12.88.5 Cancel

Cancels the linking operation.

4.12.88.6 Update Now

Updates linked OLE entities.

4.12.88.7 Open Source

Opens the source file in the source application. This allows you to edit the source file.

4.12.88.8 Change Source

Changes the source for the linked OLE entity.

4.12.88.9 Break Link

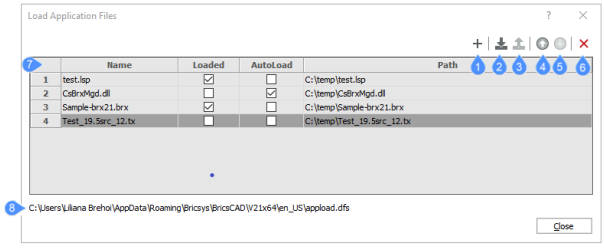
Breaks the link with the source file.

Nota: When the link is broken, the OLE entity in the drawing no longer changes when the source file changes.

4.12.89 Load application files dialog box

The **Load Application Files** dialog box allows you to load and unload the following applications type to run inside BricsCAD:

- LSP - LISP applications from BricsCAD and other CAD programs.
- DES - DES LISP applications.
- BRX - BricsCAD runtime extension applications.
- TX - Teigha extension applications, from Open Design Alliance.
- ARX - AutoCAD runtime extension applications from Autodesk.
- DBX - DBX applications.
- DRX - Drawing runtime extension applications.
- DVD - Visual Basic applications.
- MNL - LISP expressions for menu files.



1 Add application file

2 Load selected application

3 Unload selected application

4 Move application upward

5 Move application downward

6 Remove selected application

7 Application Files List

8 Application File Path

4.12.89.1 Add application file

Opens the **Select Application Files** dialog box.

4.12.89.2 Load selected application

Loads the selected application(s) into BricsCAD. Only compatible applications with BricsCAD will be loaded.

4.12.89.3 Unload selected application

Unloads the selected application(s) from BricsCAD. The LISP applications have to be removed in order to be unloaded in the current BricsCAD session.

4.12.89.4 Move application upward

Moves the selected application(s) up in the list.

4.12.89.5 Move application downward

Moves the selected application(s) down in the list.

4.12.89.6 Remove selected application

Removes the selected application(s) from the list.

4.12.89.7 Application Files List

Displays a list with application files that are available for load, unload, move up and down or remove.

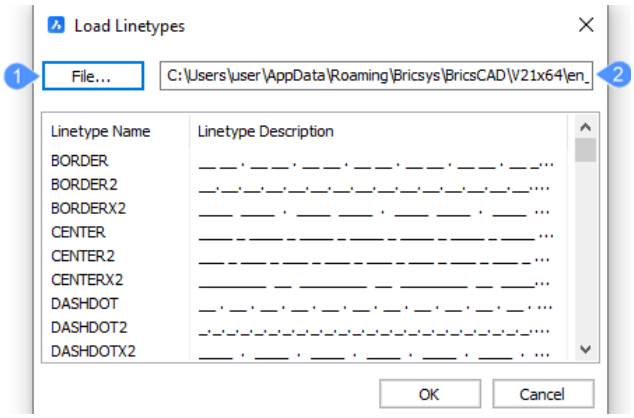
- **Name**: specifies the name and the extension of the application file from the list.
- **Loaded**: specifies if a file is loaded or unloaded in the current BricsCAD session.
- **Autoload**: specifies if a file will be loaded automatically in the next BricsCAD session.
- **Path**: displays the location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

4.12.89.8 Application File Path

Displays the location folder for appload.dfs file that contains all applications added, using the **Select Application Files** dialog box. You can open the file using Notepad.

4.12.90 Load linetypes dialog box

The **Load Linetypes** dialog box allows you to load additional linetype definitions.



1 File

2 File's path

4.12.90.1 File

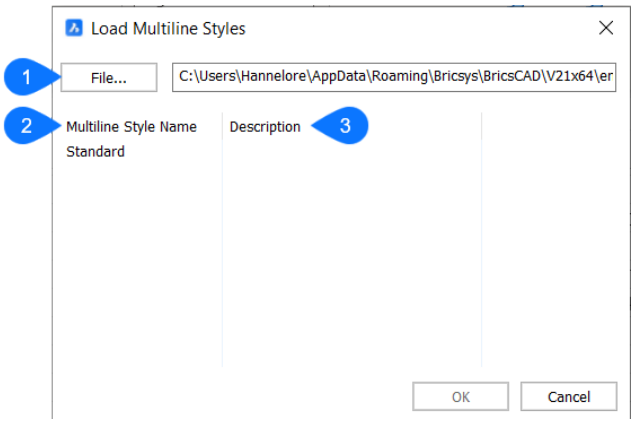
Opens the **Choose a Linetype File** dialog box.

4.12.90.2 File's path

Displays the path of the selected *.lin (linetype) file.

4.12.91 Load Multiline Styles dialog box

The Load Multiline Style dialog box allows you to load multiline styles from a multiline (*mln) file.



1 File

2 Multiline Style Name

3 Description

4.12.91.1 File

Displays the Choose a Multiline Style File dialog box where you can select a multiline style file to open.

4.12.91.2 Multiline Style Name

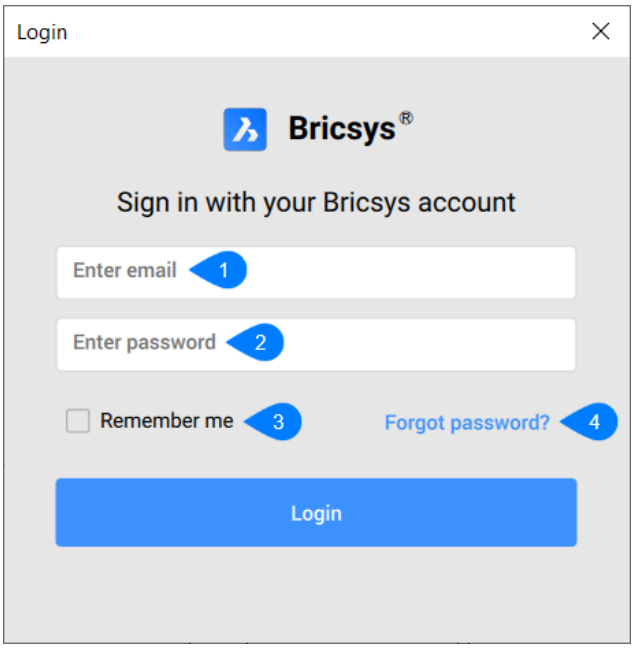
Displays the names of the available multiline style files in the drawing. You can select a multiline style file from this list.

4.12.91.3 Description

Displays a description of the multiline style files, if available.

4.12.92 Login dialog box

The Login dialog box allows you to login to BricsCAD with your personal Bricsys account.



1 Enter email

2 Enter password

3 Remember me

4 Forgot password?

4.12.92.1 Enter email

Specifies the email address connected to your Bricsys account.

4.12.92.2 Enter password

Specifies the password.

4.12.92.3 Remember me

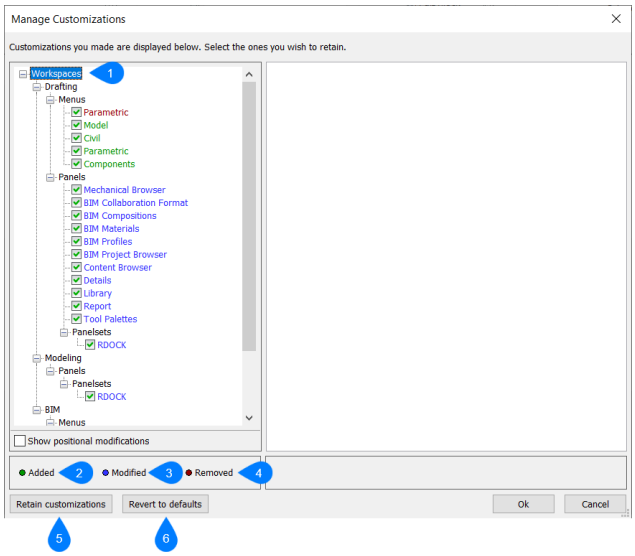
Check this box to keep being logged in.

4.12.92.4 Forgot password?

Redirects you to an internet browser to set up a new password.

4.12.93 Manage Customizations dialog box

The Manage Customizations dialog box provides a list of all customizations for confirming and reversing changes made to the user interface.



1 Customizations

2 Added

3 Modified

4 Removed

5 Retain customizations

6 Revert to defaults

4.12.93.1 Customizations

Lists all added, modified or removed customizations.

Nota: You can check a customization to retain the modification in the CUI.

4.12.93.2 Added

Items you have created display in green.

4.12.93.3 Modified

Items you have modified display in blue.

4.12.93.4 Removed

Items you have removed display in red.

4.12.93.5 Retain customizations

Keeps your local default CUI as it is.

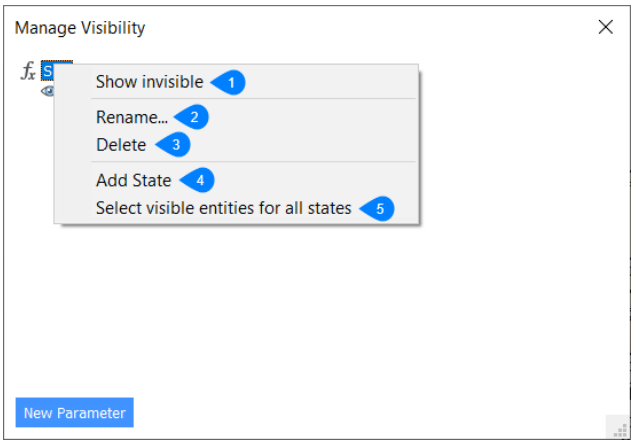
4.12.93.6 Revert to defaults

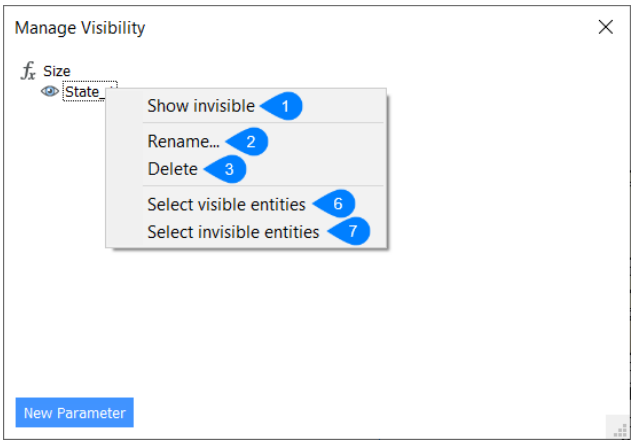
Reverts your local default CUI to the default one installed by BricsCAD in:

C:\Program Files\Bricsys\BricsCAD V21 en_US\UserDataCache\Support\en_US\default.cui.

4.12.94 Manage visibility

The Manage Visibility dialog box displays a list of visibility states in the drawing.





1 Show invisible

2 Rename

3 Delete

4 Add State

5 Select visible entities for all states

6 Select visible entities

7 Select invisible entities

4.12.94.1 Show invisible

Shows all invisible entities grayed-out.

4.12.94.2 Rename

Displays the Rename dialog box. Type a name to replace the existing name.

4.12.94.3 Delete

Deletes the parameter and its visibility states.

4.12.94.4 Add State

Displays the Rename dialog box. Type a name for the new state to replace the default name.

4.12.94.5 Select visible entities for all states

First choose Show invisible, then select the entities to remove all visibility states from the selected parameter.

4.12.94.6 Select visible entities

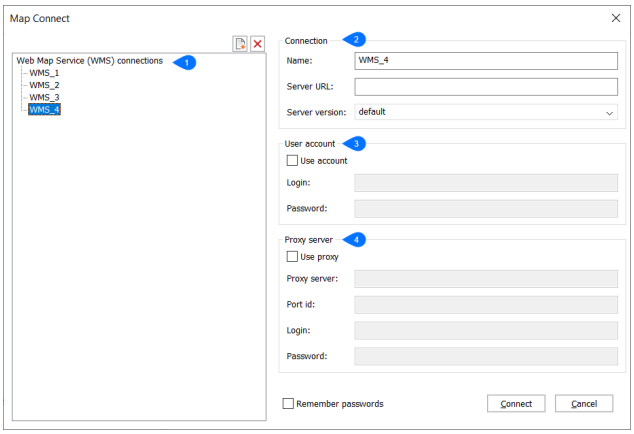
Visualizes all entities that are selected.

4.12.94.7 Select invisible entities

Hides all entities that are selected.

4.12.95 Map Connect dialog box

The Map Connectdialogboxallowsyoutoset up a connection with a Web Map Service.



1 WMS connections

2 Connection

3 User account

4 Proxy server

4.12.95.1 WMS connections

Displays a list of the different possible Web Map Service connections.

4.12.95.2 Connection

Allows you to create a Web Map Service (WMS) connection.

Name

Specifies the name for the new connection.

Server URL
Specifies the URL of the server you want to connect to.

Server version
Specifies the version of the server or else select one from the drop-down list.

4.12.95.3 User account

Allows you to log in to your account.

Use account
Toggles whether the login or password is used when making the connection.

Login
Specifies the login information.

Password
Specifies the password of your account.

4.12.95.4 Proxy server

Allows you to enter the information about a proxy server.

Use proxy
Toggles whether proxy is used to connect to the Web Map Server.

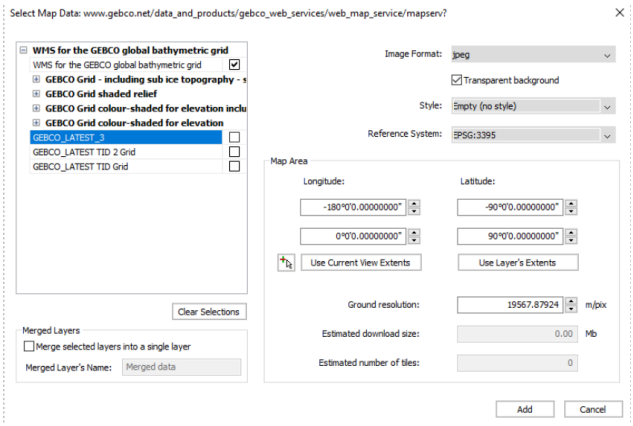
Proxy server
Specifies the URL of the proxy server.

Port id
Specifies the address of the proxy server.

Login
Specifies the login of the proxy server.

Password
Specifies the password of the proxy server.

Remember passwords
Check this box to keep being logged in. When is set to Off the program does not remember your passwords for greater security.
After connecting to a Web Map Server, the following dialog box is displayed:



Do the following:

1 Select one or more data layers.

2 Specify the desired ground resolution and the preferred maximum dimensions of a tile.

3 An estimate of the download size displays in the Estimated download size field.

4 Press the **Add** button.

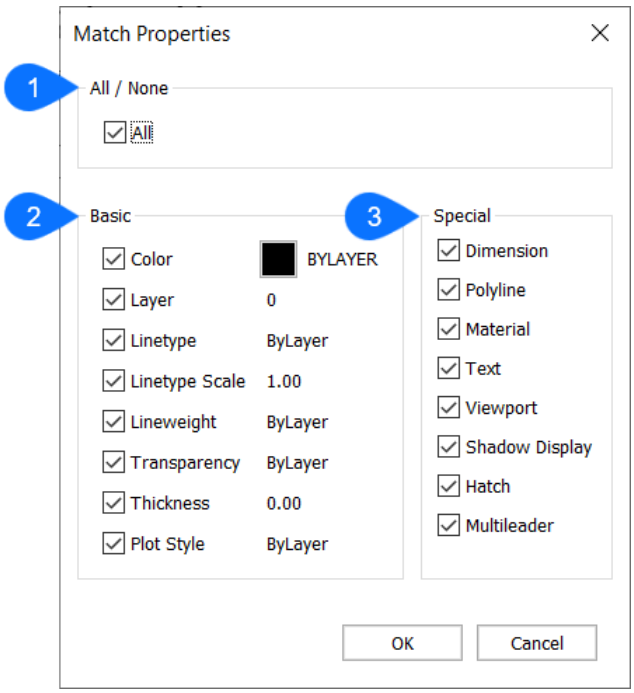
The program stores downloaded map tiles as regular image files in the folder in which the drawing is saved.

The definition of the supported Coordinate Reference Systems (CRS) is stored in the new geodatabase.xml file located in the folder pointed to by the ROAMABLEROOTPREFIX variable, such as *C:\Users\John\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support*

The geodatabase.xml files lists cities and their WGS84 geo coordinates used by the **Choose Location** function of the Geographic Location dialog box.

4.12.96 Match Properties dialog box

The Match Properties dialog box allows you to select which properties you want to copy from one entity to one or more other entities.



1 All/None

2 Basic

3 Special

4.12.96.1 All/None

When checked on, all properties are automatically selected. When checked off, all entities are deselected.

4.12.96.2 Basic

Selects the basic entity properties to copy. On the right side, the entity properties of the source entity are displayed.

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

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

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

Linetype Scale
When checked on, the linetype scale of the source entity will be copied to the selected entities. The linetype scale of the source entity is displayed on the right.

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

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

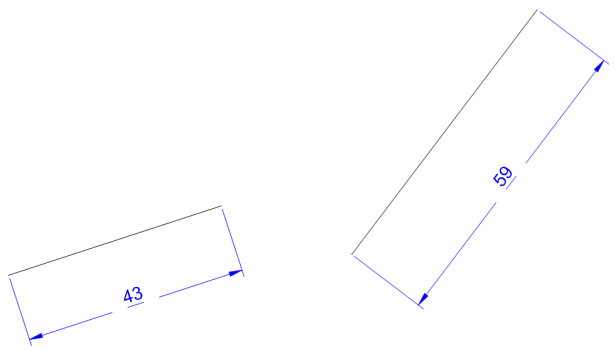
Thickness
When checked on, the thickness of the source entity will be copied to the selected entities. The thickness of the source entity is displayed on the right.

Plot Style
When checked on, the plot style of the source entity will be copied to the selected entities. The plot style of the source entity is displayed on the right.

4.12.96.3 Special

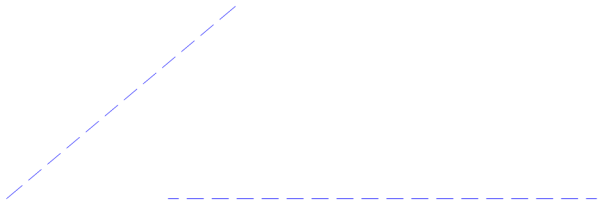
Selects other less basic properties to copy.

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



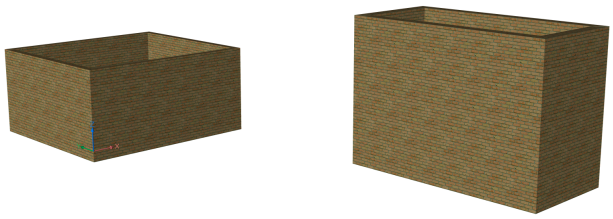
Polyline

When checked on, the width and linetype generation settings are copied. Other settings are ignored, such as taper and spline.



Material

When checked on, the material properties are copied. Adds or removes materials, depending on whether the source entities has a material definition or not.



Text

When checked on, text style and annotative property of text and mtext is copied. Other properties, like the color, are not copied.

Viewport

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

Shadow Display

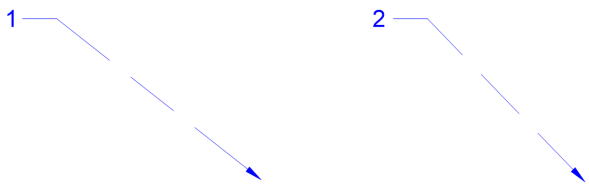
When checked on, the shadows of the source entity are copied.

Hatch

When checked on, all hatch properties and annotative scaling are copied.

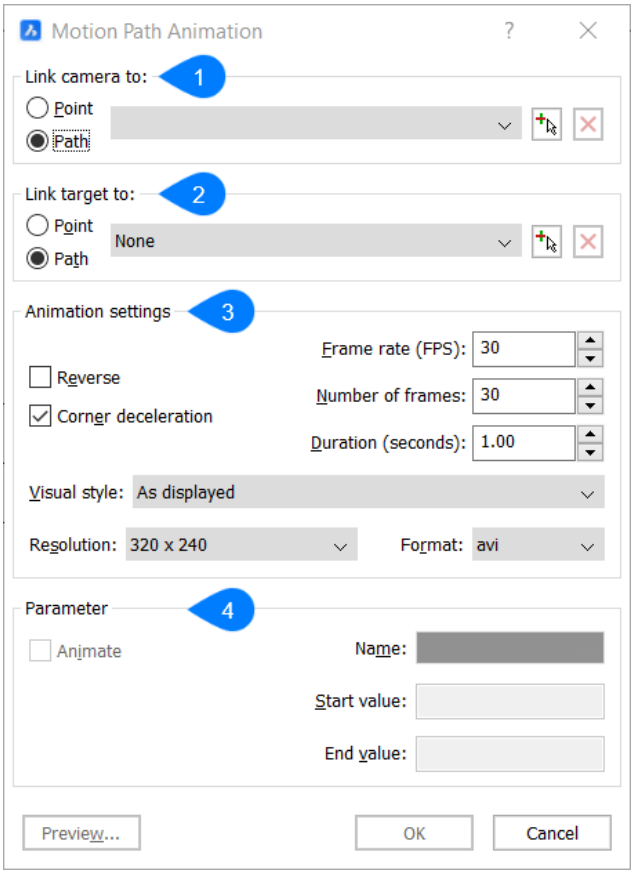
Multileader

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



4.12.97 Motion path animation

The Motion Path Animation dialog box allows you to animate a camera moving along a path or panning in a 3D model and saves it to a movie file.



1 Link camera to

2 Link target to

3 Animation settings

4 Parameter

4.12.97.1 Link camera to

Defines the location of the camera by selecting a point or a path in the model space.

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

Point
Links a camera to a static point in the drawing.

Path
Links a camera to a linear entity in the drawing.

4.12.97.2 Link target to

Defines the target point of the camera by selecting a point or path in the model space.

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

Nota: If the camera is linked to a point, the target must be linked to a path. If the camera is linked to a path, the target should be linked to either a point or a path.

Point
Links the target to a static point in the drawing.

Path
Link the target to a linear entity in the drawing.

4.12.97.3 Animation settings

Allows to control the output of the movie file.

Frame rate
Defines the number of frames per second. When the frame rate is changed, the number of frames will be adjusted according to the duration.

Number of frames
Specifies the total number of frames in the movie. When the number of frames is changed, the duration will be changed accordingly.

Duration
Specifies the duration of the movie in seconds. When the duration is changed, the number of frames will be changed accordingly.

Reverse
Reverses the sequence of the frames.

Corner deceleration
Slows down the camera at corners in the animation path.

Visual style
Defines the visual style for the movie. Click the button, then select a visual style in the list. All visual styles in the current drawing are available.

Resolution
Sets the width x height resolution of the movie. Click the button to select a resolution.
Available resolutions are: 160x120, 320x240, 640x480, 800x600, 1024x768, 1280x1024, 1600x1024, 1920x1080, 1920x1200.

Format
Specifies the file format of the movie. Available formats are: avi, mpg and wmv.

4.12.97.4 Parameter

Allows to animate the change of a parameter.

Animate
Check the box to create realistic movies with moving mechanisms.

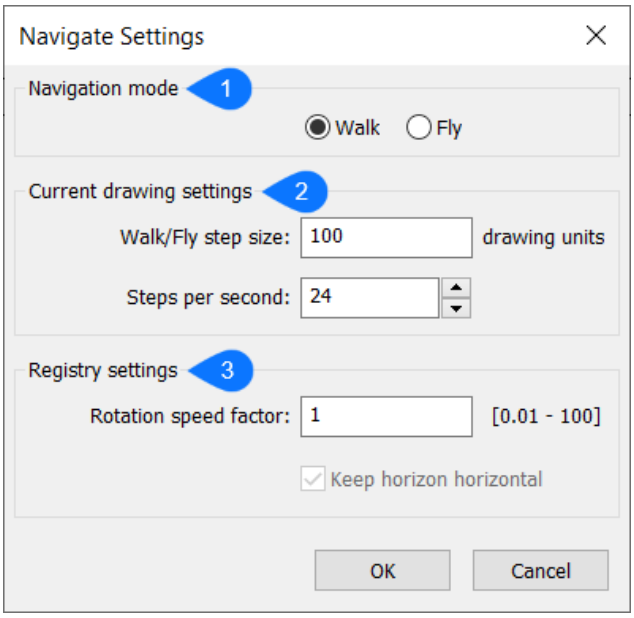
Name
Specify the name of the movie.

Start value
Specifies a start frame for the selected parameter.

End value
The parameter step is set to the Start-End range divided by the number of frames. On each frame, the parameter value should be incremented with the step.

4.12.98 Navigate Settings dialog box

The Navigate Settings Dialog box allows you to adjust the navigation settings in a drawing.



1 Navigation mode

2 Current drawing settings

3 Registry settings

4.12.98.1 Navigation mode

Toggles between Walk and Fly modes:

- Flight mode: permits movements in all three dimensions.
- Walk mode: restricts movement to the xy-plane.

4.12.98.2 Current drawing settings

Sets the settings in the current drawing, and so will be different in other drawings.

Walk/Fly step size
Specifies the distance the view travels with each press of the key. The distance is measured in drawing units.

Steps per second
Specifies how quickly the view changes when a travel key is held down.

4.12.98.3 Registry settings

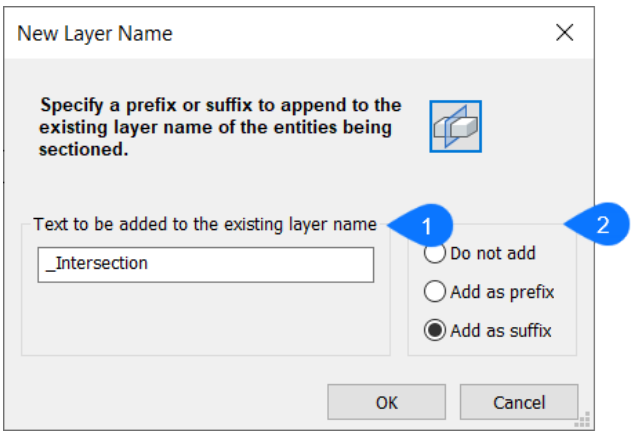
Settings which are saved with the registry, they will be the same in other drawings.

Rotation speed factor
Specifies how quickly the view rotates.

Keep horizon horizontal
When toggled on, the flight level is kept. This option is grayed-out for Walk mode.

4.12.99 New Layer Name dialog box

The New Name dialog box allows you to specify the text which is added to the name of the layer when new layers are created as a copy from the existing layer for section planes.



1 Text to be added to the existing layer name

2 Manner to add the text

4.12.99.1 Text to be added to the existing layer name

Specifies the text that will be added to the existing layer name.

4.12.99.2 Manner to add the text

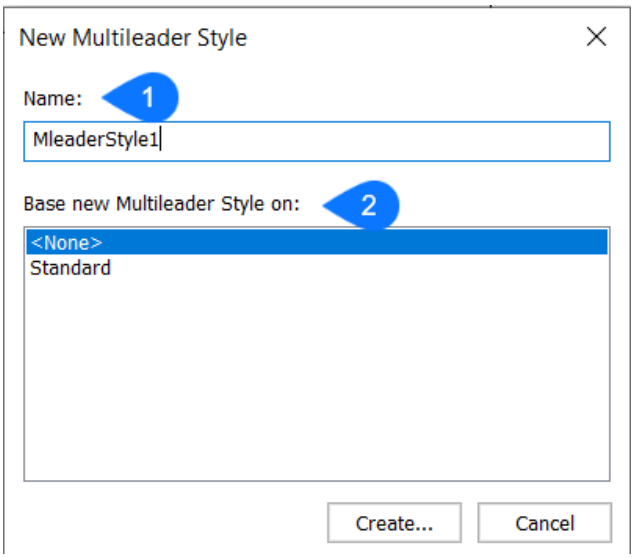
Do not add
No text will be added to name of the layer when a copy is created.

Add as prefix
The specified text will be added as a prefix to the existing name.

Add as suffix
The specified text will be added as a suffix to the existing name.

4.12.100 New Multileader Style dialog box

The New Multileader Style dialog box allows you to create a new multileader style.



1 Name

2 Base new Multileader Style on

4.12.100. Name

1

Sets the name of the new multileader style.

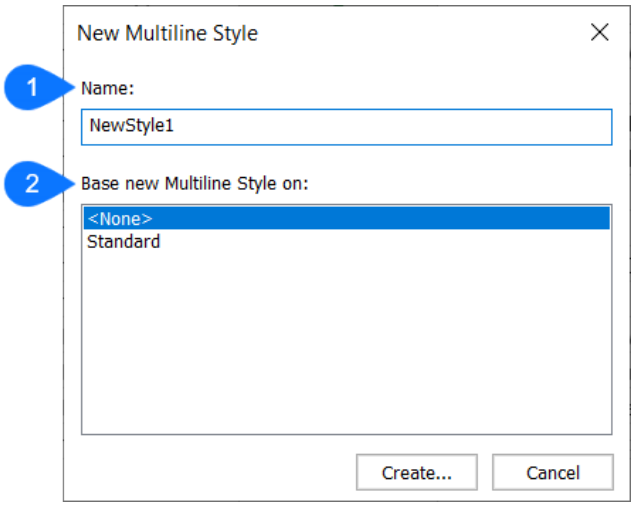
4.12.100. Base new Multileader Style on

2

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

4.12.101 New Multiline Style dialog box

The New Multiline Style dialog box allows you to create a new multiline style definition.



1 Name

2 Base new Multiline Style on

4.12.101. Name

1

Sets the name of the new multiline style.

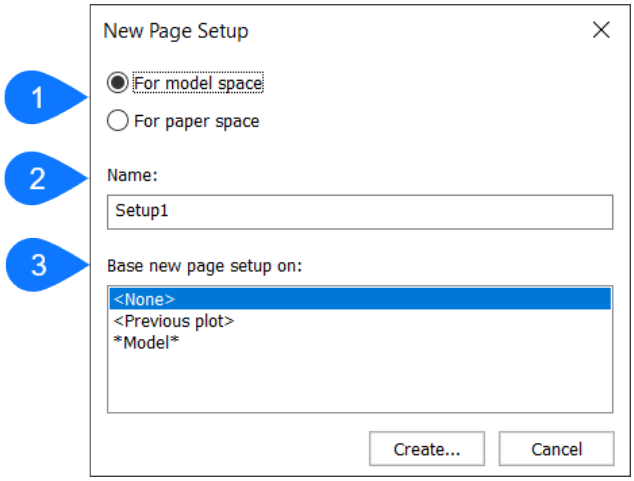
4.12.101. Base new Multiline Style on

2

Selects an existing style to create the new one as a copy of it. To start from scratch, select <None>.

4.12.102 New Page Setup dialog box

The New Page Setup dialog box allows you to create a new page setup for the paper space or model space.



1 Model space or Paper space

2 Name

3 Base new page setup on

4.12.102. Model space or Paper space

1

Sets the space to create a new page setup to the model space or paper space.

4.12.102. Name

2

Sets the name of the new page setup.

4.12.102. Base new page setup on

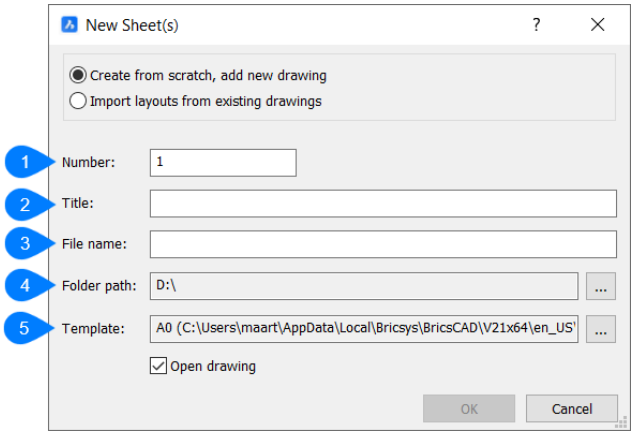
3

Selects an existing page setup to create a new one as a copy of the selected one. To start from scratch, select <None>. To use the previous plot, select <Previous plot>.

4.12.103 New Sheet(s) dialog box

The New sheets dialog box allows you to create (a) new sheet(s) within an existing sheet set.

When starting from scratch, the dialog box includes 5 areas. By selecting the Import layouts from existing drawings option, the dialog box includes 4 areas.



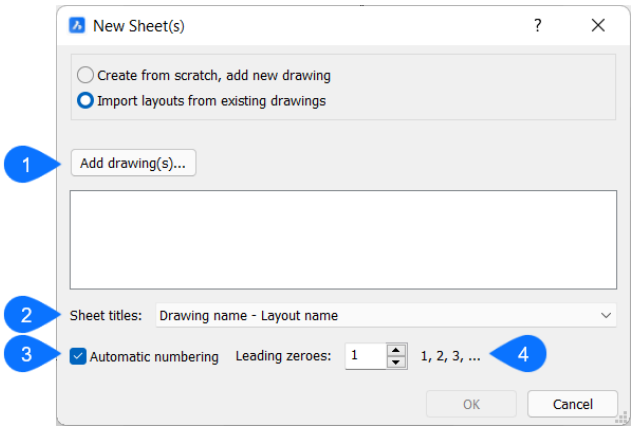
1 Number

2 Title

3 File name

4 Folder path

5 Template



1 Add drawing(s)...

2 Sheet titles

3 Automatic numbering

4 Leading zeroes

4.12.103. Create from scratch

1

Allows you to create a new sheet from scratch

- **Number**: defines the number of sheets.
- **Title**: defines the title of the new sheet(s).
- **File name**: defines the name of the file in which the new sheet(s) will be saved.
- **Folder path**: defines the location of the file.
- **Template**: choose the template of the new sheet(s).

4.12.103. Import layout

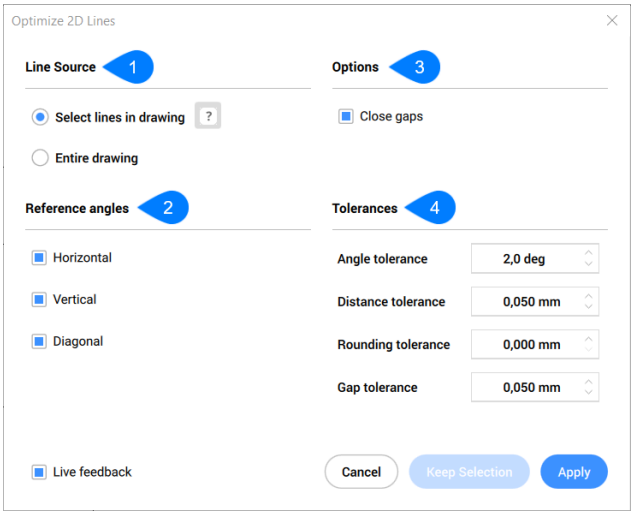
2

Allows you to create a new sheet with properties of an existing layout

- **Add drawing(s)...**: selects drawings to import a layout from trough the Select Drawing(s) dialog box.
- **Sheet titles**: specifies how the selected drawing should be listed by selecting one of the options in the drop-down list.
    - **Drawing name**: displays the drawing name of the selected drawings.
    - **Drawing name- Layout name**: displays the drawing name and the layout name of the selected drawings.
    - **Layout name**: displays the layout name of the selected drawings.
- **Automatic numbering**: numbers the layouts automatically.
- **Leading zeroes**: specifies how the Number value of new sheets is formatted by adding zeroes at the start of a sheet number.

4.12.104 Optimize 2D lines dialog box

The Optimize 2D Lines dialog box allows you to correct inaccuracies in drawings, such as small gaps between lines or near vertical, horizontal and diagonal lines. It includes 4 areas.



1 Line Source

2 Reference angles

3 Options

4 Tolerances

4.12.104. Line Source

1

Specifies the lines in the drawing to work with.

Select lines in drawing

Selects line entities in drawing trough the model space. Press enter when done.

Entire drawing

All entities in the drawing are taken into account.

4.12.104. Reference angles

2

Specifies the angles for which lines will be corrected.

Horizontal

Corrects near horizontal lines according to the angle tolerance.

Vertical

Corrects near vertical lines according to the angle tolerance.

Diagonal

Corrects near 45° lines according to the angle tolerance.

Live feedback

Highlights the lines which match the selected options.

4.12.104. Options

3

Close gaps

Closes small gaps between line entities when checked.

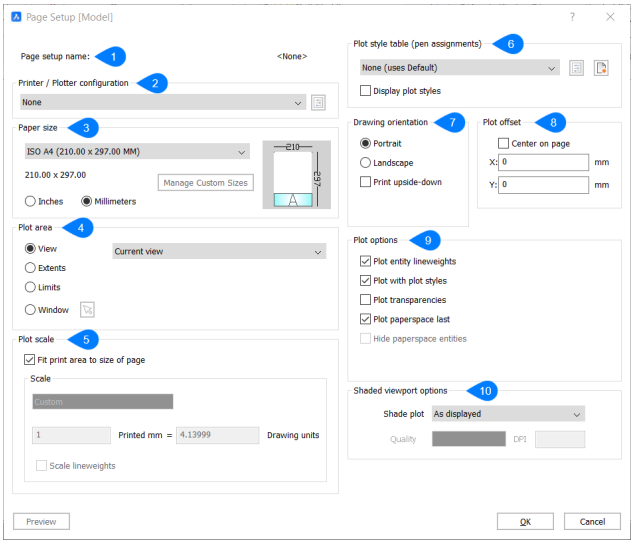
4.12.104. Tolerances

4

Specifies the tolerance value.

4.12.105 Page setup dialog box

The Page Setup dialog box allows you to edit the page setup of the current layout or model space in a dialog box.



1 Page setup name

2 Printer / Plotter configuration

3 Paper size

4 Plot area

5 Plot scale

6 Plot style table

7 Drawing orientation

8 Plot offset

9 Plot options

10 Shaded viewport options

4.12.105. Page setup name

1

Displays the name of the current setup.

4.12.105. Printer / Plotter configuration
2

Selects the printer or plotter.

4.12.105. Paper size
3

Specifies the size of paper by selecting a standard size from the drop-down list. These are sizes supported by the printer.

Nota: The size can be displayed in inches or millimeters by checking one of the two options.

4.12.105. Plot area
4

Specifies which area of the drawing to print.

View

Prints the current view or a named view. Choose a view from the drop-down list.

Extents

Prints the extents of the drawing, which ensures every visible entity is printed.

Nota: Entities on frozen layers are not taken into account when calculating the extents.

Limits

Prints the limits of the drawing, as specified by the Limits command.

Window

Prints a rectangular area of the drawing.

4.12.105. Plot scale
5

Scales the drawing to fit to the paper.

Fit print area to size of paper

Calculates the scale automatically, taking into account the print area of the drawing.

Scale

Specifies the scale factor to use for plotting.

Scale lineweights

Scales lineweights with respect to the plot scale.

4.12.105. Plot style table
6

Specifies the plot style table to use for the printed output, which assigns properties to "pens," colors, and entities.

4.12.105. Drawing orientation
7

Specifies the orientation of the drawing on rectangular paper.

Portrait
The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.

Landscape
The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

Print upside down
Prints the drawing upside-down.

4.12.105. Plot offset
8
Specifies the offset distance for the print.

Nota: To align the print in the center of the page, check the Center on page option.

Center on page
Centers the print on the page, taking into account margins.

4.12.105. Plot options
9
Plot entity lineweights
Toggles the use of lineweights.

Plot with plot styles
Toggles the use of plot styles.

Plot transparencies
Plots entities who have been assigned the transparency property, either by entity or by layer.

Plot paperspace last
Specifies the printing order.

Nota: This option is only available when printing layouts.

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

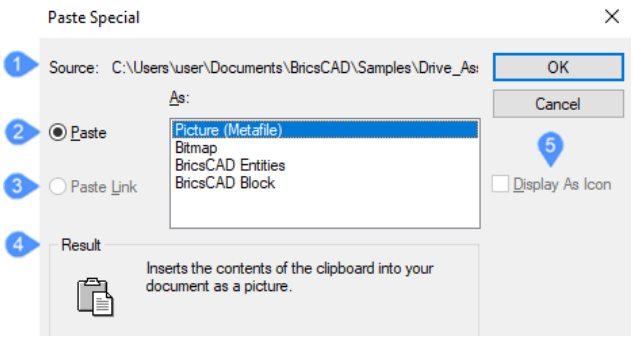
Nota: This option is disabled when printing model space.

4.12.105. Shaded viewport options
10
Override the visual style of the current view when printing model space.

Shade plot
Choose a shade style to plot the drawing with.

4.12.106 Paste special dialog box

The **Paste Special** dialog box allows you the select the type of object to paste from the Clipboard into the drawing. This command is available on Windows only. Shortcut Keystroke: CTRL+ALT+V.



1 Source

2 Paste As

3 Paste Link

4 Result

5 Display As Icon

4.12.106. Source

1

Displays the path where the drawing/entity/block can be found.

4.12.106. Paste As

2

Chooses the format in which to paste the entities. This list varies, depending on the content of the Clipboard. When the Clipboard contains data copied from a BricsCAD drawing, then the following formats are available:

- **Picture Metafile**: entities are converted to WMF (Windows Metafile) format image and then pasted as an OLE (Object Linking and Embedding) image.
- **Bitmap**: Entities are converted to a BMP format image and then pasted as an OLE image.
- **BricsCAD Entities**: pastes as individual entities or table entries including layers, linetypes, etc.
- **BricsCAD Block**: inserts the contents of the clipboard into your document as a block.

Paste As BricsCAD Entities

Pastes CAD entities into the drawing as BricsCAD entities.

Select insertion point or [Rotate/Scale/Mirror]: - *(enter an option)*

1 **Select insertion point** – *(specifies a point in the drawing by picking a point or entering X, Y coordinates)*

2 **Rotate** *-prompts you:*

Specify rotation angle – *(enter an angle by which to rotate the entities)*

3 ***Scale*** *-prompts you:*

Specify scale factor for XYZ axes – *(enter a scale factor)*

4 ***Mirror*** *– prompts you:*

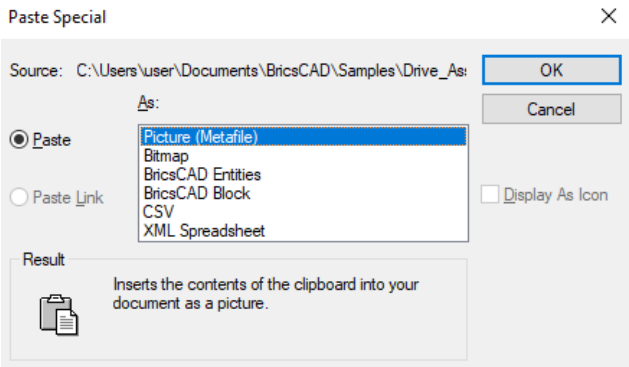
Select mirroring control point – *(Pick a second point for the mirror line)*

The insertion point is the first point for the mirror line. When you enter 0,0 for the insertion point, the entities are pasted to their original coordinates.

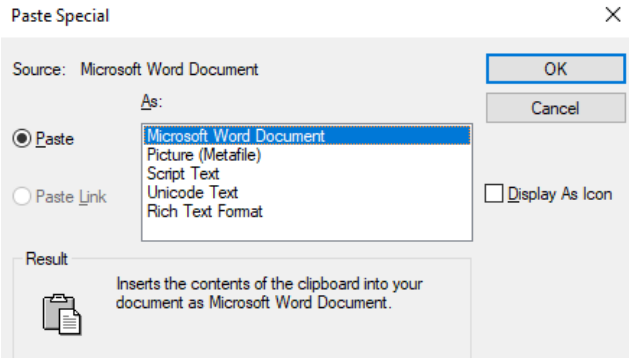
Paste As BricsCAD Block

Opens the **Insert Block** dialog box.

Here is the same dialog box, but it has some new options as the entities are copied from an Excel spreadsheet.



If you copy a text string from outside of BricsCAD, there are some new options as shown in the picture below:



4.12.106. Paste Link

3

This option is not available for CAD entities. Pastes the entities in the drawing as an OLE object and then creates a link to the originating application.

4.12.106. Result

4

Displays a short description of the selection option you want to use.

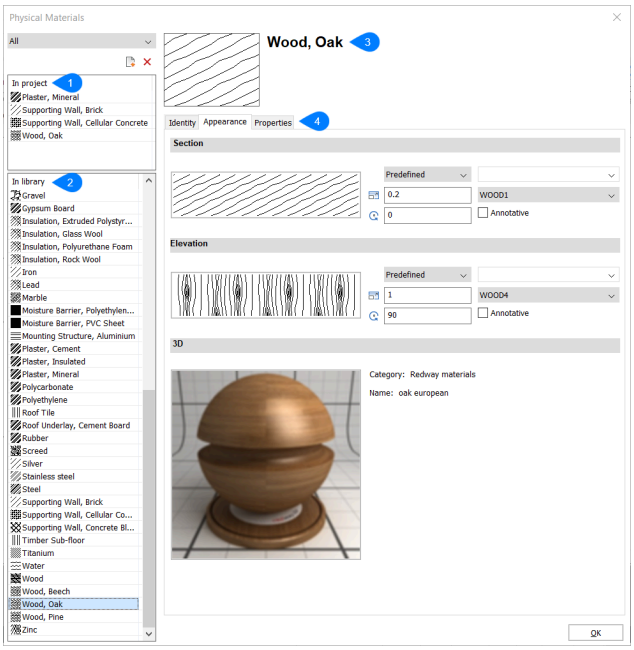
4.12.106. Display As Icon

5

This option is not available for CAD entities. Displays the pasted entities as an icon identifying their source application.

4.12.107 Physical materials dialog box

The Physical Materials dialog box opens the Physical Materials library in the current project and the referenced library.



1 In project

2 In library

3 Material name

4 Material specification tabs

4.12.107. In project

1

Displays all materials which are currently loaded in the project.

4.12.107. In library

2

Displays all materials which are loaded in the library.

4.12.107. Material name

3

Displays the name of the selected material.

4.12.107. Material specification tabs

4

Material specification tabs

Lists the broader specifications of the selected material into three tabs.

Identity

- **Name**: specifies the name of the material.
- **Class**: specifies the material class the material belongs to.
- **Description**: displays a short description of the material.

Appearance

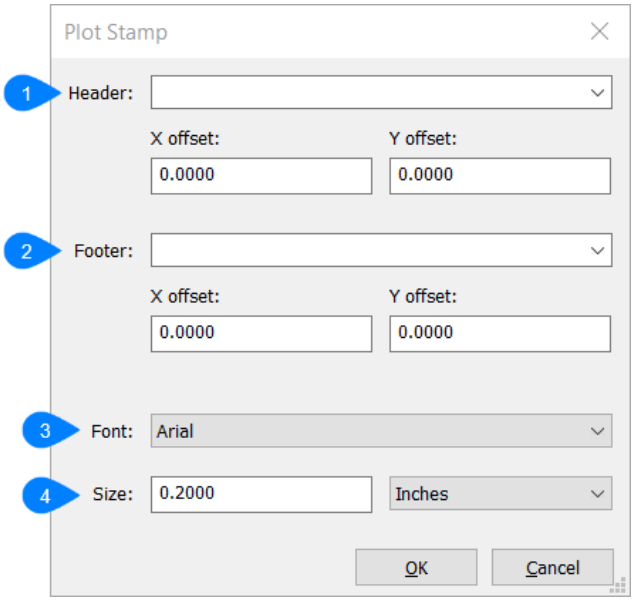
- **Section**: specifies the hatch pattern used to display the material in a section view.
- **Elevation**: specifies the hatch pattern used to display the material in an elevation view.
- **3D**: shows the three dimensional texture of the material that will be used in all 3D views.

Properties

Specifies additional properties of the material such as the density.

4.12.108 Plot stamp dialog box

The Plot Stamp dialog box allows you to add information about drawings and prints to the edge of the plotted drawing.



1 Header

2 Footer

3 Font

4 Size

4.12.108. Header

1

Appears at the top of the plot. A meta-phrase for the header can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

4.12.108. Footer

2

Appears at the bottom of the plot. A meta-phrase for the footer can be chosen by clicking the drop-down list or by specifying the X and Y offsets.

4.12.108. Font

3

Specifies the font used for the plotstamp.

4.12.108. Size

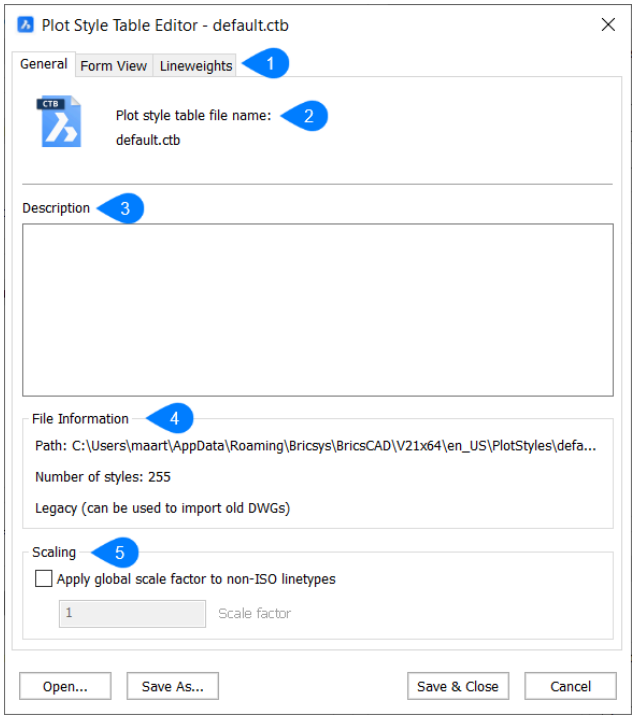
4

Specifies the height of the text.

- Inches: specifies the height of the text in inches.
- Millimeters: specifies the height of the text in millimeters.

4.12.109 Plot style table editor dialog box

The Plot Style Table Editor displays the plot styles contained in the specified plot style table.



1 Editor tabs

2 Plot style table file name

3 Description

4 File Information

5 Scaling

4.12.109. Editor tabs

1

The Plot Style Table Editor includes the following tabs.

General tab

The General tab lists general information about the plot style table.

Form View tab

The Form View tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

Lineweights tab

The Lineweights tab lists the available lineweights.

4.12.109. Plot style table file name

2

Displays the name of the plot style table file you are editing.

4.12.109. Description

3

Displays a description you want to include about the plot style table.

4.12.109. File Information

4

Displays information about path, number of plot styles and the version number of the Plot Style Table Editor.

4.12.109. Scaling

5

Apply global scale factor to non-ISO linetypes

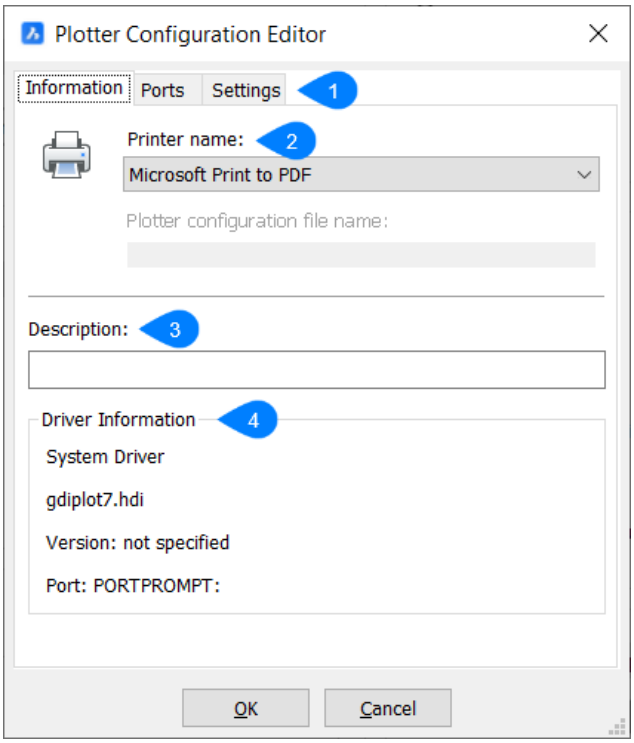
Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.

Scale factor

Specifies the factor by which to scale non-ISO linetypes and fill patterns.

4.12.110 Plotter Configuration Editor dialog box

The Plotter Configuration Editor dialog box allows you to create and edit customized parameter PC3 files for printers and other output devices.



1 Editor tabs

2 Printer name

3 Description

4 Driver Information

4.12.110. Editor tabs

1

Information

Gives basic information about the plotter.

Ports

Gives options on how the file will be printed.

Settings

Provides additional settings to alter. The settings available depend on the capabilities of the printer selected.

4.12.110. Printer name

2

Lists the names of printers supported by the operating system.

4.12.110. Description

3

Describes the plotter configuration. It is displayed by tooltips.

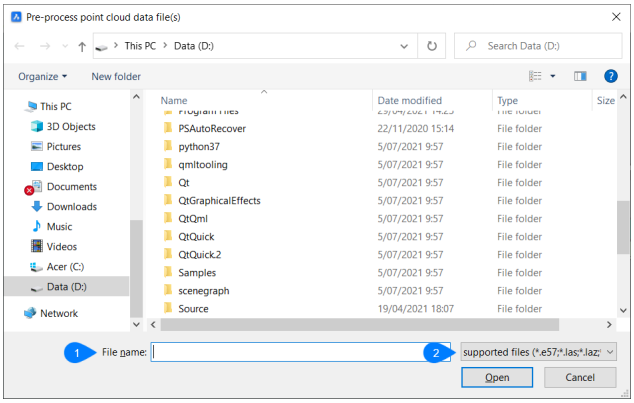
4.12.110. Driver Information

4

Displays basic information about the driver.

4.12.111 Pre-process point cloud data file(s) dialog box

The Pre-process Point Cloud Data File's dialog box allows you to attach point cloud files (that were previously converted to BricsCAD's BPT format) to the current drawing.



1 File Name

2 Supported files

4.12.111. File Name

1

Specifies the name of the point cloud data file you want to attach.

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

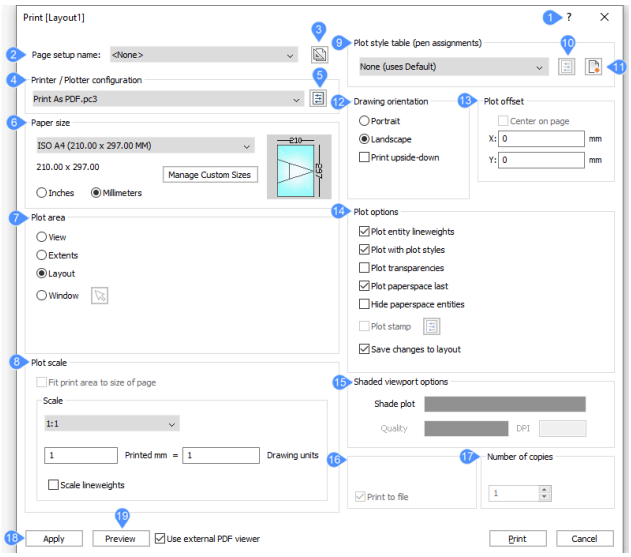
4.12.111. Supported files

2

Specifies the file types you will be able to choose from.

4.12.112 Print dialog box

The **Print** dialog box allows you to print and preview drawings to plotters and files.



1 Command reference

2 Page setup name

3 Create page setup

4 Printer/Plotter configuration

5 Edit plotter configuration

6 Paper size

7 Plot area

8 Plot scale

9 Plot style table

10 Edit Plot Style

11 Create New Plot Style

12 Drawing orientation

13 Plot offset

14 Plot options

15 Shaded viewport options

16 Print to file

17 Number of copies

18 Apply

19 Preview

4.12.112. Command reference

1

Opens the Bricsys Help article about the PRINT command.

4.12.112. Page setup name
2

You can select plot settings used previously from the drop-down list:

- **<None>**: Uses options saved in the current layout or model space.
- **<Previous Plot>**: Uses options that were saved from the last time when this dialog box was used.
- Names of other page setups stored in the drawing.

4.12.112. Create page setup
3

Opens the **Create Page Setup** dialog box that allows you to create new page setups. See the PAGESETUP command.

4.12.112. Printer/Plotter configuration
4

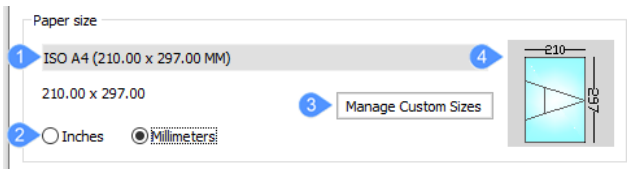
Allows you to select from the drop-down list the printer or plotter.

The program works with any output device installed on the system, including networked printers, print to file, faxes, and PostScript devices -- as well as predefined printer parameters stored in PC3 files.

4.12.112. Edit plotter configuration
5

Opens the **Plotter Configuration Editor** dialog box for customizing the printer's parameters and creating PC3files. See the PLOTTERMANAGER command.

4.12.112. Paper size
6



1 Standard Paper Sizes list

2 Units

3 Manage Custom Sizes

4 Preview

Standard Paper Sizes list

Specifies the size of paper. You can select standard sizes listed by the drop-down list; these are sizes supported by the printer.

While printers may appear to support many different sizes, you should select only the size of paper that is in the printer.

Units

- **Inches**: Use imperial units for measurements involving printing.
- **Millimeters**: Use metric units for measurements involving printing.

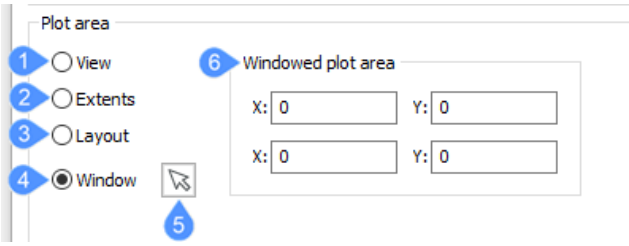
Manage Custom Sizes

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

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

Preview
This section indicates the size, position and orientation of the current plot area on the selected paper size.

4.12.112. Plot area
7



1 View

2 Extents

3 Layout

4 Window

5 Select area to be printed

6 Windowed plot area

View
Prints the current view or a named view. Choose a view from the drop-down list. Use the VIEW command to create named views.

Extents
Prints the extents of the drawing, which ensures every visible entity is printed. Entities on frozen layers are not considered when calculating the extents.

Layout
Prints the current layout.

Window
Prints a rectangular area of the drawing.

- You define the rectangular area by entering the x, y coordinates or by clicking the **Select Area to be Printed** button.
- The x, y coordinates resulting from the selection or from entering in the Command line are added to the **Windows Plot Area** boxes. You can edit the values in a convenient way.

4.12.112. Plot scale
8

Scales the drawing to fit it to the paper:

- **Fit print area to the size of paper**: Scale is calculated automatically by the program, considering the print area of the drawing. When this option is turned on, you cannot specify the scale factor:
  - **Printable area** is the size of the paper, minus the margins.
  - **Margins** are the strips along the four edges that the printer uses for handling the paper.
- **Scale**: Allows you to specify the scale factor to use for plotting; choose a scale factor from the drop-down list or choose the **Custom** option that will allow you to enter your own scale factors in **Printed inches/mm** and **Drawing units** fields. The scale factor shown can be edited with the SCALELISTEDIT command.
  - For scale factors like 1:5, the drawing is printed smaller.
  - For scale factors like 5:1, the drawing is printed larger.
- **Scale lineweights**: When on, lineweights are scaled with respect to the plot scale.

4.12.112. Plot style table
9

Specifies the plot style table to use for the printed output, which assigns properties to pens, colors, and entities.

- When the drawing does not use plot styles, then only CTB (color-based table) files are listed.
- When the drawing uses plot styles, then only STB (style-based table) files are listed.

When you switch from **None** to a named plot style, BricsCAD prompts you:

Assign plot style table to all layouts?

- **Yes**: Assigns the CTB or STB plot style file to all layouts.
- **No**: Assigns the plot style file to the current layout only.

4.12.112. Edit Plot Style
10

Opens the **Plot Style Table Editor** dialog box. See the PLOTSTYLE command. This button is available only when the drawing uses STB plot styles.

4.12.112. Create New Plot Style
11

Opens the **Add Plot Style Table** dialog box to create new plot styles. See the STYLESMANAGER command.

- When the drawing does not use named plot styles, then the wizard creates new color-dependent tables (CTB files).
- If the drawing uses named plot styles, then the wizard creates new named plot style tables (STB files).

4.12.112. Drawing orientation
12

Specifies the orientation of the drawing on rectangular paper:

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.
- **Print upside down**: Prints the drawing upside-down. This is useful when paper with a drawing border is loaded backwards in the printer.

4.12.112. Plot offset
13

Specifies the offset distance for the print.

- **Center on Page**: Centers the print on the page, considering margins. Together with Fit Print Area to Size of Page, this option is excellent for prints where scale is unimportant.
- **X** and **Y**: Enter positive or negative distances to move the drawing in the x and/or y distances:
  - **Positive values**: Move the drawing up and to the right.
  - **Negative values**: Move the drawing down and to the left.

The lower left corner of the drawing is moved by the specified distance. This is useful when the paper has a title block area that might interfere with the drawing.

4.12.112. Plot options
14

- **Plot entity lineweights**: Toggles the use of lineweights:
  - On: Mimics the lineweights assigned to the drawing in the print.
  - Off: Ignores lineweights.
- **Plot with Plot Styles**: Toggles the use of plot styles:
  - On: Uses plot styles to determine the look of the printed drawing and overrides lineweight settings.
  - Off: Does not use plot styles.
- **Plot Transparencies**: Plots entities who have been assigned the Transparency property, either by-entity or by layer.
- **Plot Paperspace Last**: Specifies the printing order:
  - On: Prints model space entities, followed by paper space entities.
  - Off: Prints paper space entities first, followed by entities in model space.
- **Hide Paperspace Entities**: When on, removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
- **Plot Stamp**: Toggles use of the plot stamp; click the Edit Plot Stamp button to change plot stamp data. See the PLOTSTAMP command. This option is unavailable when the Printer/Plotter configuration is set to *Print As PDF.pc3*.
  - On: Applies plot stamp data to the print.

- Off: Does not apply plot stamps.

- **Save Changes to Layout**: Determines if the options you changed in this dialog box are saved; the next time you use this dialog box, you can select "Layout" from the Use Plot Settings From drop-down list.
  - On: Saves changes made to this dialog box with the layout.
  - Off: Does not save changes.

4.12.112. Shaded viewport options
15

Override the visual style of the current view when printing model space:

Choose from the drop-down list a shade style to plot the drawing with.

This option is disabled when printing a paper space layout. The **Shade Plot** mode of a paper space viewport is defined in the **Shade Plot** property of the viewport. The quality of the **Rendered** option is defined through the current render preset; see the RENDERPRESETS command.

4.12.112. Print to file
16

When checked, redirects the print output to a *.plt file, which can be processed by certain types of software.

4.12.112. Number of copies
17

Specifies the number of copies to print. Enter a number or click the buttons to change the value.

4.12.112. Apply
18

Applies the changes you made to this dialog box. The changes are remembered the next time you use this command. This is like using the PAGESETUP command.

4.12.112. Preview
19

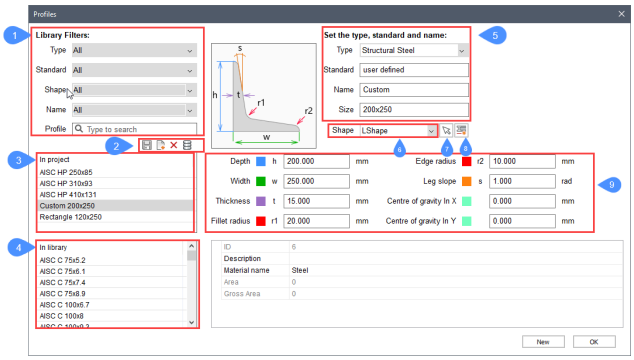
Displays a preview of the print. In the preview window, click the **Print Settings** button to return to this dialog box. See the PREVIEW command.

Nota: On Mac or Linux, printing will always print to PDF. So, a PDF document will be generated that then still needs to be sent to the printer for physical printing.

On MAC and Linux there is a relevant PAGESETUP command.

4.12.113 Profiles dialog box

Opens the profiles library in the current project and the referenced library.



1 Library Filters

2 Tools

3 In Project

4 In Library

5 Custom fields

6 Preselect shape

7 Pick profile in model

8 Set profile offset

9 Properties of the profile

4.12.113. Library Filters

1

Allows to filter the profiles list by type, standard, shape or name.

- **Type**: The **Type** options are: **All**, **Generic**, **HVAC**, **Piping** or **Structural Steel**.
- **Standard**: The Localized standard options are: **All**, **AISC** (American Institute of Steel Construction), **BS** (British Steel), **EURO** (European standard steel profiles) and **User Defined**.
- **Shape**: The **Shape** options are: **All**, **Circle Hollow**, **I Shape**, **L Shape**, **Rectangle**, **Rectangle Hollow**, **T Shape** and **U Shape**. Depending on the selected standard, some shape options are not available.
- **Name**: Filters the profiles by a selected character string.
- **Profile**: Select a character string. Only profiles that contain the selected string are listed.

4.12.113. Tools

2

- **Save Profile**: Save the currently edited profile.
- **New Profile**: Create a new profile.
- **Delete Profile**: Delete the selected profile.
- **Project and Library Information**: Opens the BIM **Project Info** dialog box.

4.12.113. In Project

3

Lists the available profiles used in the current project with respect with above filters.

4.12.113. In Library

4

Lists the available profiles with respect with above filters.

4.12.113. Custom fields

5

Lists the characteristics of the chosen profile or in case of creating a new profile, sets the type, standard and name for it.

4.12.113. Preselect shape

6

Select a shape from the drop-down list.

4.12.113. Pick profile in model

7

Select a closed 2D entity or the boundary entities of an enclosed area.

4.12.113. Set profile offset

8

Sets the profile offset.

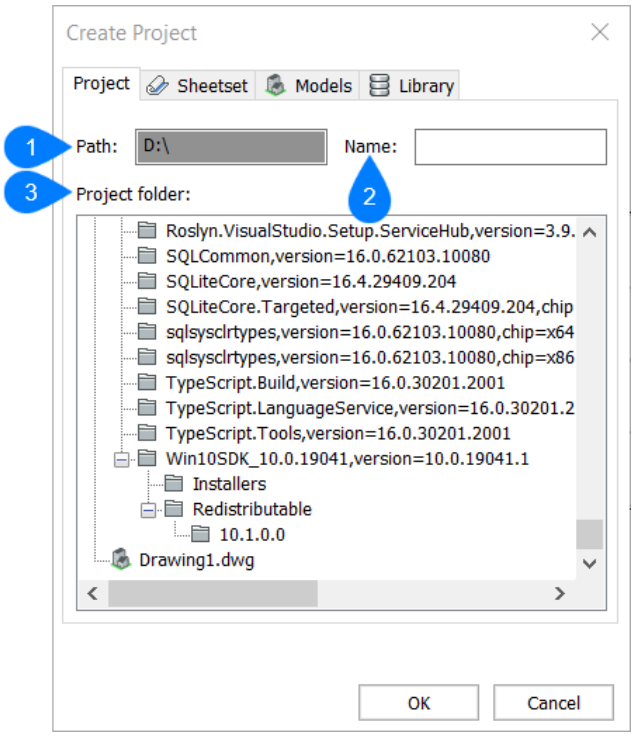
4.12.113. Properties of the profile

9

Shows the properties of the selected profile or sets the properties of a new profile.

4.12.114 Project setup dialog box

The Project Setup dialog box allows you to create a new project and choose the file location.



1 Path

2 Name

3 Project folder

4.12.114. Path

1

Specifies the location of the project by selecting the parental folder.

4.12.114. Name

2

Creates a new project file by entering a new name.

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

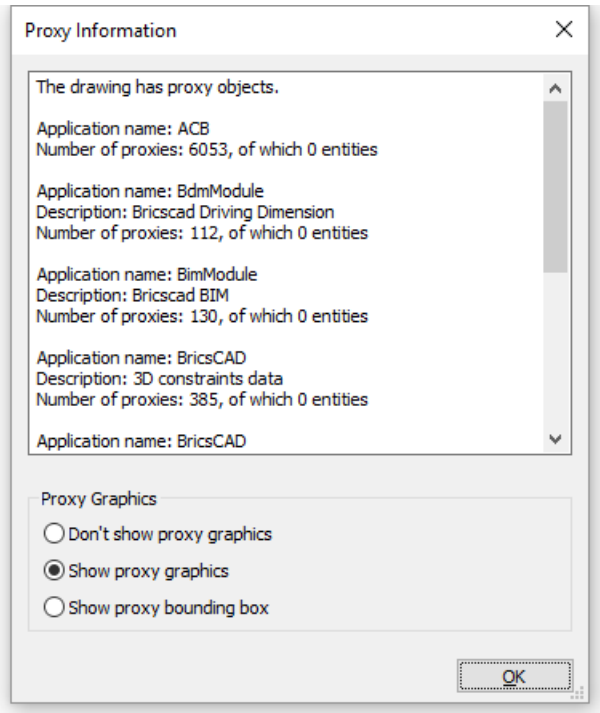
4.12.114. Project folder

3

Specifies the folder in which the project will be saved by selecting the destination folder.

4.12.115 Proxy information dialog box

The **Proxy Information** dialog box displays information about proxy entities and toggles their display in the current drawing.



4.12.115. Proxy Graphics

1

Toggles the display of proxy graphics in the drawing:

- **Don't show proxy graphics**: turns off the display of proxy graphics.
- **Show proxy graphics**: turns on the display of proxy graphics
- **Show proxy bounding box**: displays a rectangle or cube instead of the proxy.

4.12.116 Publish dialog box

The Publish dialog box allows you to send one or more drawings, layouts and sheets to the printer or export them in PDF format. This is useful for batch printing of books of drawings.

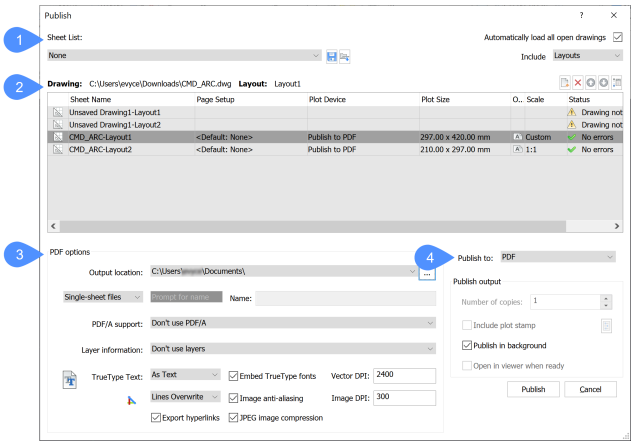
To print a batch of drawings when BricsCAD starts, write in the Terminal the following command

bricscad.exe /pl <"the path of your *.dwg file"> <"the path of your *.dsd file">

For example:

bricscad.exe /pl "C:\Users\user\Desktop\doc.dwg" "C:\Users\user\Desktop\doc.dsd"

Also, in the Page Setups section of the Drawing Explorer dialog box set the printer you use in the Device name.



1 Sheet List

2 Drawing

3 PDF options

4 Publish to

4.12.116. Sheet List

1

Displays a list of sheets. If you have a *.dsd file (drawing sheets for publishing), you can select it and the Load Sheet List dialog box will appear.

Save Sheet List

Saves the current sheet list to a *.dsd (Drawing Set Description) file. Opens the Save Sheet List dialog box.

Load Sheet List

Loads a saved sheet list. Opens the Load Sheet List dialog box. If the current list is not saved yet, you are prompted to save it.

If you are trying to load a file with an existing name, after you press the Open button in the Load Sheet List dialog box you are prompted to replace or append the sheets to the list.

Automatically load all open drawings

Determines how open drawings are handled:

- **On**(checked): adds all model and/or paper space layouts to the list of sheets to be plotted
- **Off**(unchecked): adds the model and/or paper space layouts of the current drawing only

This option also sets the value of the PUBLISHALLSHEETS system variable.

Include

Determines which sheets must be included:

- **Model**: includes only model tabs of open drawings.
- **Layouts**: includes only layout tabs of open drawings.
- **Model & Layouts**: includes both model and layout tabs of open drawings.

4.12.116. Drawing

2

Layout

Page Setup

Specifies the Page Setup name of each sheet. Click the Page Setup name, then click the down arrow to select a different page setup or choose **Import…** to import page setups of another drawing. This will open the Import Pagesetups dialog box.

Plot Device

Displays the name of the plot device, as defined in the Page Setup dialog box of the sheet.

4.12.116. PDF options

3

Output location

Displays the path where the PDF will be saved. Opens the Choose a folder dialog box to change the path.

File

Select an option:

- **Single – sheet files**: each layout is published to a separate *.pdf. The file names are: <drawing>_<layout>.pdf
- **One Multi – sheet file**: layouts are published to a single file. You can choose to specify a name or to be prompted for a name when the publishing procedure starts.

Name options

Select an option:

- **Prompt name**: after you press the Publish button you are prompted to insert the name you want for your *.pdf in the Specify PDF file name dialog box.
- **Specify name**: the field for Name becomes available. Insert here the name you want for your *.pdf.
- **Generate name**: the *.pdf will have the same name as the drawing.

Name

Allows you to type the name you want for your *.pdf. This field is available only for the **Specify name** option.

PDF/A Support

Allows you to select from the drop-down list one of the options.

Layer information

- **Don't use layers**: no layers are exported.
- **Use all layers with visible entities**: only layers which are current ON and Thawed are exported.
- **Use layers with entities, including OFF and FROZEN layers**: all layers containing entities are exported.

TrueType Text

Select an option:

- **As text**: ensures that the text in the PDF looks exactly as in the drawing.
- **As geometry**: converting to geometry increases the file size and pixelation might occur when viewed at a high zoom level.

Embed TrueType fonts
If the font is not embedded, a substitute font might be used by the PDF viewer.

Nota: Licensed fonts cannot be embedded.

Vector DPI
Controls the resolution of vector graphics and gradients.

Merge control
Determine whether overlapping lines are merged or lie on top of each other.
Select an option:

- **Lines Overwrite**: the lines on top overwrite the lines below.
- **Lines Merge**: the colors of crossing lines are blended into a new color.

Image anti-aliasing
If the resolution of an image is too low to match the Image DPI setting, then the raster image is anti-aliased (generate pixels artificially) to make the raster image look more smoothly. Without that setting an image with a too low resolution is embedded "as-is".

Image DPI
Defines the resolution of the resulting image in the PDF for raster images. This setting allows you to lower the resolution of the original raster image to avoid including a gigabyte raster image in the PDF. For printing the resolution should match the output resolution of the printer. For on-screen viewing high resolutions (2400 dpi +) are required.

Export hyperlinks
If checked, it allows you to export hyperlinks in the *.pdf.

JPEG image compression
This setting applies to all included raster images in the PDF, both raster images and rendered views. It applies a JPEG compression to these images to reduce the PDF file size. When switched off, the images are embedded as raw bitmaps. The JPEG compression is lossy compared to the raw bitmap introducing small imperfections.

4.12.116. Publish to

4

Nota: When **Publish to** is set to **Plotter named in page setup**, the PDF options from this dialog box disappear.

Number of copies
Sets the number of copies for each sheet to be printed.

Include plot stamp
If this option is checked, it adds a plot stamp.

Plot Stamp
Opens the Plot Stamp dialog box.

Publish in background
Determines if the sheet is published in the background:

- **Yes**: sheets are published in the background, which takes longer to produce the prints, but allows you to continue to work in the program.

- **No**: sheets are published in the foreground, which is faster, but prevents you from working with the program until the print job is completed.

The BACKGROUNDPLOT system variable controls whether documents are published in the foreground or background.

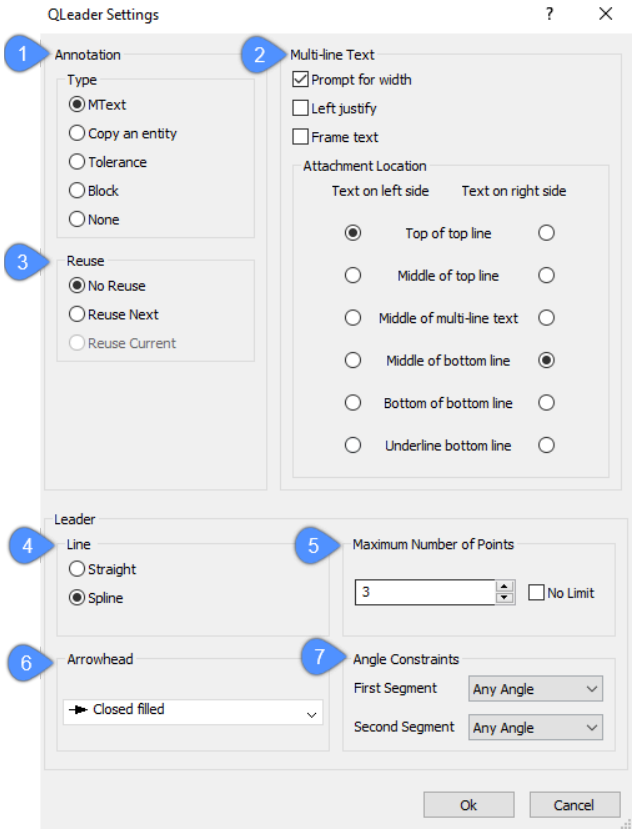
Open in viewer when ready

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

Nota: This option is not available when **Publish to** is set to **Plotter named in page setup**, only when **Publish to** is set to **PDF**.

4.12.117 QLeader Settings dialog box

The **Qleader Settings** dialog box allows you to specify the properties of leaders.



1 Annotation

2 Multi-line Text

3 Reuse

4 Leader Line

5 Maximum Number of Points

6 Arrowhead

7 Angle Constraints

4.12.117. Annotation

1

Specifies the type of annotation:

- **Mtext**: Text is placed as the leader line's annotation; a leader can have one or more lines of text.

  Nota: Although the mtext editor does not appear when entering text (as with the MLeader command), the mtext editor does appear when editing the annotation (edit the text by double-clicking it).
- **Copy of an entity**: Uses any other entity (than the leader itself) in the drawing as the annotation.
- **Tolerance**: Uses tolerance notation for the annotation; displays the **Geometric Tolerance** dialog box. See the TOLERANCE command.
- **Block**: Uses a block from the drawing or an external block as the annotation. The prompt displayed is identical to that of the -INSERT command.
- **None**: No annotation is used.

4.12.117. Multi-line Text

2

Nota: This option is only available when the annotation type is MText.

Specifies some multi-line text options:

- **Prompt for width**: Toggles the use of the 'Specify text width' prompt:
  - On: Displays the 'Specify text width' prompt and so wraps the text to the specified width.
  - Off: Disables the prompt.
- **Left justify**: Determines the justification of the text:
  - On: Forces the text to be left-justified always.
  - Off: The text is left or right-justified, depending on the orientation of the landing line.
- **Frame text**: Toggles the use of a frame:
  - On: Places a rectangle around the text.
  - Off: No frame.
- **Attachment location**: Determines the relative location of where the mtext is attached to the landing line.

4.12.117. Reuse

3

Specifies how to reuse the annotation:

- **No reuse**: Does not reuse the previous annotation text; you will be prompted to enter new text each time you use this command.
- **Reuse next**: You will be prompted to enter new text, which is then reused for subsequent leader(s).
- **Reuse current**: The current text is reused for each new leader, and you will not be prompted to enter new text.

4.12.117. Leader Line
4

Specifies the style of leader line:

- **Straight**: Draws the leader lines as straight segments.
- **Spline**: Draws the leader line as a smooth continuous spline.

4.12.117. Maximum Number of Points
5

Specifies the maximum number of vertices for the leader, before it stops drawing them automatically:

- Enter a number; the minimum number is 2.
- **No limit**: Keeps drawing leader segments until you press Enter at the 'To point' prompt.

4.12.117. Arrowhead
6

Specifies the type of arrow at the start of the leader; choose from the drop-down list:

- Choose an arrowhead type.
- **None**: No arrow is drawn.
- **Select arrow**: Choose a block from those in the current drawing; displays the **Select Custom Arrow Block** dialog box.

To create a custom arrow, draw it to your liking, and then use the BLOCK command to turn it into a block.

4.12.117. Angle Constraints
7

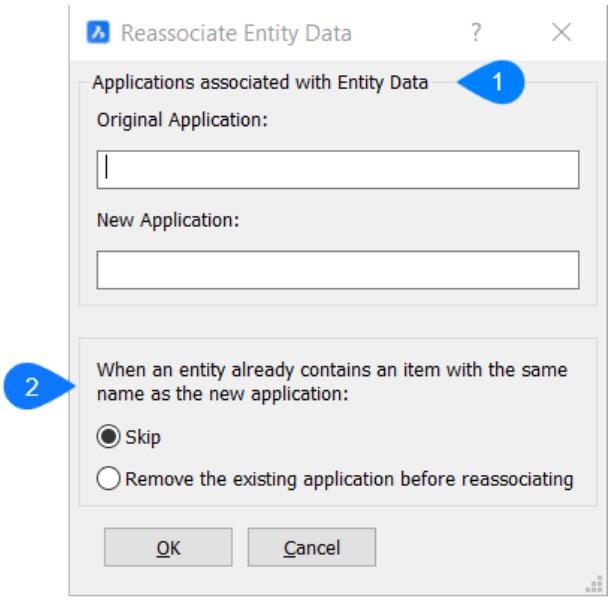
Set an angle to constrain the first and second leader segments. The angles are increments; for instance, choose 15 allows you to draw the leader at 0, 15, 30, 45, and so on degrees.

Click the drop-down list, then choose an option:

- **Any Angle**: No constraint; hold down the Shift key to draw the leader at 90 degrees.
- **Horizontal**: Forces the leader line to be drawn at 90 degrees only.
- **90°**: Draws leader lines at 90-degree increments.
- **45°**: Draws leader lines at 45-degree increments.
- **30°**: Draws leader lines at 30-degree increments.
- **15°**: Draws leader lines at 15-degree increments.

4.12.118 Reassociate entity data dialog box

The Reassociate Entity Data dialog box allows you to associate extended entity data with a specific application.



1 Applications associated with Entity Data

2 Options

4.12.118. Applications associated with Entity Data

1

Original application

Specifies the name of the original application.

New application

Specifies the name of the new application. Entities will be reassociated with this application name.

4.12.118. Options

2

Skip

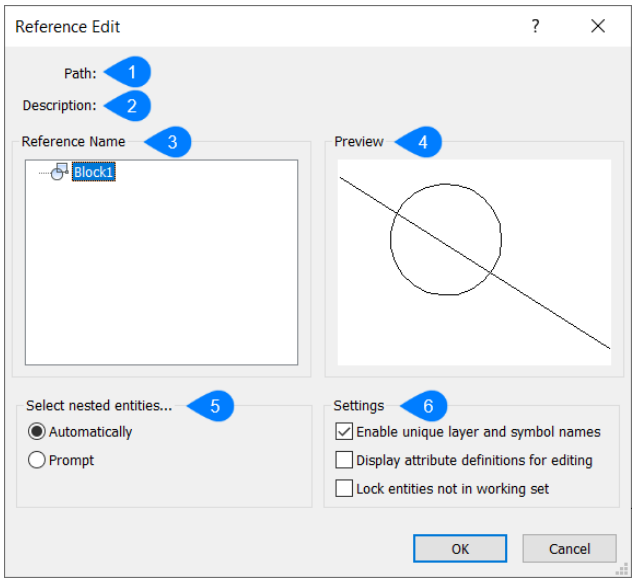
Skips entities that already contains an item with the same name as the new application.

Remove the existing application before reassociating

Removes the existing application before reassociating.

4.12.119 Reference Edit dialog box

The Reference Edit Dialog Box allows you to edit block references and externally-referenced drawings.



1 Path

2 Description

3 Reference Name

4 Preview

5 Select nested entities

6 Settings

4.12.119. Path

1

Displays the name of the path.

4.12.119. Description

2

Displays the description of the path.

4.12.119. Reference Name

3

Lists the name of the block and nested blocks, if any.

4.12.119. Preview

4

Displays a preview of the current reference.

4.12.119. Select nested entities

5

Determines how nested blocks/xrefs are handled.

Automatically

Includes nested entities automatically.

Prompt
Includes nested entities by selecting them after the dialog box is closed by pressing OK.

4.12.119. Settings

6
Determines how entities are treated during editing.

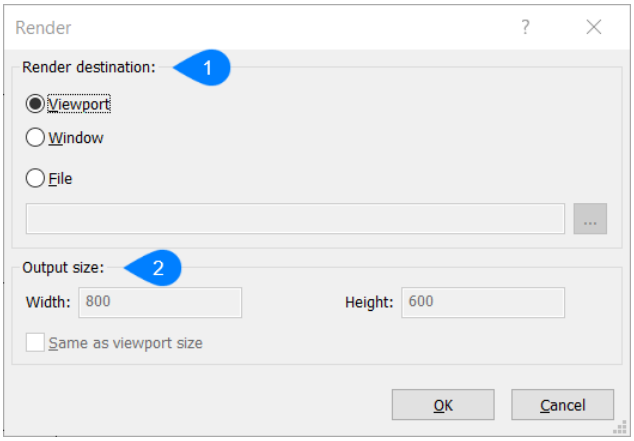
Enable unique layer and symbol names
Toggles the use of 0 in layer and other names.

Display attribute definitions for editing
Toggles the inclusion of attributes.

Lock objects not in working set
Toggles locking of entities not being edited.

4.12.120 Render dialog box

The Render dialog box allows you to generate a photo-realistic rendering using materials and lights.



1 Render destination

2 Output size

4.12.120. Render destination

1
Specifies where the render will appear.

Viewport
Generates the render in the current viewport.

Window
Generates the render in a new window.

File
Saves the render to a file. Selecting the three dots opens the render output file dialog box.

4.12.120. Output size

2

Specifies the size of the rendering, in pixels. Larger sizes provide more detail but take longer to render.

Width

Specifies the width of the rendering, in pixels.

Height

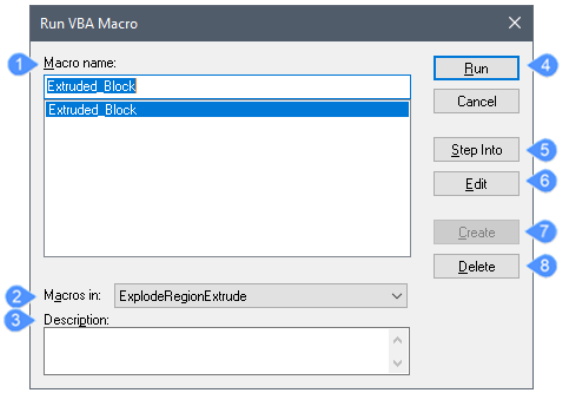
Specifies the height of the rendering, in pixels.

Same as viewport size

Matches the output size or the size of the output window to the current viewport size.

4.12.121 Run BricsCAD VBA macro dialog box

The **Run VBA Macro** dialog box runs, creates, edits, and deletes VBA macros.



1 Macro name

2 Macros in

3 Description

4 Run

5 Step Into

6 Edit

7 Create

8 Delete

4.12.121. Macro name

1

Lists the names of macros found in the project file. Choose one to run.

4.12.121. Macros in

2

Lists the names of projects loaded into the current drawing. Choose one from the drop-down list.

4.12.121. Description
3

Describes the macro. You can edit this text and save it pressing the **Close** button.

4.12.121. Run
4

Runs the selected macro.

4.12.121. Step Into
5

Opens the integrated development environment window. The macro is in debug mode, which means that it runs one line of code at a time.

See the VBAIDE command.

4.12.121. Edit
6

Opens the macro in the integrated development environment window. The macro is in editing mode, which allows you to change the code.

See the VBAIDE command.

4.12.121. Create
7

Creates new macros; follow these steps:

1 In the **Macro Name** field, enter a name for the new macro.

2 Click **Create**.

3 Notice the integrated development environment window. Enter the VBA code.

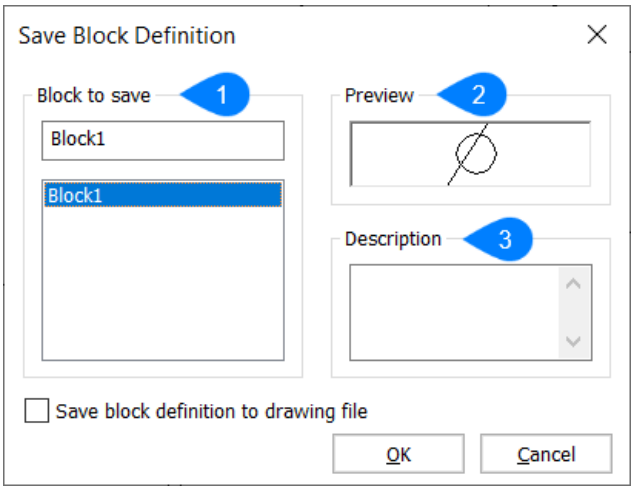
See the VBAIDE command.

4.12.121. Delete
8

Deletes the macro. A dialog asks to confirm the action.

4.12.122 Save block definition dialog box

The Save Block Definition dialog box allows you to copy the current block definition to a new name.



1 Block to save

2 Preview

3 Description

4.12.122. Block to save

1

Specifies a new name for a selected block to make a copy of.

4.12.122. Preview

2

Displays a preview of the selected block.

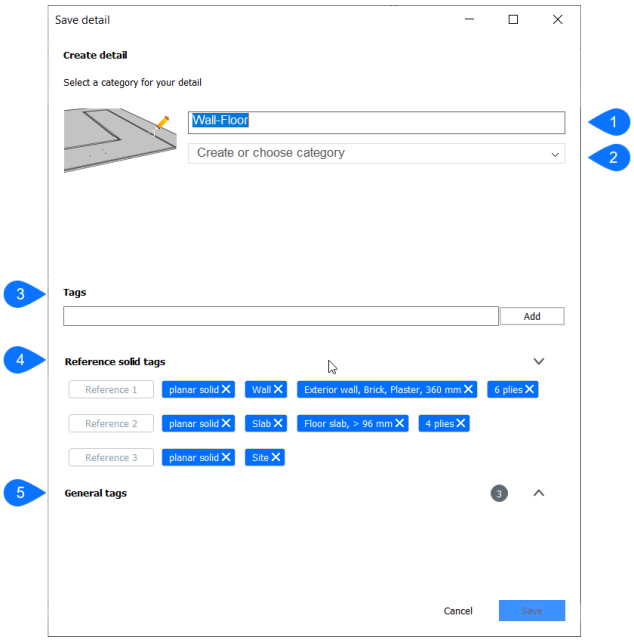
4.12.122. Description

3

Displays the description of the selected block, if it has one.

4.12.123 Save detail dialog box

The Save Detail dialog box allows you to save the selected elements as a detail.



1 Name

2 Category

3 Tags

4 Reference solid tags

5 General tags

4.12.123. Name

1

Sets the name of the new BIM detail.

4.12.123. Category

2

Sets the category of the new BIM detail.

4.12.123. Tags

3

Adds tags to the detail so it can be found easily when searching for this specific detail.

4.12.123. Reference solid tags

4

Displays tags that specify the detail reference entities.

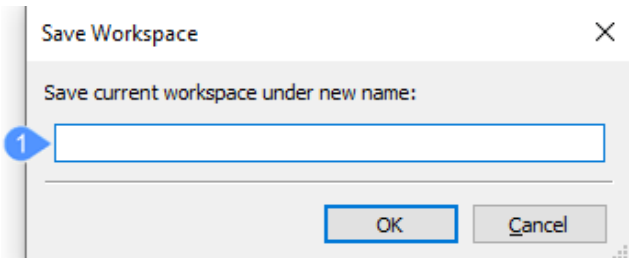
4.12.123. General tags

5

Displays tags that specify the detail in general.

4.12.124 Save workspace dialog box

The **Save Workspace** dialog box allows you save the current workspace under a different name, creating a new workspace.



1 Save current workspace under new name

4.12.124. Save current workspace under new name

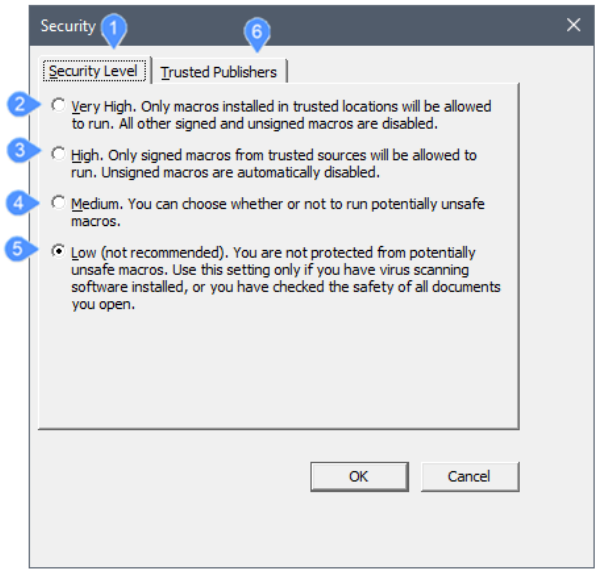
1

Specifies the name for the new workspace. Press **OK** to apply the changes.

You can access it later through the **Workspaces** toolbar or from the status bar.

4.12.125 Security dialog box

The Security dialog box sets the security level to Low, Medium, High or Very High to determine whether VBA macros are allowed to run automatically.



1 Security Level

2 Very High

3 High

4 Medium

5 Low

6 Trusted Publisher

4.12.125. Very High

1

Only VBA macros installed in trusted locations will be allowed to run. All other signed or unsigned macros are disabled.

4.12.125. High

2

Only signed macros from trusted sources will be allowed to run. Unsigned macros are automatically disabled.

4.12.125. Medium

3

BricsCAD displays a dialog box when a VBA macro runs.

The dialog box allows you to choose whether or not to run the macro.

4.12.125. Low

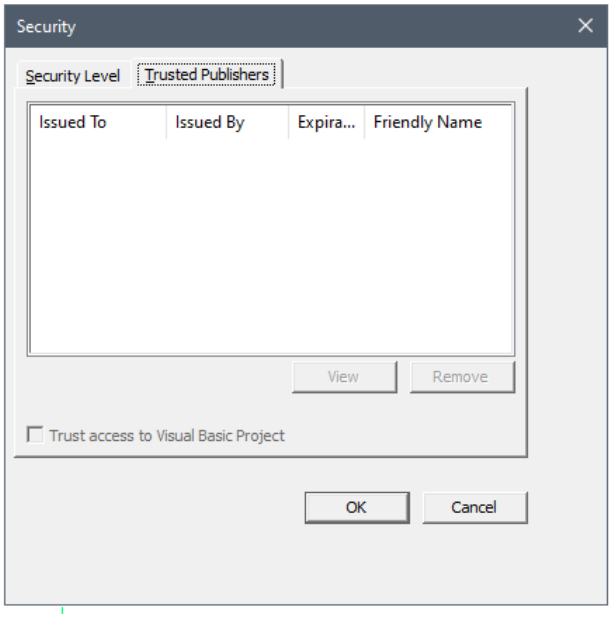
4

All VBA macros are run.

4.12.125. Trusted Sources Option

5

Lists the names of VBA macros that can be trusted. These are the only macros that will run when security is set to High.



Remove

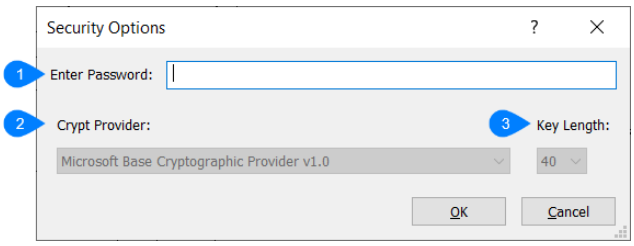
Removes the selected VBA macro from the list.

Trust access to Visual Basic Project

Allows access to the VBA Project.

4.12.126 Security options dialog box

The Security Options dialog box allows you to set a password to protect your drawing from unauthorized access.



1 Enter Password

2 Crypt Provider

3 Key Length

4.12.126. Enter Password

1

Specify a password by entering one or more characters.

4.12.126. Crypt Provider

2

Specifies the encryption level for drawing properties. Choose an encryption provider supplied by the operating system.

4.12.126. Key Length

3

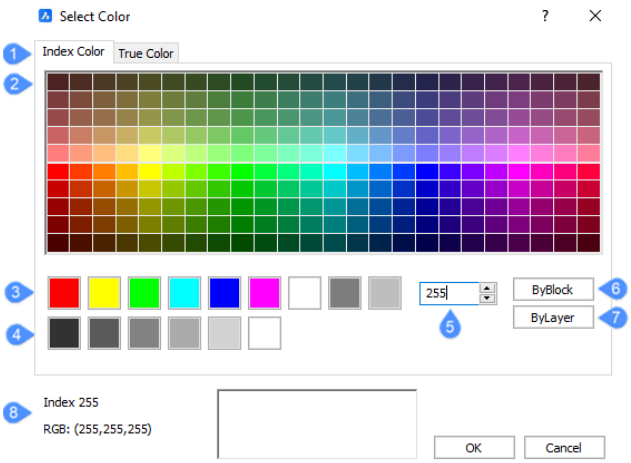
Select a length for the encryption key. The longer the key, the higher the level of protection is, because it takes longer to break the encrypted password.

4.12.127 Select color dialog box

The Select Color dialog box allows you to select a color for layers, grids, dimensions, background etc. It is composed from two tabs: **Index Color** tab and **True Color** tab.

4.12.127. Index Color tab

1



1 Index Color tab

2 Colors

3 Basic colors

4 Shades of gray

5 Color index

6 ByBlock

7 ByLayer

8 Color sample

Colors
Displays the other 240 colors.

Basic colors
Displays the basic colors.

Shades of gray
Displays the shades of grays.

Color index
Displays the index of the selected color. You can insert an index and see the preview of the color at field number 8.

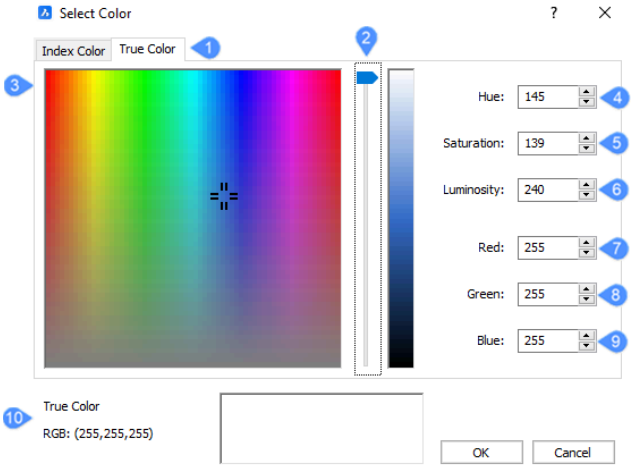
ByBlock
Sets the color to ByBlock in which entities take on the color defined by their block.

ByLayer
Sets the color to ByLayer in which entities take on the color defined by their layer's property.

Color sample
Displays a preview for the selected color and the RGB color value.

4.12.127. True Color Tab

2



1 True Color Tab

2 Luminosity

3 Color

4 Hue

5 Saturation

6 Luminosity

7 Red

8 Green

9 Blue

10 Color sample

Luminosity
Chooses the luminosity of the color.

Color
Chooses hue (left and right) and saturation (up and down).

Hue
Specifies the color's hue. It is ranging from red through green, blue, and pink.
It ranges from 0 to 240.

Saturation
Specifies the intensity of the color, ranges from 0 to 240. Saturation is the intensity of the color, where 0 = gray and 240 = full color.

Luminosity
Specifies the brightness of the color, ranges from 0 to 240. Luminosity is the lightness or darkness, where 0 = black and 240 = white. Hue, saturation and luminosity work together, while red, green and blue work separately.

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

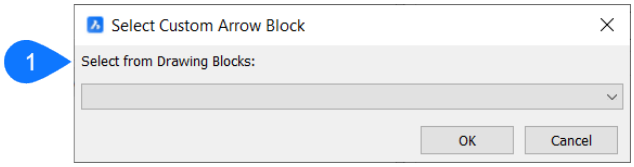
Green
Specifies the amount of green in the color, ranges from 0 (no green) to 255 (full green).

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

Color sample
Displays a preview for the selected color and the RGB color value.

4.12.128 Select Custom Arrow Block dialog box

The Select Custom Arrow Block dialog box allows you to save the selected elements as a detail.



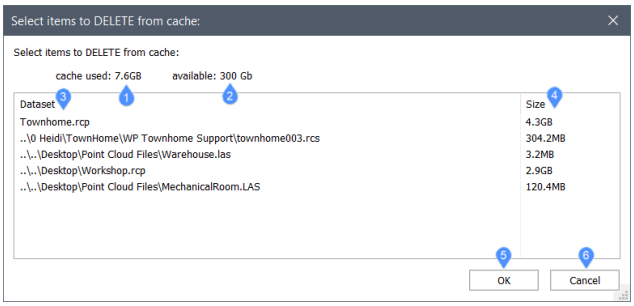
1 Select from Drawing Blocks

4.12.128. Select from Drawing Blocks

1
Specifies one of the blocks in the drawing to use as arrowhead.

4.12.129 Select items to delete from cache dialog box

TheSelect Items To Delete from Cache dialog box enables you to delete point cloud files that are stored in the cache.



1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

4.12.129. Cache used

1
Specifies the total amount of used cache.

4.12.129. Available

2

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

4.12.129. Dataset

3

Displays a list of cached point cloud files including their names and locations. You can select point cloud files from the list to delete them from the cache.

4.12.129. Size

4

Displays the amount of cache used for each point cloud file.

4.12.129. OK

5

Displays the Confirm deleting selected cache items dialog box where you can review the list of point clouds to be deleted.

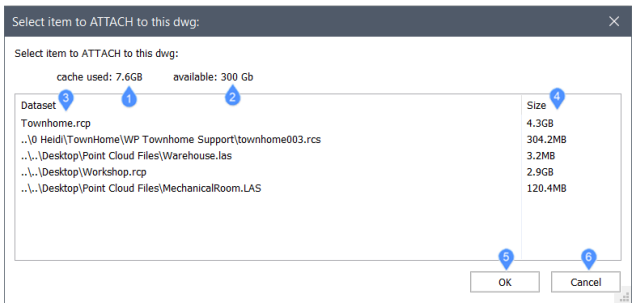
4.12.129. Cancel

6

Closes the dialog box without deleting point cloud files from the cache.

4.12.130 Select item to attach to this DWG

TheSelect item to attach to this DWG dialog box enables you to attach point cloud files that have already been processed and stored in the cache.



1 Cache used

2 Available

3 Dataset

4 Size

5 OK

6 Cancel

4.12.130. Cache used

1

Specifies the total amount of used cache.

4.12.130. Available

2

Specifies the total amount of available cache. The available cache is controlled by the POINTCLOUDCACHESIZE system variable.

4.12.130. Dataset

3

Displays a list of cached point cloud files including their names and locations. You can select a cached file from the list to attach to the current drawing.

4.12.130. Size

4

Displays the amount of cache used for each point cloud file.

4.12.130. OK

5

Displays the Attach point cloud dialog box where you can specify how to attach the selected point cloud file to the current drawing.

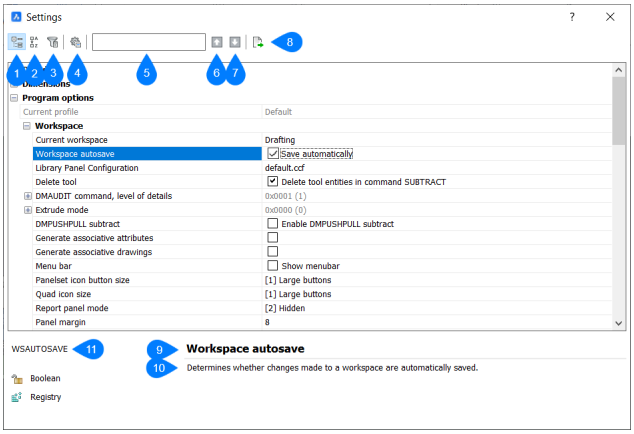
4.12.130. Cancel

6

Closes the dialog box without attaching a point cloud file.

4.12.131 Settings dialog box

The Settings dialog box allows you to change the values of system variables and user preferences.



1 Categorized

2 Alphabetic

3 Show differences

4 Dialog configurations

5 Search bar

6 Next

7 Previous

8 Export

9 Name

10 Description

11 Properties

4.12.131. Categorized
1

Lists variables in related groups.

4.12.131. Alphabetic
2

Lists variables in related groups.

4.12.131. Show differences
3

Lists variables whose values are changed from the initial (default) values.

4.12.131. Dialog configurations
4

Searches for variables, configures how the search is carried out, and specifies how differences are handled. The Configure Settings dialog box opens when clicking the icon.

4.12.131. Search bar
5

Lists variables trough a real-time search field when entering the first few letters of their names or descriptions.

4.12.131. Next
6

Jumps to the next candidate that matches the search phrase.

4.12.131. Previous
7

Jumps to the previous candidate that matches the search phrase.

4.12.131. Export
8

Opens the Export Settings dialog box to allow you to save all settings and values to a CSV file.

4.12.131. Name
9

Displays the name of the selected system variable.

4.12.131. Description
10

Displays a brief description of the selected system variable.

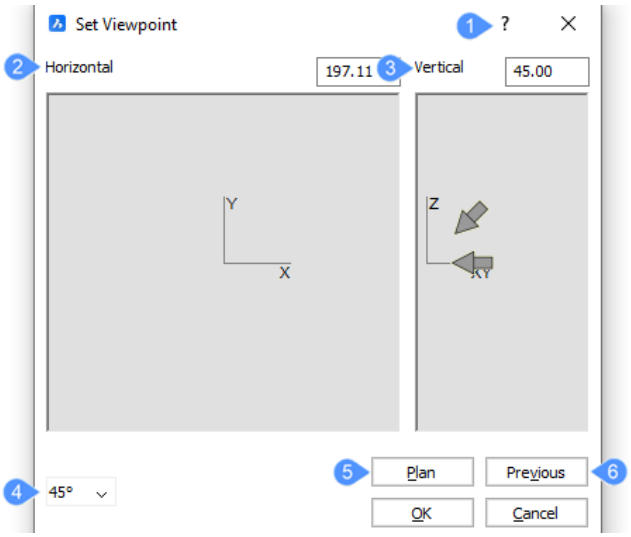
4.12.131. Properties
11

Lists several basic properties/characteristics of the selected system variable.

4.12.132 Set viewpoint dialog box

The Set Viewpoint dialog box allows you to set the current 3D viewport or plan view.

Nota: Turn off the perspective mode (view PERSPECTIVE command), because this command is not available in the Perspective mode.



1 Command reference

2 Horizontal

3 Vertical

4 Angle Preset

5 Plan

6 Previous

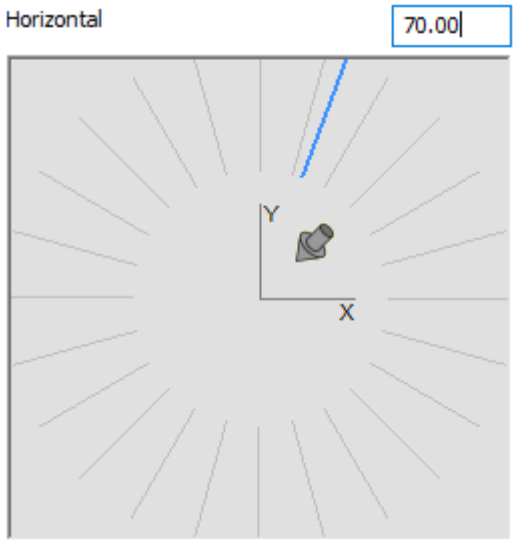
4.12.132. Command reference
1

Opens the Bricsys Help article about the DDVPOINT command.

4.12.132. Horizontal

2

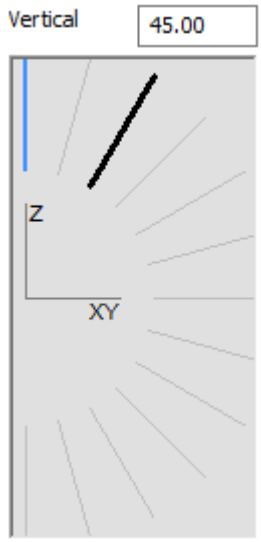
Specifies the horizontal angle from the X - axis. Click a preset angle or type a value in the **Horizontal** field. Click at 0°, 45°, 90°, 135°, 180°, 225°, 270° or 315° in the inner circle. The shape of the arrow indicates whether the view is horizontal or at an angle.



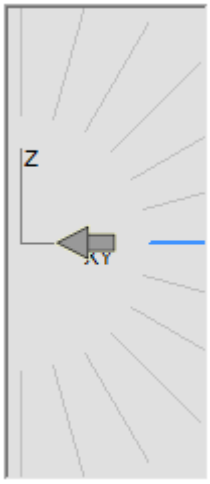
4.12.132. Vertical

3

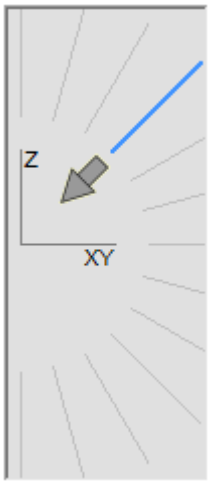
Specifies the vertical angle from the XY plane. Click a preset angle, or else type a value in the Vertical field.



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

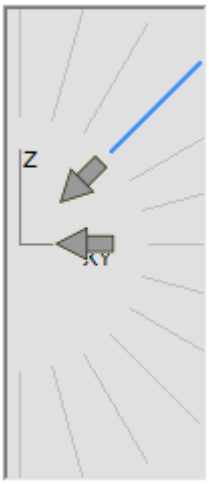


(1)

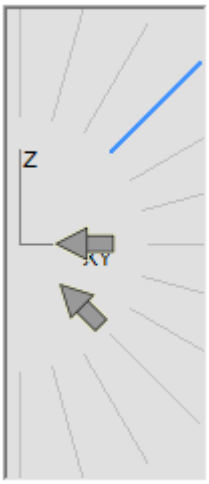


(2)

View direction is 45° down, except for 0°, 90°, 180° and 270° horizontal angles, which generate orthographic views: front, back, left and right. (3) First select the horizontal arrow, then (4) click the 45° down arrow.



(3)



(4)

4.12.132. Angle Preset

4

Sets markers at 5, 15, or 45 degrees; default = 45.

4.12.132. Plan

5

Reverts the 3D viewpoint to 2D plan view.

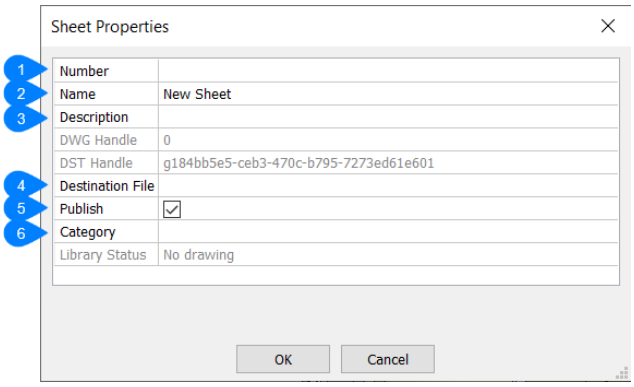
4.12.132. Previous

6

Reverts to the previous viewpoint.

4.12.133 Sheet Properties dialog box

The Sheet Properties dialog box allows you to view and edit properties of a sheet.



1 Number

2 Name

3 Description

4 Destination File

5 Publish

6 Category

4.12.133. Number

1

Specifies the number of the selected sheet.

4.12.133. Name

2

Specifies the name of the selected sheet.

4.12.133. Description

3

Specifies a short description of the selected sheet.

4.12.133. Destination File

4

Specifies the file in which the sheet will be saved.

4.12.133. Publish

5

Toggles the option to publish the currently selected sheet.

4.12.133. Category

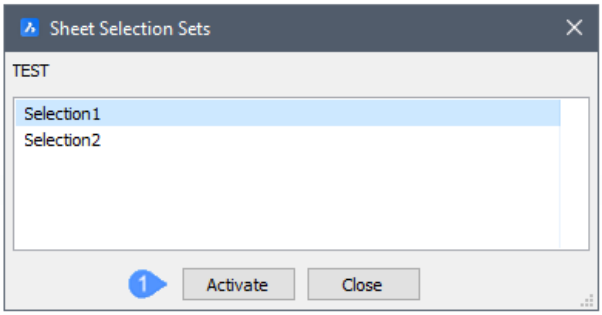
6

Specifies the category of the selected sheet.

4.12.134 Sheet selection sets dialog box

The Sheet Selection Sets dialog box allows you to manage the sheet selections.

To create a sheet selection, right click when multiple sheets are selected to open the **Sheet Selection** context menu and **Save Sheet** selection with a name.



1 Activate

4.12.134. Activate

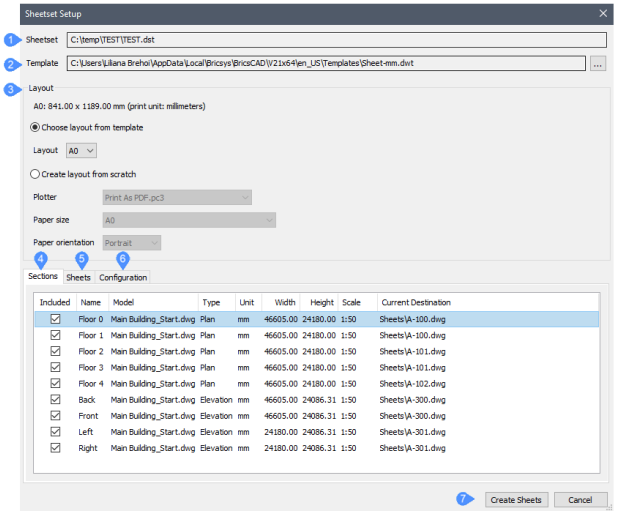
1

Restores the selected selection sheet set.

Right click a selection sheet set to display a context menu.

4.12.135 Sheetset setup dialog box

The **Sheetset Setup** dialog box provides the settings that are used for plotting and publishing.



1 Sheetset

2 Template

3 Layout

4 Sections Tab

5 Sheets Tab

6 Configuration Tab

7 Create Sheets

4.12.135. Sheetset

1

Shows the path of the sheetset data file (.dst) that stores the associations and information that define a sheet set.

4.12.135. Template

2

Shows the path of the current template that contains the page setups for all new sheets.

Click **Browse** () to change the template.

4.12.135. Layout

3

Selects which layout (different paper sizes) the section results will be generated.

- **Choose layout from template**: Allows to select a predefined layout from a drop-down list.
- **Create layout from scratch**: Allows to set:
  - **Plotter**: Choose from the drop-down list with the available plotters.
  - **Paper size**: Choose the paper size from the drop-down list.
  - **Paper orientation**: Choose the paper orientation from the drop-down list that opened.

4.12.135. Sections Tab

4

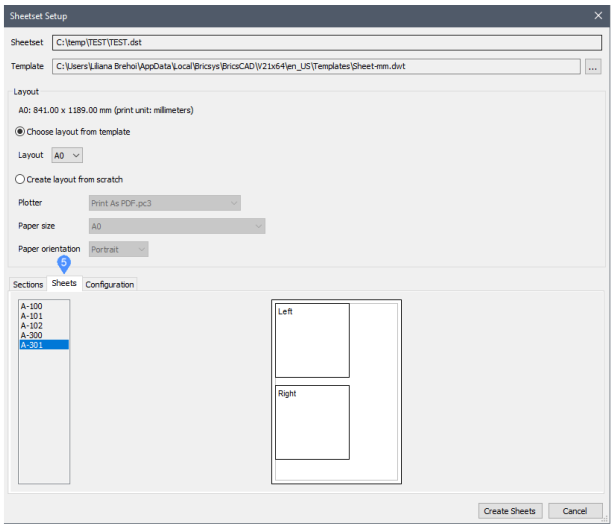
Lists all sections. For each section with an empty **Current Destination** field, a viewport and a sheet view is generated.

- **Included**: Specifies which sections you want to be generated.
- **Name**: Displays the name of the sections.
- **Model**: Displays the name of the model from the sections area.
- **Type**: Displays the section type.
- **Unit**: Displays the drawings unit.
- **Width**: Displays the maximum model width dimension.
- **Height**: Displays the maximum model height dimension.
- **Scale**: Displays the scale for the section sheet. It may be changed pressing right click and selecting a new scale form the drop-down list.
- **Current Destination**: Displays the destination .dwg file of the current section.

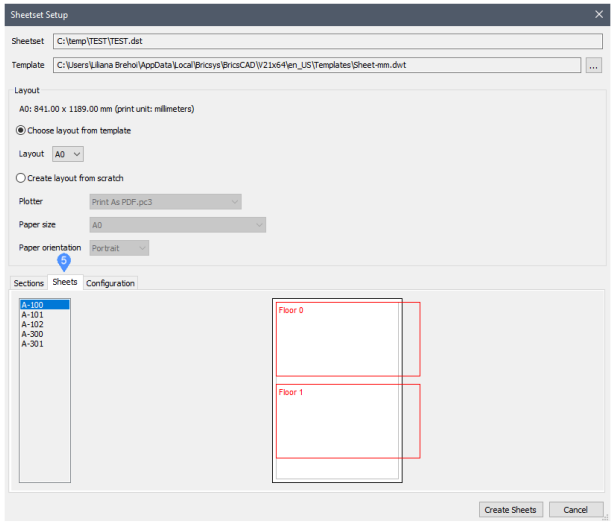
4.12.135. Sheets Tab

5

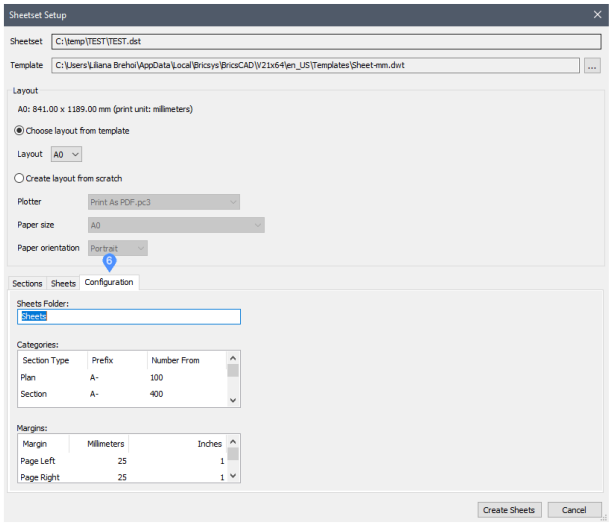
Shows how the viewports will be displayed in the layout.



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



4.12.135. Configuration Tab
6



Slowly double-click (or click and hit Space/Enter) to change the properties:

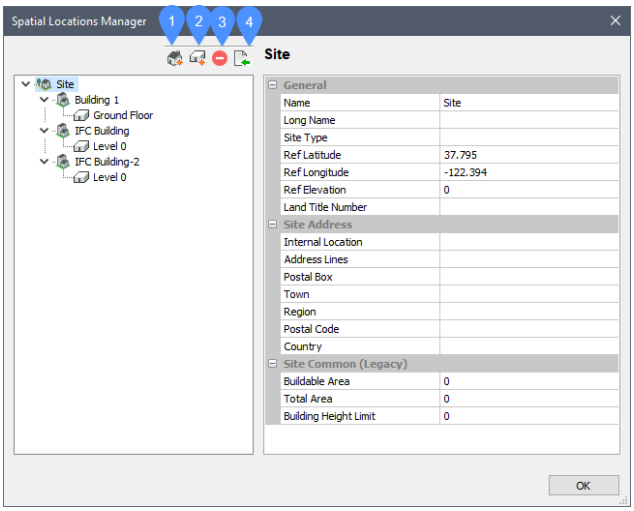
- Under **Sheets Folder** you may change the name of the folder where the sheets will be stored.
- Under **Categories** you can change the **Prefix** and **Number From** for each Section Type.
- Under **Margins** you can set the margins of the page.

4.12.135. Create Sheets
7

The sheets are generated.

4.12.136 Spatial Locations Manager dialog box

The Spatial Locations Manager dialog box allows you to create and edit sites, buildings and stories.



1 New Building

2 New Story

3 Delete

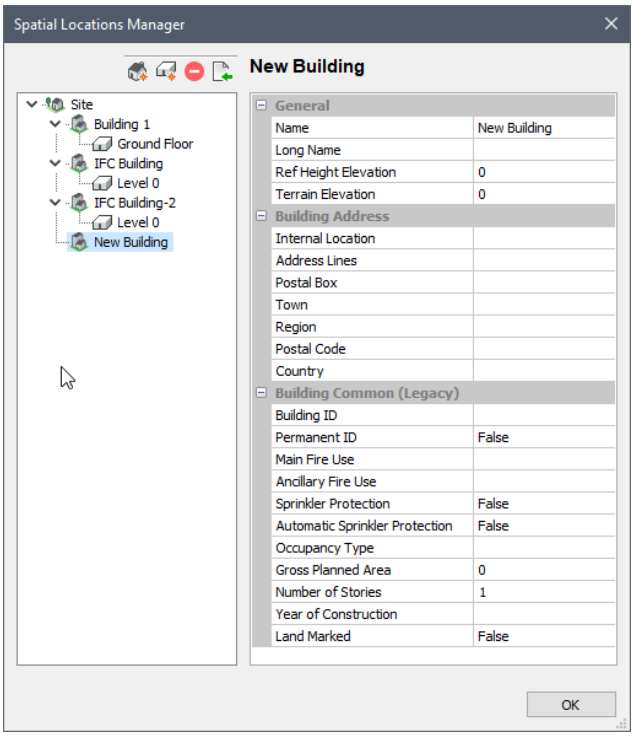
4 Import

4.12.136. New Building

1

Adds a new building to the model.

Fill out the properties grid:

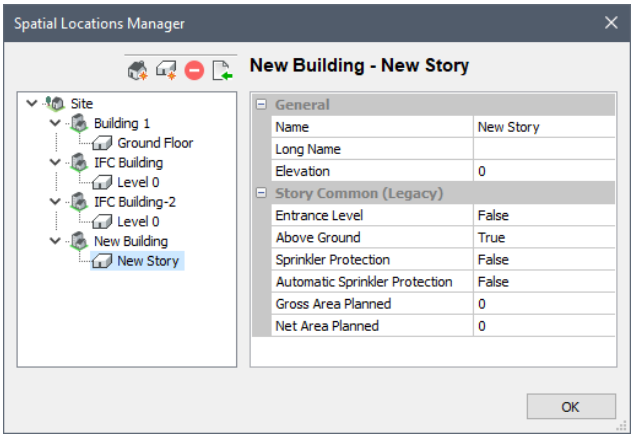


4.12.136. New Story

2

Adds a new story in the selected building.

The properties grid is completed.



4.12.136. Delete

3

Deletes the selected building and story.

4.12.136. Import

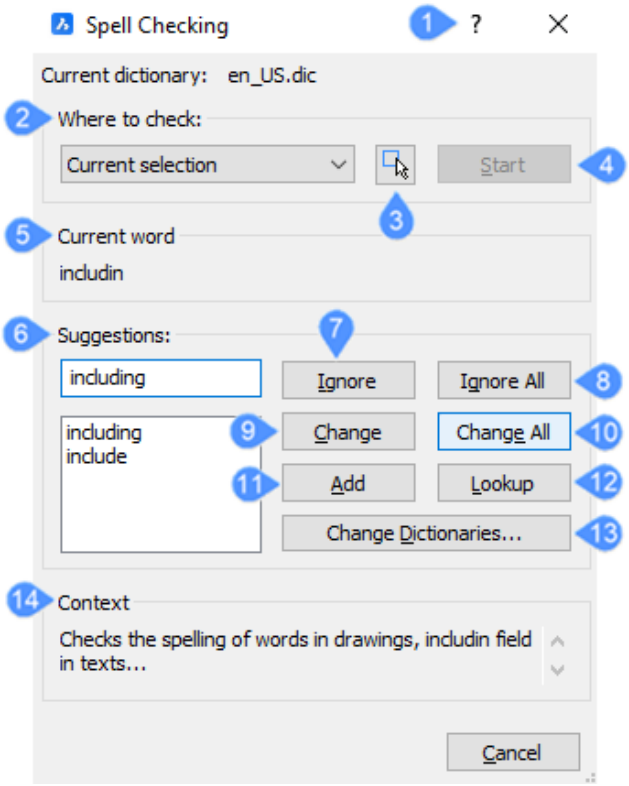
4

Imports spatial locations from a text file.

Displays a Select File to Insert dialog box from which will open the *.txt spatial location file.

4.12.137 Spell checking

The **Spell Checking** dialog box allows you to check the spelling of words in drawings, including fields in texts, multiline texts, leader, multileader, table and block attributes.



1 Command reference

2 Where to check

3 Select entities

4 Start

5 Current word

6 Suggestions

7 Ignore

8 Ignore All

9 Change

10 Change All

11 Add

12 Lookup

13 Change Dictionaries…

14 Context

4.12.137. Command reference

1

Opens the Bricsys Help article about the SPELL command.

4.12.137. Where to check

2

Checks text in:

- **Entire drawing**: Checks all text in the drawing.
- **Selected entities**: Checks text in selected entities. Click the **Select Entities** button to choose text.

4.12.137. Select entities

3

Select the text to be checked. Choose one or more pieces of text and press Enter to return to the dialog box.

4.12.137. Start

4

Starts checking the spelling of words.

4.12.137. Current word

5

Displays the misspelled word.

4.12.137. Suggestions

6

Lists the suggested replacements for misspelled words. Choose a replacement word and then click **Change** or **Change All**.

4.12.137. Ignore

7

Ignores the spelling of the word.

4.12.137. Ignore All

8

Ignores all instances of the word.

4.12.137. Change
9

Changes the word to the one selected from the **Suggestion** list.

4.12.137. Change All
10

Changes all instances of the word to the one selected from the **Suggestion** list.

4.12.137. Add
11

Adds the word to the word list.

4.12.137. Lookup
12

Provides additional alternative spellings for the word selected in the **Suggestions** list.

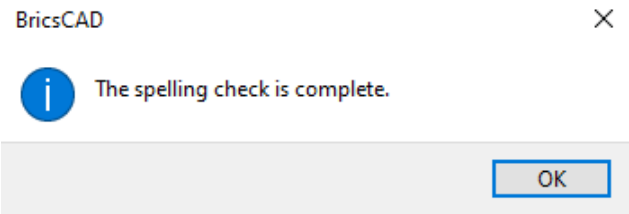
4.12.137. Change Dictionaries...
13

Changes the dictionary used for spell checking. Displays the **Change Dictionaries** dialog box.

4.12.137. Context
14

Displays the context in which the misspelled word appears.

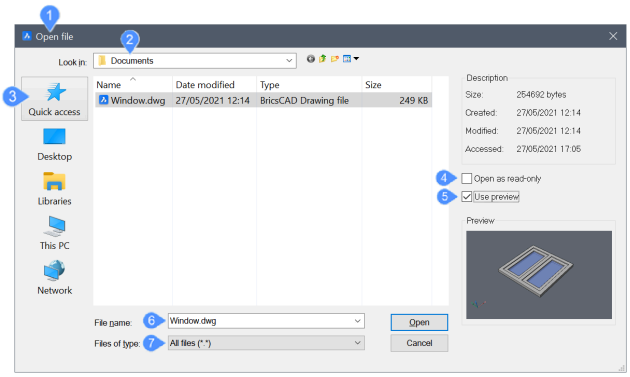
When the spelling check is completed, the following message box appears:



4.12.138 Standard open file dialog box

The Standard open file dialog box allows you to open or import a file.

This dialog box can be accessed using multiple commands.



1 Dialog name

2 Look in

3 Quick access

4 Open as read-only

5 Use preview

6 File name

7 Files of type

4.12.138. Dialog name

1

Displays the dialog name depending on the command used.

4.12.138. Look in

2

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

4.12.138. Quick access

3

The Quick access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate the Quick access with folders that you want to be always available, not only with folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

The Quick access folders are available during both open and save operations.

4.12.138. Open as read-only

4

Tick the checkbox to open the file as read-only. Consequently, you do not have permission to write new information or edit the data in the file.

4.12.138. Use preview
5

Tick the checkbox to view a part of the file before opening.

4.12.138. File name
6

Specifies the name of the file.

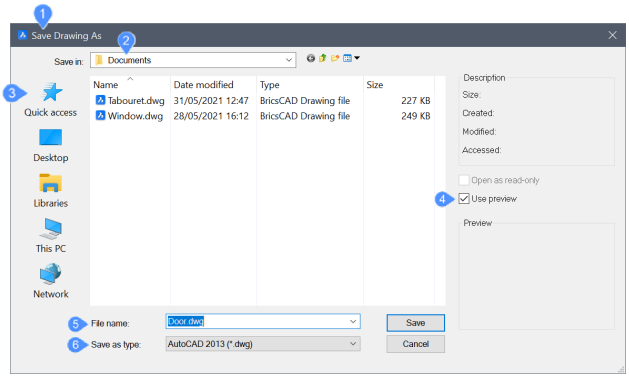
4.12.138. Files of type
7

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

4.12.139 Standard save file dialog box

The Standard Save File dialog box allows you to save or export a file.

This dialog box can be accessed using multiple commands.



1 Dialog name

2 Save in

3 Quick access

4 Use preview

5 File name

6 Files of type

4.12.139. Dialog name
1

Displays the dialog name depending on the command used.

4.12.139. Save in
2

Allows you to specify the location of your file. To change it to a different disk drive or folder, click the arrow and select another disk drive or folder from the drop-down list.

To scroll the list, press the F4 key.

4.12.139. Quick access

3

The Quick Access is available if USESTANDARDOPENFILEDIALOG = 1.

You can populate Quick access with folders that you want to be always available, not only for folders that have being used recently. The folders shown in Quick access can be configured in Windows Explorer.

Quick Access folders are available during both open and save operations.

4.12.139. Use preview

4

Tick the checkbox to see a part of the file before saving.

Nota: A preview is available when selecting an existing file to replace it.

4.12.139. File name

5

Specifies the name of the file. You can accept the predefined name or you can specify a different name.

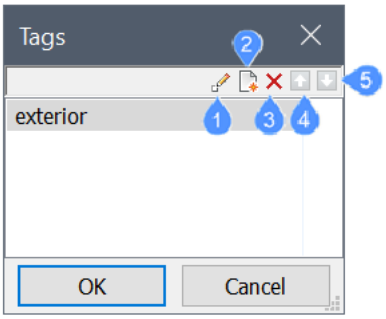
4.12.139. Files of type

6

Displays a list of valid file types. You can select a file type to filter the files list so that it only shows files matching that type.

4.12.140 Tags dialog box

The Tags dialog box allows you to add or edit compositions and physical materials tags.



1 Edit

2 New

3 Delete

4 Move up

5 Move down

4.12.140. Edit

1

Edits the name of the selected tag.

4.12.140. New

2

Adds a new tag at the bottom of the list.

4.12.140. Delete

3

Deletes the selected tag.

4.12.140. Move up

4

Moves up the selected tag.

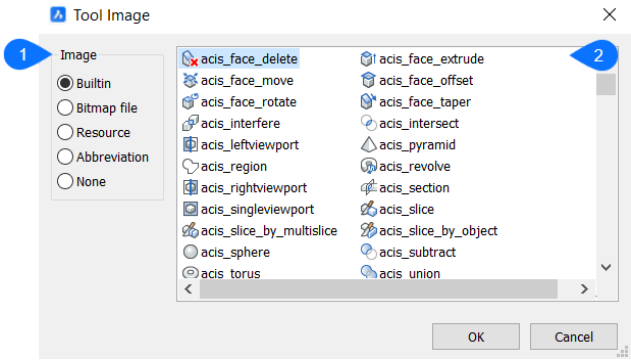
4.12.140. Move down

5

Moves down the selected tag.

4.12.141 Tool image dialog box

The Tool Image dialog box allows you to specify the image of tools.



1 Image

2 List of tool images

4.12.141. Image

1

Specifies the way to set the tool images.

Builtin

Specifies the tool image from one of the builtin tool images from the list.

Bitmap File

Specifies the tool image from a bitmap file.

Resource

Specifies the resource of the tool image.

Abbreviation

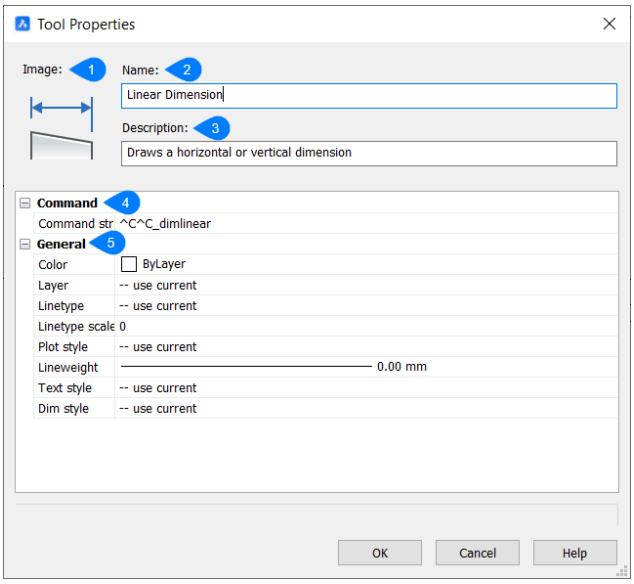
Specifies an abbreviation for the tool instead of an image.

None
No image or abbreviation is used for the tool.

4.12.141. List of tools

2
Displays a list of built in tool images when the **Builtin** option is selected.

4.12.142 Tool Properties dialog box



1 Image

2 Name

3 Description

4 Command String

5 General

4.12.142. Image

1
Displays an image of the selected tool.

4.12.142. Name

2
Specifies the name of the selected tool.

4.12.142. Description

3
Specifies the description of the selected tool.

4.12.142. Command String

4

Specifies the macro to run when you click the icon; you can use any macro characters used for menus and toolbar buttons.

4.12.142. General

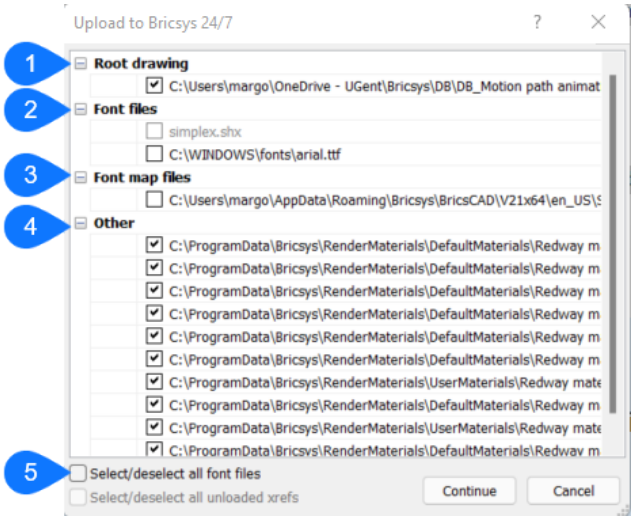
5

Specifies the properties of the entity; the properties vary, depending on the tool.

4.12.143 Upload to Bricsys 24/7 dialog box

The Upload to Bricsys 24/7 dialog box allows you to upload packages of drawing files and it dependencies to Bricsys 24/7

Nota: The project needs to be saved before uploading it to Bricsys 24/7.



1 Root drawing

2 Font files

3 Font map files

4 Other

5 Select/deselect all font files

4.12.143. Root drawing

1

Displays a list of the current drawing files.

4.12.143. Font files

2

Displays a list of the currently used font files.

4.12.143. Font map files
3

Displays a list of the currently used font map files.

4.12.143. Other
4

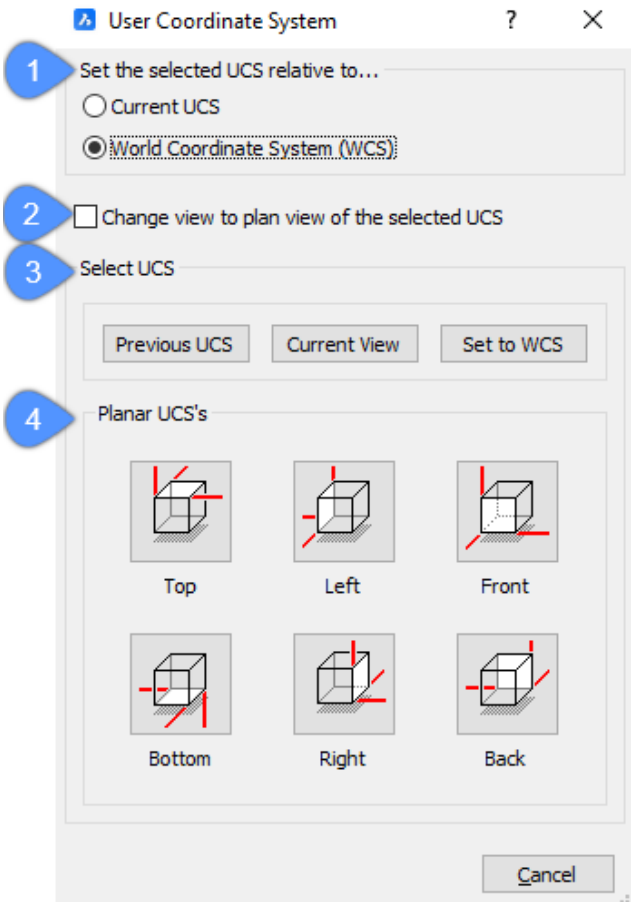
Displays a list of other used files such as render material images.

4.12.143. Select/deselect all font files
5

When checked, all font files and font map files are checked if this was not the case before. When unchecked, all font files are unchecked if there were any checked before. Pressing the **Continue** button will ask you to sign in Bricsys 24/7 to complete the uploading process.

4.12.144 User coordinate system dialog box

The **User Coordinate System** dialog box allows you to set the UCS to a specified viewpoint.



1 Set the selected UCS relative to...

2 Change view to plan view of the selected UCS

3 Select UCS

4.12.144. Set the selected UCS relative to...

1

Specifies how the new UCS is set:

- **Current UCS**: Sets the new UCS relative to the current UCS.
- **World Coordinate System**: Sets the new UCS relative to the WCS. World Coordinate System is the default coordinate system when no UCS is set and it cannot be changed.

4.12.144. Change view to plan view of the selected UCS

2

Toggles whether the plan view is displayed when switching UCSES:

- **Yes**: plan view of the new UCS is displayed.
- **No**: the viewpoint does not change.

4.12.144. Select UCS

3

Sets the UCS to one of the following coordinate system. Clicking one of these buttons sets the UCS and then immediately exits the dialog box.

- **Previous UCS**: Changes the UCS to the previous UCS.
- **Current View**: Changes the UCS to match the current viewpoint.
- **Set to UCS**: Changes the UCS to match the WCS.

4.12.144. Planar UCS's

4

Sets the UCS to one of the following standard orthographic views. After you click one of the options, the program sets the UCS and then exits the dialog box.

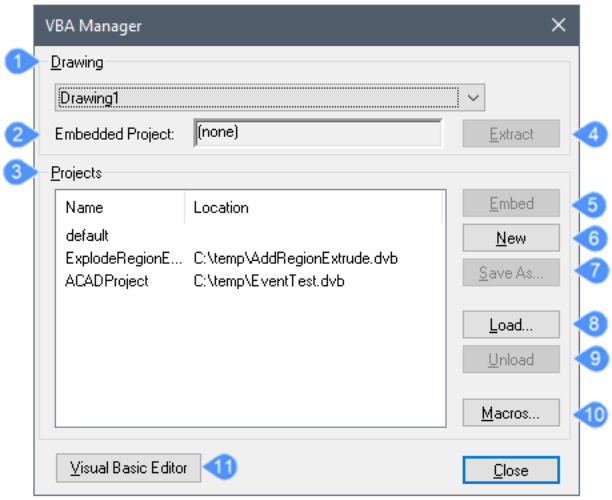
- **Top**: Changes the UCS to match the top view.
- **Left**: Changes the UCS to match the left view.
- **Front**: Changes the UCS to match the front view.
- **Bottom**: Changes the UCS to match the bottom view.
- **Right**: Changes the UCS to match the right view.
- **Back**: Changes the UCS to match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

4.12.145 User profile manger dialog box

The User Profile Manager dialog box allows you to set the current user profile, to create, copy, delete, import, and export profiles.

4.12.146 VBA manager dialog box

The VBA Manager dialog box allows you to manage VBA projects.



1 Drawing

2 Embedded Project

3 Projects

4 Extract

5 Embed

6 New

7 Save As

8 Load

9 Unload

10 Macros

11 Visual Basic Editor

4.12.146. Drawing

1

Displays the names of drawings currently open in BricsCAD:

Choose a drawing name from the drop-down list; the embedded project is listed in the Embedded Project text box, if any.

4.12.146. Embedded Project

2

Displays the name of the embedded project, if any.

4.12.146. Projects

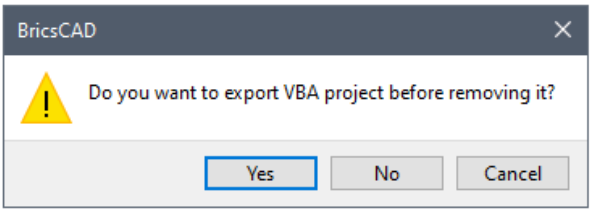
3

Name

Displays the names of projects loaded into BricsCAD at this time.

Location
Displays the drive and folder names of DVB and VBI files.

4.12.146. Extract
4
Removes the project from the current drawing; displays dialog box:



Yes: displays the Save As dialog box:

Name the project and click **Save**.

No: removes the project from the drawing without saving it as a DVB file.

Cancel: cancels the command; the project remains in the drawing.

4.12.146. Embed
5
Embeds a project in the current drawing. If the project is a new one, and has not yet been saved, this option displays the *Save As* dialog box.

Each drawing can contain just one embedded project; after one project is embedded, the Embed button becomes unavailable. To change embedded projects, use the Extract option to remove the current project.

4.12.146. New
6
Creates a new VBA project; gives the generic name VbaProject . Use the Save As option to rename and save the DVB or VBI project file.

4.12.146. Save As
7
Saves the selected project by another name. Displays the *Save As* dialog box.

4.12.146. Load
8
Loads DVB and VBI project files; displays the *Open* dialog box.

4.12.146. Unload
9
Unloads the selected project without a warning.

4.12.146. Macros

10

Displays the *Run BricsCAD VBA Macro* dialog box.

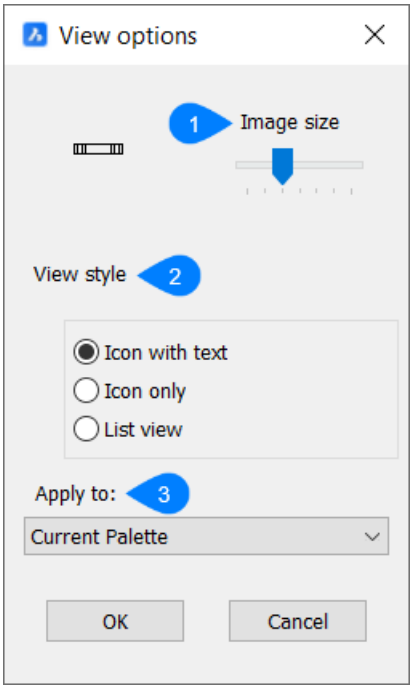
4.12.146. Visual Basic Editor

11

Opens the VBA integrated development environment.

4.12.147 View Options dialog box

The View Options Dialog Box allows you to customize how the tools in the tool palettes panel are shown.



1 Image size

2 View style

3 Apply to

4.12.147. Image size

1

Sets the size of the displayed icons by sliding the indication arrow.

4.12.147. View style

2

Specifies how the tools are shown in the palette.

Icon with text

Displays the tools in a grid view with their icon and name.

Icon only

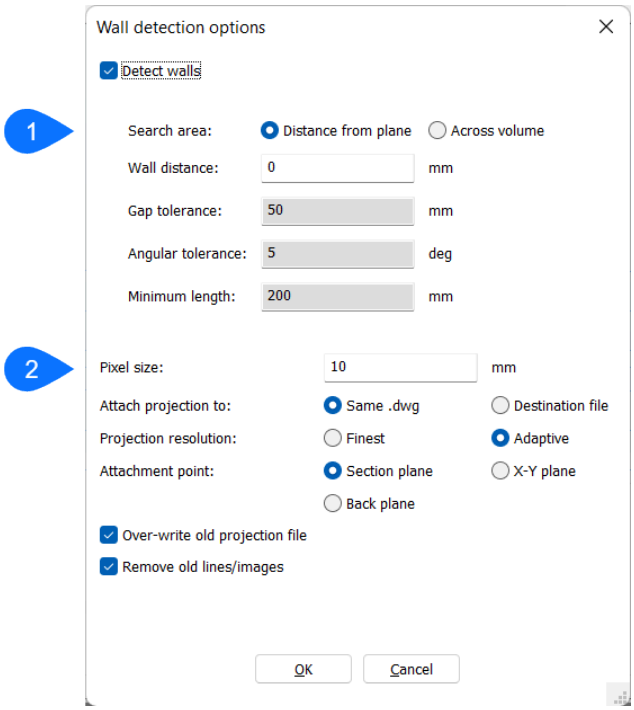
Displays the tools in a grid view with their icon only.

List view
Displays the tools in a list view with their icon and name next to it.

4.12.147. Apply to
3
The changes made can either be applied to the current palette only, or to all palettes.

4.12.148 Visual settings dialog box

4.12.149 Wall detection options dialog box



1 Parameters that control 2D line drawing

2 Parameters that control the generated raster image

4.12.149. Parameters that control 2D line drawing
1
Detect walls
When checked, lines representing walls will be generated with the raster image. When unchecked, only a raster image is created.

Search area

- **Distance from plane**: detects walls in a given radius from a plane. This option is often used to generate vertical sections.
- **Across volume**: detects all walls in the point clouds. This option is often used to generate floor plans.

Wall distance
Controls the search area for wall detection.

Nota: This option is only available when 'Distance from plane' is selected.

Gap tolerance
Controls the search area for the algorithm to draw lines. Gaps smaller than the specified value will be closed.

Angular tolerance
Controls the angular tolerance of the generated lines. A high tolerance will result in pronounced right angles. This is useful for perpendicular plans.
Generated lines with a smaller angle than the specified value will be merged.

Minimum length
Controls the minimum length of a generated line. Lines with a smaller length than the specified value will not be generated.

4.12.149. Parameters that control the generated raster image 2

Pixel size
Controls the pixel size of the generated image.

Nota: the pixel size is in model units. Higher pixel size results in a lower resolution, resulting in a faster process.

Attach projection to
Determines where the resulting image is generated.

Same.dwg
The projection is made in the same file

New destination
Creates a new .dwg file to make the projection.

Nota: You can overwrite a previous file or create a new one.

Projection resolution

Finest
Produces a sharp and detailed background picture.

Adaptive
Produces a less detailed and softer background image.

Attach point
Determines the insertion point for the generated image.

- In-place: places the lines and image in the same location and orientation as the sectionplane.
- X-Y plane: aligns the section with the X_Y axis.

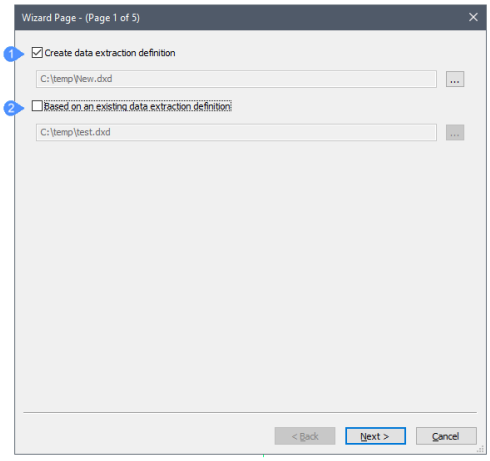
  Nota: The section can be edited in Top view as a regular 2D drawing. This is practical for vertical sections.

Over-write old projection file
When checked, the existing projection file will be overwritten.

4.12.150 Wizard page dialog box

The Wizard Page dialog box guides you to export entity properties, block attributes, and drawing information to a table or to an external data file.

4.12.150. Wizard Page 1 Dialog Box 1



1 Create data extraction definition

2 Based on an existing data extraction definition

Create data extraction definition

Creates a new .dxd (Data eXtraction Definition) file:

1 Tick the checkbox.

2 Click the Browse button.

3 Select a folder in the Save Data Extraction file dialog box.

4 Enter a name in the File name field.

5 Click the Save button.

Based on an existing data extraction definition

Uses an existing .dxd file as the template for the new one to perform the same type of extraction in a different drawing or to edit the data extraction definition:

1 Tick the checkbox.

2 Click the Browse button.

3 In the Open Data Extraction Definition file dialog box, select a .dxd file.

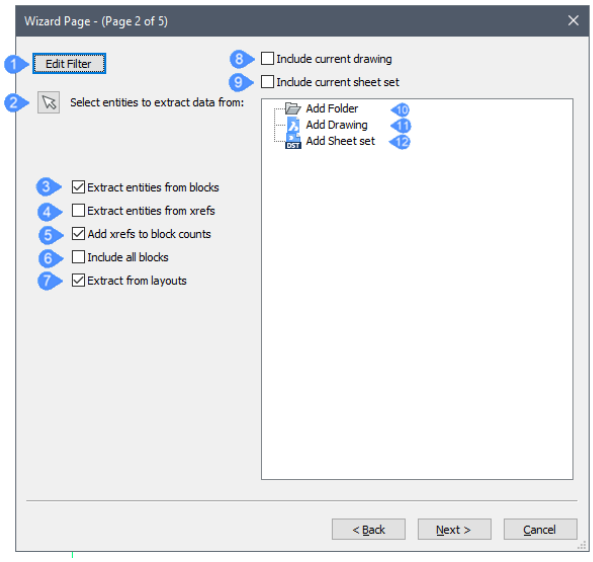
4 Click the Open button.

Newly created custom *.dxd files can be used to create schedules in the Project Browser.

.dxd files can be edited in a text editor such as MS Notepad.

Click **Next** for the next page.

4.12.150. Wizard Page 2 Dialog Box 2



1 Edit filter

2 Select entities to extract data from

3 Extract entities from blocks

4 Extract entities from Xrefs

5 Add Xrefs to block counts

6 Include all blocks

7 Extract from layouts

8 Include current drawing

9 Include current sheet set

10 Add Folder

11 Add Drawing

12 Add Sheet Set

Guides to select entities for Data Extraction and apply filters for them before it is extracted in order to have less information to sort later.

The DXEVAL system variable controls when the update notification occurs in the drawings where changes affect the extracted data.

Edit filter
Creates or edits a Filter.

Select entities to extract data from
Dismisses the dialog box temporarily so that you can select entities in the drawing; enter **Ctrl+A** to select all entities in the drawing. Press **Enter** or right-click to stop selecting entities.

Extract entities from blocks
Includes entities nested in blocks.

Extract entities from Xrefs
Includes entities in externally referenced drawings (Xref files).

Add Xrefs to block counts
Counts selected Xrefs as blocks.

Include all blocks
Includes all block entities.

Extract from layouts
Includes entities from paper space of the drawing.

Include current drawing
Adds the current drawing from which to extract data.

Include current sheet set
Adds the current sheet set from which to extract data.

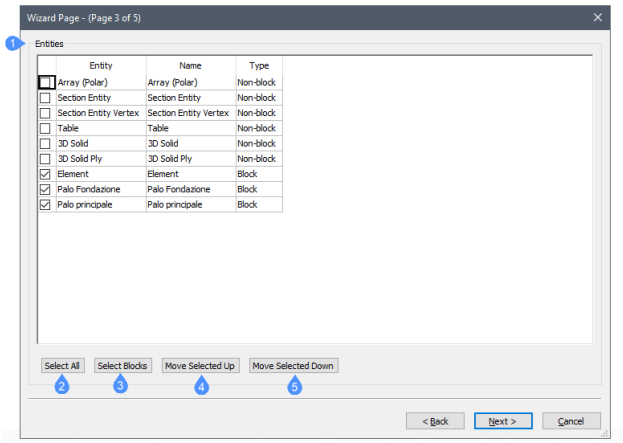
Add Folder
Adds a folder of drawing from which to extract data.
Double-click to open the *Folder Options* dialog box.

- **Folder**: click the **Browse** button, then select a folder in *Choose a Folder* dialog box.
- **Options**
  - **Include subfolders**: tick the checkbox to include subfolders.
  - **Use a wildcard to filter drawings to be included**: tick the checkbox, then type a text string in the filter field. For example, House*.dwg includes all files starting with 'House'.

Add Drawing
Adds a drawing from which to extract data with a double click.

Add Sheet Set
Adds a sheet set from which to extract data with a double click.
Click **Next** for the next page.

4.12.150. Wizard Page 3 Dialog Box
3



1 Entities

2 Select All

3 Select Blocks

4 Move Selected Up

5 Move Selected Down

Controls which entity types are included or not in the data extraction procedure.

Select multiple items:

- To select more than one item in a row: select an item, and then hold down the **Shift** key while clicking additional items.
- To select multiple items: hold down the **Ctrl** key while selecting multiple entities.
- To unselect items, click them again while holding down the **Ctrl** key.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

Entities

- **Entity**: lists the entity types and blocks in the selection, according to the options set in the *Data Extraction* dialog box.
- **Name**: entity names. Accept the default names or type a customized name. The entity type name displays in the default Name field in the data extraction export.
- **Type**: reports the type of the selected entity: Non-Block, Block or Block with attributes.

Select All

Selects all items for data extraction.

Select Blocks

Selects only blocks in the list, because they usually contain attributes.

Move Selected Up

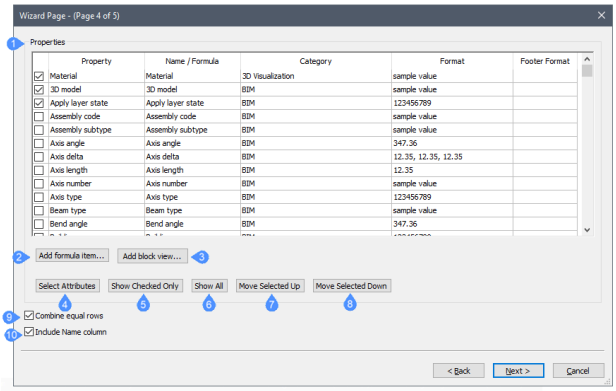
Move the selected item(s) up the list.

Move Selected Down

Move the selected item(s) down the list.

Click **Next** for the next page.

4.12.150. Wizard Page 4 Dialog Box 4



1 Properties

2 Add formula item

3 Add block view

4 Select Attributes

5 Show checked only

6 Show All

7 Move Selected Up

8 Move Selected Down

9 Combine equal rows

10 Include Name column

Select the properties to be included in the data extraction file: tick the checkbox to check/uncheck a property. Use the same techniques to select more than one item as described in Wizard Page 3 Dialog Box.

To sort the list, click the title in the column heading. Click again to sort in reverse order.

Properties

- **Property**: lists all properties of the selected entities.
- **Name /Formula**: lists the name of the selected entities or the formula. Accept the default names or type a customized name. The property names display in the column headings in the data extraction export.
- **Category**: reports the category of the selected entities.
- **Format**: formats the item in the export file. Click to display the dialog box. The layout of the dialog box depends on the selected item, just like the Format section of the Field dialog.
- **Footer Format**: selects from the drop-down list opened with right mouse button the Footer settings:

Add formula item

Adds a formula item.

Add block view
Adds BlockViewProperty as Block view category.

Select Attributes
Select only items that contain attributes.

Show checked only
Show only selected items, hiding those that are not selected.

Show All
Shows all items, whether checked or not.

Move Selected Up
Move the selected item(s) up the list.

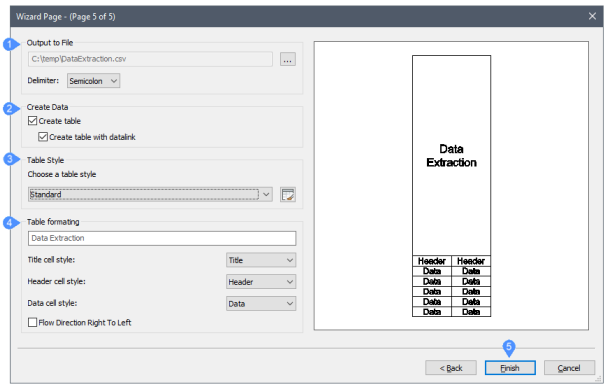
Move Selected Down
Move the selected item(s) down the list.

Combine equal rows
Combine rows that contain the same entities.

Include Name column
Toggles the display of the Name column in the exported data.

Click **Next** for the next page.

4.12.150. Wizard Page 5 Dialog Box 5



1 Output to File

2 Create Data

3 Table Style

4 Table formatting

5 Finish

Output to File
Exports the data in a delimiter separated format (.csv):
Click the **Browse** button, give the data file a name, and then click **Save**. If you select an existing .csv file, you will be prompted to overwrite it.

Delimiter: specifies the separator between fields of data:

- Semicolon
- Comma

  Nota: Multiply composition names contain a comma, therefore do not select comma as the delimiter character in case multiply compositions are processed.
- Space
- Tab

Create Data
Exports the data in a Table:

- Create table: creates a static table.
- Create table with datalink: creates a dynamic table. Executing the DataLinkUpdate command synchronizes the data in the table with the model. Save the model first. You will be prompted to save the *.dxd definition file with the table.

Table Style
Select a table style from the drop-down list.
Click the **Table Style** icon; edit an existing table style or create a new one.

Table formatting
Type a title in the Title field.

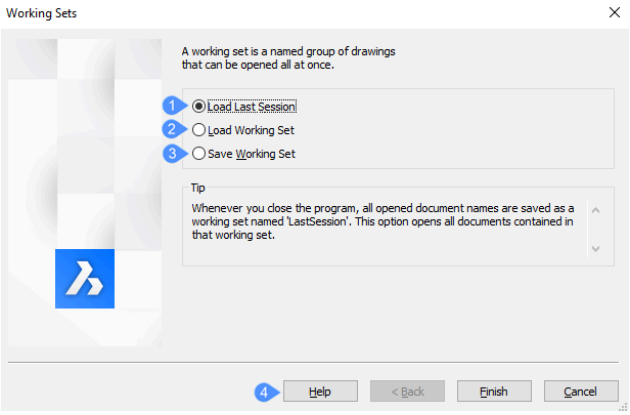
- Title cell style: sets the title cell style.
- Header cell style: sets the Header cell style.
- Data cell style: sets the Data cell style.
- Flow Direction Right to Left: indicates the flow direction from right to left.

Finish
Depending on the selected output:

- Extracts the data and saves them to the .csv file. The file can be opened in a spreadsheet or database program.
- Prompts you to specify an insertion point for the table(s).

4.12.151 Working sets dialog box

The **Working Sets** dialog box allows you to create and load sets of drawing files, making it convenient to load and entire group of drawings at once.



1 Load Last Session

2 Load Working Set

3 Save Working Set

4 Help

4.12.151. Load Last Session

1

Opens all drawings that where open when the last BricsCAD session was closed.

After you click the **Finish** button, BricsCAD opens all the drawings that were open the last time you closed the program.

4.12.151. Load Working Set

2

Click the **Next** button and choose the name of a workset from the **Select a Workset** list. Click the **Finish** button. The program opens all the drawings that are part of the workset.

4.12.151. Save Working Set

3

Saves drawings currently open by name as a working set. Click the **Next** button.

When you click on the **Save** button, the **Save workset** dialog box opens. Click the **Finish** button to close the dialog box.

Nota: Working sets are saved in the registry and so they are defined differently for each user profile. See the PROFILEMANAGER command to learn more about user profiles.

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

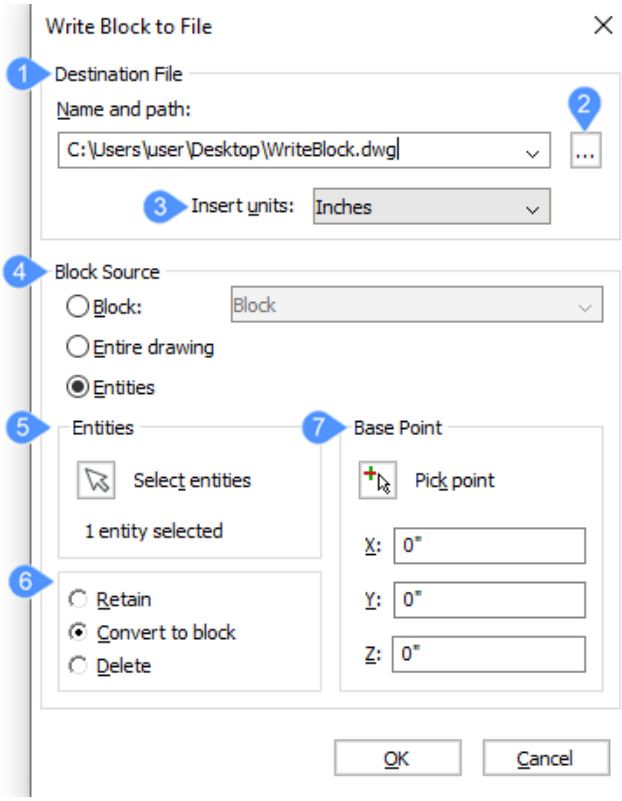
4.12.151. Help

4

Opens the Bricsys Help article about WORKSETS command.

4.12.152 Write block to file dialog box

The **Write Block to File** dialog box allows you to write blocks and other drawing parts to a separate drawing file.



1 Destination file

2 Save Block

3 Insert units

4 Block Source

5 Entities

6 Options

7 Base Point

4.12.152. Destination file

1

Specifies what the file should be named and where it should be saved.

- **Name and path**: Specify the default path and file name (new block.dwg).

4.12.152. Save Block

2

Changes the path and file name. Opens the **Save Block** dialog box.

4.12.152. Insert units
3

Specifies the units to be used by the exported entities when they are later inserted into other drawings.

You can choose the measurement unit from the drop-down list.

Units are saved in the INSUNITS variable.

4.12.152. Block Source
4

Selects the part of the drawing to save to a new block:

- **Block**: Saves a block from the drawing. This option is unavailable when no blocks exist in the drawing. When this option is selected, the other options become unavailable. You can choose from the drop-down list the name of a block you want to save.
- **Entire drawing**: Saves the entire drawing to the file. When this option is selected, all other options become unavailable.
- **Entities**: Saves entities that you select to the file.

4.12.152. Entities
5

When the **Block Source** is set to **Entities**, you are prompted at the Command line to choose one or more entities using any selection method.

4.12.152. Options
6

Specifies what to do with the selected entities after you click **OK** to close the dialog box:

- **Retain**: Keeps the selected entities as they are; this is like copying the selection to a new drawing.
- **Convert to block**: Converts the selected entities to a block in the current drawing.
- **Delete**: Deletes the entities you had selected in the drawing; this is like moving the selection to a new drawing.

  Nota: If the **Delete** option was set, the OOPS command brings back the erased block entities.

4.12.152. Base Point
7

Specifies the origin point of the saved entities in the new file. You can pick a point in the drawing or specify the X, Y, Z coordinates.

5. Gestione disegni

5.1 Creating a drawing

A new drawing is always created as a copy of a template drawing. The default template is saved through the BASEFILE system variable.

In BricsCAD you can create new drawings:

- In the **Start** tab.
- Using the QNEW command.
- Using the NEW command.
- Using the NEWWIZ command.
- Double-clicking a *.dwt file in a Windows Explorer window.

5.1.1 Specifying the template folder

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select **Template Path** (TEMPLATEPATH system variable).

3 Click the **Browse** button.

4 On the **Browse for folder** dialog box, select a folder, then click the **OK** button.

5.1.2 Setting the default template drawing

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Files** > **Templates**, select the BASEFILE system variable.

3 Click the **Browse** button.

4 On the **Choose a File** dialog box, select a drawing file (*.dwg) or drawing template (*.dwt), then click the **Open** button.

5.1.3 Using the Start tab

1 Click the **Start** tab.

Launch the GOTOSTART command to open the **Start** tab if necessary.

The name of the current drawing template is displayed beneath the **New Drawing** button.

2 Do the following:

- (option) Click the template list* button, to choose a different template drawing.
- Click the **New Drawing** button.

* All template drawings in the **Template** folder matching the current workspace (Drafting, Modeling, Mechanical or BIM), are listed here. Choose **More…** to see the full content of the **Template** folder.

5.1.4 Creating a drawing using the default template drawing

First method:

Click the + button at the right hand side of the document tabs.

Second method:

Launch the QNEW command.

Third method:

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog, choose **Start from Default template**, then click the **Next** button.

Nota: If the default template drawing is not specified, the **Create New Drawing** wizard is launched depending on the current value of the STARTUP system variable, otherwise the **Select Template** dialog box appears.

5.1.5 Creating a drawing using the New command

1 Launch the NEW command.

2 On the **Create New Drawing** dialog box, choose **Start from Scratch**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings.

Click the **Finish** button to create the drawing.

Nota: Using the above procedure, it depends on the status of the BASEFILE and PSTYLEPOLICY variables whether **CTB** (Color table) or **STB** (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

5.1.6 Setting the Plot style policy

1 Open the **Settings** dialog box.

2 Under **Program Options** > **Plot and publish**, select **Plot style policy** (PSTYLEPOLICY system variable).

3 Do one of the following:

- Choose **Associate an entity's plot style with its color** for Color dependent (CTB) plot styles.
- Choose **No association between color and plot style** for Named (STB) plot styles.

5.1.7 Creating a drawing using a template drawing

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box choose **Start from template**, then click the **Next** button.

3 Do one of the following:

- Choose a template drawing in the **Select a template list**.

  By default this is the content of the **Template** folder.

- Click the **Browse...** button to select a template drawing.

  You can choose between ***.dwg** and ***.dwt** file types.

5.1.8 Creating a drawing using a wizard

1 Launch the NEWWIZ command.

2 On the **Create New Drawing** dialog box, choose **Use a Wizard**, then click the **Next** button.

3 Choose between either **Imperial** or **Metric** default settings, then click the **Next** button.

4 Select the **Unit of measurement** and the display **Precision**, then click the **Next** button.

5 Select the **Angle of measurement** and the display **Precision**, then click the **Next** button.

6 Choose between **Color Dependent** plot styles (CTB) or **Named** plot styles (STB), then click the **Next** button.

7 Select the default **Color** and **Linetype**.

8 Define the **Grid**, **Snap**, **UCS icon** and **Marker Blips** settings.

9 Click the **Finish** button to create the drawing.

(option) Choose **Save As..** in the **File** menu to save the drawing as a new template in the **Templates** folder.

5.2 Working in the Drawing Explorer dialog box

5.2.1 Adding a drawing folder

1 In the **Drawings** > **Folders** sub-window of the **Drawing Explorer** main window, click the **Add local folder...** button.

The **Browse For Folder** dialog box appears.

2 Do one of the following:

- To add an existing folder: select the folder you want to add.
- To create a new folder: click the **Make New folder** button.

3 Click the **OK** button.

The folder is added.

Nota:

- Drawings in the subfolders of the selected folder can be opened too.
- The parent folder(s) of the selected folder are greyed out in the drawing folder tree. Drawings in such folders cannot be opened.

5.2.2 Opening a drawing

To open a drawing in one of your favorite folders:

1 In the **Drawings** > **Folders** sub-window, open the folder of the drawing.

2 Select a drawing.

3 A preview is shown in the **Preview** sub-window.

4 Do one of the following:
   - Double-click the drawing.
   - Right-click and choose **Open** in the context menu.

5.2.3 Logging on to Bricsys 24/7

1 In the **Drawings** pane, select **Folders**.

2 Click **Logon to Bricsys 24/7**.

3 Fill out your username and password.

4 When logged in, all of your projects on Bricsys 24/7 are listed.

5.2.4 Opening a drawing from a Bricsys 24/7 project

1 Expand the project, then select the drawing folder.

2 Select the drawing.

3 A preview is shown in the **Preview** sub-window.

4 Right-click the drawing, then choose **Download** in the context menu.

   A **Save File As** dialog box appears.

5 Select a folder and optionally rename the drawing, then click the **Save** button.

   The drawing is downloaded and opened in the current BricsCAD session.

5.2.5 Copying definitions between drawings

1 Open both the source and the target drawing.

2 Launch the **Drawing Explorer** dialog box.

3 In the source drawing, select the category you want to copy items from, e.g. **Blocks**.

4 (option) Adjust the view option of the **Details** pane, e.g. **Icon View** button ( ).

5 Select the definition(s) you want to copy, e.g. a series of blocks.

6 Press and hold the Ctrl key to select multiple items.

7 Do one of the following:
   - Press and hold the left mouse button to drag the selection set to the **Open Drawings** section of the Drawing Explorer.

     Release the left mouse button when on the target drawing name.

     The selected items are copied to the target drawing.
   - Right-click and choose **Copy** in the context menu.

     Select the same definition category in the target drawing, then right-click and choose **Paste** in the context menu.

Nota:

- You will be prompted to overwrite items of the same name.
- Drag-and-drop is not supported for some definition categories, e.g. MultiLeader Styles, Visual styles, Render Presets and Page Setups.

5.3 Drawing properties

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored within a drawing.

The **Drawing Properties** dialog box includes:

- General information: drawing data
- Summary: editable default properties
- Statistics: history data
- Sheetset: info about the associated sheetset
- User defined custom properties

Drawing properties can be referenced in fields.

5.3.1 Opening the Drawing Properties dialog

Do one of the following:

- Choose **Drawing Properties...** in the **File** menu.
- Execute the DWGPROPS command.

5.3.2 Defining default properties

1 On the **Drawing Properties** dialog box, click the **Summary** tab page.

2 Fill out the property fields.

3 Click the **OK** button.

5.3.3 Creating custom properties

1 On the **Drawing Properties** dialog box, click the **Custom** tab page.

2 Click the **Add** button.

The **Custom Property** dialog appears.

3 Type a name in the **Name** field.

The custom property name is listed under **Document** in the **Field** dialog.

4 Type a value in the **Value** field.

The value is displayed where the custom property is inserted as a field.

5.4 Drawing security options

The SECURITYOPTIONS command allows you to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

5.4.1 Setting the password

1 Do one of the following:

- Choose **Security Options...** in the **File | Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

The **Security options** dialog box opens.

2 Type the password in the **Enter Password** field.

3 Choose a crypt provider from the **Crypt Provider** list.

4 Set the **Key Length** to **40**, **48** or **56**.

5 Click the **OK** button or press Enter.

The **Security options** dialog box closes.

The **Password** dialog opens.

6 Type the password in the **Confirm Password** field.

7 Click the **OK** button or press Enter.

The password is defined.

5.4.2 Removing the password

1 Do one of the following:

- Choose **Security Options...** in the **File** | **Drawing Utilities** menu.
- Execute the SECURITYOPTIONS command.

2 The **Security options** dialog box opens.

3 Empty the **Enter Password** field.

4 Click the **OK** button or press Enter.

A message box appears to confirm that the password is removed.

5 Click the **OK** button or press Enter to close the message box.

5.5 Opening and saving drawings

The way BricsCAD opens and saves drawings is controlled through a series of system variables.

Name	Title	Description
	Display full path in title	Controls whether the full path of a drawing displays in the title bar, or the file name only.
	Drawings path	Specifies an additional folder in the file dialog for theOPENandSAVEAScommands. On the Windows platform: the fifth folder in the**Open File** dialog's **Places** bar.
-	First, Second, Third and Fourth folder	Sets the folders 1 through 4 in the**Places**bar of the non-standard**Open File**dialog (Windows platform only).
	Thumbnail preview image size	Sets the size of the thumbnail image in the Preview on file dialogs.

Name	Title	Description
	Index Control	Determines whether layer and/or spatial indexes are created and saved. • The layer index lists all entities in the drawing and what layer they are on. • The spatial index stores the location of all the entities in 3D space. The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded. The demand loading of Xrefs is controlled through the XLOADCTL system variable). Creating indexes slows down the saving of a drawing, but increases the speed of handling Xrefs. Therefore create indexes only for drawings that will be used as an external reference. 0 = No indexes.
	Xref load control	Controls demand loading of entities in external references. If demand loading is turned on you will experience a performance increase: • When clipping external references which are saved with a spatial index. Entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing. • When freezing layers in external references which are saved with a layer index, entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded.
	Multi-Threading flags	Bitcode 1024: Delayed XREF loading. Delays the loading of Xrefs to idle time.
	Incremental save backup	Controls the creation of a backup files (BAK). Improves the speed of incremental saves if switched off, especially for large drawings.

Name	Title	Description
	Save percent	This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes at the end of the drawing data base. A full save takes longer to be executed than a partial save.
	Recent File List	Specifies the number of files shown in the**Recent Files**flyout of the**File**menu and on the **Start** tab.
	Proxy graphics	Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.
	Proxy notice	Displays a notice when you open a drawing containing proxy entities (1) created by an application that is not present.
	Proxy show	Controls how proxy entities (1) display in a drawing.
	Proxy web search	Determines whether BricsCAD checks for object enablers.
	Save format	Sets the default dwg version to save a drawing.
	Save file path	Specifies the path and folder where automatic saves (.SV$) and temporary files are stored. TheSAVEFILEFOLDERcommand opens the SAVEFILEPATH location in the system file browser.
	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a.SV$extension in the folder defined by the SAVEFILEPATH system variable.
	Enable macros	Defines whether macros are enabled when loading a VBA-project.

Name	Title	Description
	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no auto-save file will be created for files that are viewed only.
	Ignore all but first bit of DBMOD(2) for close	If this user preference is switched on, you can view a file, then close it without getting a request to save the file.

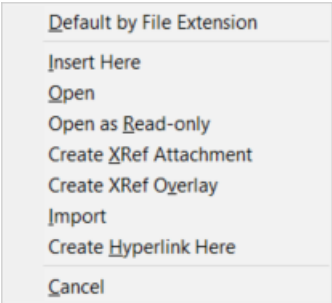
(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

(2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

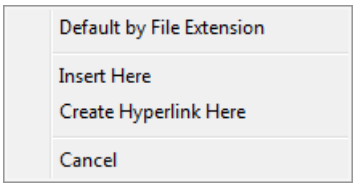
5.5.1 Using drag-and-drop to open a drawing

You can open a drawing by dragging it from a File dialog box onto the BricsCAD application window. Select the file, then hold down the left mouse button and move the cursor to the BricsCAD application window and release the left mouse button to load the drawing. When you drag a file holding down the right mouse button, a context menu displays:

When a drawing file is selected:



When another file type is selected:



Choose**Default by File Extension**to:

- Open a drawing file.
- Attach a PDF.
- Attach a raster image file.

- Insert a file as an OLE object (e.g. an Excel spreadsheet).

Choose**Insert Here**to:

- Insert a drawing file as a block in the current drawing.
- Attach a PDF.
- Attach a raster image file.
- Insert the file as an OLE object.

Choose**Open**to open a drawing file .

Choose **Open as Read-only** to open a drawing file and preserve the original from being overwritten. You can only save such drawing under a different name.

Choose**Create Xref**or **Create Xref Overlay** to attach a drawing file as an external reference to the current drawing.

- **Attachment**: if the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.
- **Overlay**: an overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.

Choose **Import** to import the geometry in the current drawing.

Choose**Create Hyperlink Here**to create an hyperlink to the selected file.

Nota:

- Drawing file extensions are: .DWG, .DXF or .DWT
- When multiple files are selected, the files are processed one after the other.

5.6 Getting drawing information

You can retrieve two types of information from a drawing:

- General information about the drawing.
- Information about the entities in the drawing.

5.6.1 General drawing information

The DWGPROPS command opens the **Drawing Properties** dialog box, showing the general information and user defined properties stored with a drawing.

The SETVAR command displays or changes the values of system variables (short for "set variables").

The STATUS command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing.

The TIME command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

5.6.2 Information about entities

The STATUS command reports the total number of entities in the drawing.

Using **Quick Select**, you can see the number of entities for each category.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The ID command reports the X, Y, Z coordinates of picked points.

The LIST command lists the properties of selected entities.

5.7 Comparing drawings

The DWGCOMPARE command compares a selected drawing with the current drawing during a comparison session.

The ENDCOMPARE command concludes a comparison session.

The KEEPME command allows you to merge the compared drawings visually.

5.7.1 Comparison system variables

Name	Title	Description
CMPCLRMISS	Compare missing entities display color	Sets the color for displaying entities which are missing in the other drawing. The default color is Red.
CMPCLRMOD1	Compare modified entities display color 1	Sets the color for displaying modified entities in the current drawing. The default color is index color 253.
CMPCLRMOD2	Compare modified entities display color 2	Sets the color for displaying modified entities in the other drawing. The default color is Yellow.
CMPCLRNEW	Compare new entities display color	Sets the color for displaying entities which are new in the other drawing. The default color is Green.
CMPDIFFLIMIT	Maximal number of output entities	Sets the limit for entities comparison. The comparison will be aborted after reaching the difference limit. Values between 1 and 10 000 000 are accepted.

5.7.2 Executing a comparison session

1 Launch the DWGCOMPARE command.

1 You are prompted: CMPDIFFLIMIT = 10000000. Select the file to compare with or [Limit]:

2 (option) Choose the **Limit** option to set a new value for entities comparison.

3 Press Enter to display a **File** dialog box.

4 Select the drawing to be compared with the current drawing, then click the **Open** button or double-click the drawing name.

The differences between the two drawings are displayed using the **Compare Display Colors**.

The **Compare** toolbar appears.

5 To check the **Diff Type** of an entity, do one of the following:
- If **Rollover Tips** (RT) are on, hover over the entity.
- Select the entity and check the **Diff type** field in the **Properties** panel.

6 Click the **Endcompare** tool button on the **Compare** toolbar to conclude the comparison session.

5.7.3 Analyzing the comparison in the Structure panel

1 Load the **Default.cst** configuration in the **Structure** panel.

2 Expand the **Comparison** node.

Four subnodes are shown: **Missing**, **Modified (Original)**, **New** and **Modified (Compared)**.

The number of entities in each subnode is indicated between brackets.

3 Do the following:
- Click a subnode to select the corresponding entities in the drawing.
- Expand a subnode, then click an entity to highlight the corresponding entity in the drawing.
- Press and hold the Ctrl key to select multiple entities.
- Right-click, then choose **Zoom** in the context menu to zoom onto the selected entity or entities.

Nota: When selecting entities in the **Structure** panel, entities are either selected or highlighted in the drawing, or tree selection is ignored. (See the **Structure** panel).

5.7.4 Merging the compared drawings

1 **Missing entities** only exist in the current drawing.
- Select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.
- Select the entities you don't want to keep, then click the **Delete** tool button.

2 **Modified entities** exist in both drawings.
- In the **Modified (Original)** node: select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.

  The corresponding entities in the **Modified (Compared)** node are removed.
- In the **Modified (Compared)** node: select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.

3 **New entities** exist in the compared drawing only.
- Select the entities you want to keep, then click the **Keep Me** tool button on the **Compare** toolbar.
- Select the entities you don't want to keep, then click the **Delete** tool button.

When the merge is complete, the **Comparison** node in the **Structure** panel disappears. A message appears in the Command line: No more differences to show. Session is closed.

5.8 Repairing a drawing

TheAUDIT command analyzes the integrity of the current drawing and optionally tries to fix errors.

TheRECOVER command repairs a damaged drawing, then opens the drawing.

5.8.1 Analyzing the integrity of the current drawing

1 Launch the AUDIT command.

Prompts you:Fix any errors detected? Yes/<No>

2 Do one of the following:

- Choose**Yes**in the context menu or type**Y**, then press Enter to analyze the integrity of the current drawing and fix the errors detected.
- Choose**No**in the context menu or type**N**, then press Enter to analyze the integrity of the current drawing.

The drawing is analyzed.

3 (option) Press F2 to open the**Prompt History**window in order to read the audit report:

- # objects audited
- Total errors found during audit #, fixed #.

Nota:

- Set theAUDITCTLsystem variable to 1, to letAUDIT create an ASCII file describing the detected problems and the action taken. This report, with the .adt file extension, is placed in the same directory as the current drawing .
- Use theRECOVER command to repair the errors thatAUDIT cannot fix.

5.8.2 Repairing a drawing

1 Do one of the following:

- Choose**Recover**in the**File** > **Drawing Utilities**menu.
- Launch the RECOVER command.

The**Open Drawing**dialog window displays.

2 Select the drawing, then press the**Open**button on the**Open Drawing**dialog window.

The drawing is processed and - if the repair succeeds - opened.

3 (option) Press F2 to open the**Prompt History**window in order to read the recover report.

5.9 Projects

Projects are especially useful to manage Xrefs and images when exchanging drawings. If external references (Xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images.

Projects are saved in the registry and contain one or more search paths.

5.9.1 Accessing the Project settings

1 Launch the SETTINGS command.

1 The **Settings** dialog box opens.

2 Go to **Program Options** > **Files** > **Projects**.

5.9.2 Creating projects

1 Edit the PROJECTSEARCHPATHS system variable.

2 Click the **Browse** button at the right hand side of the **Project search paths** settings field.

The **Project Settings** dialog box appears.

3 To create the first project:
- Click the **Add Project or Path** button on the **Project Settings** dialog.

  A new project is added.
- Type a name to replace the **NewProject1** default name.

4 To create additional projects:
- Click **Projects** (on top of the projects tree) or select an existing project.
- Click the **Add Project or Path** button.

  A new project is added.
- Type a name to replace the **NewProject1** default name.

5.9.3 Adding search paths to a project

1 Edit the PROJECTSEARCHPATHS system variable.

2 Select the project in the **Projects** tree.

3 Click the **Add Project or Path** button.

A path is added to the project.

4 Type a name for the new path.

5 Click the **Browse** button.

The **Choose a Path** dialog box appears.

6 Browse to a folder, then click the **Select Folder** button.

7 Repeat the previous steps to add more search paths to a project.

5.9.4 Changing the order of the search paths

1 Edit the PROJECTSEARCHPATHS system variable.

2 Expand the project in the **Projects** tree.

3 Select a search path, then click the up or down buttons to change the order of the search paths.

5.9.5 Assigning a project to the current drawing

1 Edit the PROJECTNAME system variable.

2 Type a project name in the settings field.

6. Disegnare con precisione

6.1 Orthogonal mode

When creating or manipulating entities, Orthogonal mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the currentcoordinate system.

Lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. When drawing, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor.

Nota:

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the Orthogonal Mode axes using the SNAPANG system variable.
- Holding down the Shift key reverses the Orthogonal Mode setting.
- Orthogonal Mode is overruled by entity snaps.
- Turning on Ortho Mode automatically disables Polar Tracking and vice versa.

6.1.1 Toggling Orthogonal mode

Do one of the following:

- Click theORTHOfield in theStatus Bar.
- Press theF8function key.
- Hold down the Shift key.
- Launch the ORTHO command.

6.2 Unit settings

The UNITS command sets units of linear and angular measurements through the **Settings** dialog box. The -UNITS command sets units of linear and angular measurements in the Command line.

The INSUNITS (Insertion Units) system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

Any length, area or volume property can be displayed using the unit(s) preferred by the user.

The unit defined by INSUNITS is used as the base unit for any requested unit conversions. If INSUNITS = 0 (undefined), the PROPUNITS mechanism is disabled because the base drawing unit is undefined.

Nota:

- When LUNITS = 3 (Engineering) or 4 (Architectural), property values will be formatted in imperial notation.
- Automatic unit conversion during length input can be used when bitcode 1 of PROPUNITS (Format length properties) is set. E.g. when INSUNITS is set to "mm", entering "2m" in a length property field will result in entering [2000.0000 mm]. Entering "1ft" will give [304.8000 mm].

6.2.1 System variables

Name	Title	Description
INSUNITS	Insertion Units	Defines the value of 1 Drawing Unit (DU), which is used for automatic scaling when inserting or attaching blocks, images or Xrefs. When both INSUNITS and PROPUNITS are ON; **Length**, **Area**, **Volume** and/or **Inertia** properties will be formatted with a dedicated unit.
MEASUREMENT	Measurement	Sets drawing units as **Imperial** or **Metric**. The MEASUREMENT system variable controls whether ANSI or ISO hatch pattern and linetype files are used.
PROPUNITS	Property Units	Defines which property values are automatically formatted with a dedicated unit when INSUNITS is not zero. The value is stored as a bitcode, using the sum of the values of all selected options.
LUNITS	Linear Units Type	Sets the linear unit type: **Scientific**, **Decimal**, **Engineering**, **Architectural** or **Fractional**.
LUPREC	Linear Unit Precision	Sets the number of decimal places displayed for linear units.
AREAPREC	Area Precision	Sets the number of decimal places displayed for areas when the **Area** bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
VOLUMEPREC	Volume Precision	Sets the number of decimal places displayed for volumes when the **Volume** bit of PROPUNITS is ON. If set negative, the value of LUPREC is used.
LENGTHUNITS	Length Units	Defines a list of units used for displaying lengths when the **Length** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
AREAUNITS	Area Units	Defines a list of units used for displaying areas when the **Area** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.

Name	Title	Description
VOLUMEUNITS	Volume Units	Defines a list of units used for displaying volumes when the **Volume** bit of PROPUNITS is ON. If empty, all lengths are displayed in the current drawing unit, as defined by INSUNITS. If multiple units are selected, the unit that fits best is used.
UNITMODE	Unit Mode	Specifies how Imperial units are displayed.

6.2.2 Entering angles

The AUNITS system variable defines the angular unit type.

The ANGDIR system variable sets the positive angle direction, counter clockwise (default) or clockwise. Angles are measured with respect to direction of the positive X-axis of the current coordinate system.

You can enter angles in any of 5 formats:

- Decimal degrees : 37.456 (AUNITS = 0)
- Degrees / Minutes / Seconds : 37°27'22" or 37d27'22" (AUNITS = 1)
- Gradians : 41.6178g (AUNITS = 2)
- Radians : 0.6537r (AUNITS = 3)
- Surveyor's Units : N52°32'38"E or N52d32'38"E (AUNITS = 4)
- Surveyor's units are measured from the North (N) or the South (S) to the East (E) or to the West (W).

6.3 Coordinate reference system

A Coordinate System is a set of mathematical rules for specifying how coordinates are to be assigned to points.

The coordinate system is unrelated to the Earth. A Coordinate Reference System (CRS) is a coordinate system related to the Earth through a Datum. A Geodetic Datum is a model of the Earth that specifies a reference surface (Ellipsoid or Spheroid).

A projected coordinate reference system is the result of the application of a map projection to a geographic coordinate reference system. A map projection is a type of coordinate conversion. It uses an identified method with specific formulas and a set of parameters specific to that coordinate conversion method.

Coordinates may be changed from one CRS to another through the coordinate operation. Two types of coordinate operation may be distinguished:

- **coordinate conversion**, where no change of datum is involved and the parameters are chosen and thus error free.
- **coordinate transformation**, where the target CRS is based on a different datum to the source CRS. Transformation parameters are empirically determined and thus subject to measurement errors. (Geodetic, Seven parameters, Molodenksy...)

Geographic datasets are defined in **geodatabase.xml**that is placed in *<User folder>\AppData\Roaming\Bricsys\BricsCAD\<RELEASE>\en_US\Support for Windows*. The content of this file is composed of the 5 sections:

1 CoordinateReferenceSystems

2 Datums

3 Ellipsoids

4 Transformations

5 ProjectionCode and Methods

6.3.1 CoordinateReferenceSystems

This section is the main XML node where all Coordinate Reference Systems are defined. It has many sub-nodes called**CRS**, each XML node must define a unique projected coordinate reference system and the necessary projected parameters.

The supported CRS projection types are described in the**ProjectionCode**and**Methods**sections.

CRSnode attributes:

Follow Table 1 of OGP Geomatics Guidance Note No 7, part 2

Attribute name	Description	Units
epsg	Unique EPSG database id.	Integer
name	Human-readable name of CRS, Example: "WGS 84 -- WGS84 - World Geodetic System 1984, used in GPS".	String
codeSpace	Owner of CRS. It is option.	String
x,y	Orientation of projected direction. Possible values: • "Westing" • "Easting" • "Southing" • "Northing" Directions MUST be defined.	String
xAxis, yAxis	Orientation of geographic direction. Possible values: • "Latitude" • "Longitude"	String
Greenwich	Greenwich Meridian relation.	Degree

Attribute name	Description	Units
proj	Projection method of CRS. Possible values: • "LL", "LatLon", "LonLat" is a Geodetic Latitude-Longitude. • "MercA", "Mercator_1SP" is a Mercator (variant A). • "MercB", "Mercator_2SP" is a Mercator (variant B). • "MercC" is Mercator (variant C). • "MercSp" is a Mercator (Spherical). • "MercPv" is a Popular Visualisation Pseudo Mercator. • "TMerc" is a Transverse Mercator. • "TMercSO" is a Transverse Mercator South Orientated. • "LCC_1SP" is a Lambert Conic Conformal 1SP. • "LCC_2SP" is a Lambert Conic Conformal 2SP. • "LCC_2SP_BE" is a Lambert Conic Conformal 2SP Belgium.	String
AngleSG	Angle from Rectified to Skew Grid.	Degree
Azimuth	Azimuth of initial line.	String
FE	Easting at false origin, False easting.	Float-Point value in CRS units
EC	Easting at projection center.	Float-Point value in CRS units
FN	Northing at false origin, False northing.	Float-Point value in CRS units
FC	Northing at projection center.	Float-Point value in CRS units
LonI	The longitude of the western limit of the first zone of a Transverse Mercator zoned grid system.	Degree
Lat	Latitude of natural origin, Latitude of standard parallel. It depends on Projection method.	Degree
Lat1	Latitude of 1st standard parallel.	Degree

Attribute name	Description	Units
Lat2	Latitude of 2nd standard parallel.	Degree
LatF	Latitude of false origin.	Degree
LatC	Latitude of projection center.	Degree
Lon	Longitude of natural origin, Longitude of origin.	Degree
LonF	Longitude of false origin.	Degree
LonC	Longitude of projection center.	Degree
SF	Scale factor at natural origin.	Float-Point value, unity
SFIL	Scale factor on initial line.	Float-Point value, unity
SFPSP	Scale factor on pseudo standard parallel.	Float-Point value, unity
W	Zone width in longitude.	Degree
zone	Zoned Grid System.	Degree
units	Units of conversion result from geographic coordinates to projected. Example: "Meter", unit "Degree" means no conversion, Geographic Coordinate System.	String

Example:common attributes for all map projections based on EPSG:31468.

```
<CRS epsg="31468" codeSpace="OGP" name="DHDN / 3-degree Gauss-Kruger zone 4"
 y="Easting" x="Northing" Greenwich="0" proj="TMerc" Lon="12" Lat="0" SF="1"
 FE="4500000" FN="0" zone="4" units="Meter">
```

For each node**CRS**there should be at least one sub-node**Datum**that refers via**epsg**or**alias**attributes to sub-node**Datum**in the**Datums**section. The sub-node**Datum**MUST have the**id**attribute of a unique CRS name. At least one of the epsg code and the alias of Datum must be valid.

Attribute name	Description	Units
epsg	Unique EPSG database id. Example: "4326".	Integer
alias	Unique Datum name. Example: "WGS 84".	String

Attribute name	Description	Units
id	Unique CRS name: combination of Projected method and Datum. Example: "WORLD-MERCATOR". It related to CS-MAP, AutoCAD names.	String
pjcode	Indicate code for the coordinate reference system projection method types. Corresponded to AutoCAD projection codes. For example, 3 - Transverse Mercator, 44 - UTM, 45 - Transverse Mercator of Snyder formulation and etc.	Integer

Example:there are unique CRS definitions according to Datums insideCRSnode:

```
<Datum epsg="6314" alias="DHDN/3" id="DHDN/3.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN/2" id="DHDN/2.Gauss3d-4" pjcode="3" />
<Datum epsg="6314" alias="DHDN" id="DHDN.Gauss3d-4" pjcode="3" />
```

6.3.2 Datums

This section is a main node where all Datums are stored. Datum is a combination of the Earth model (Ellipsoid or Spheroid) and the transformation method to model WGS84. A**Datum**node specifies common parameters and contains at least one inner sub-nodes**Datum**exactly one sub-node**Ellipsoid**. Each inner sub-node**Datum**should define a unique name within the**id**attribute and may specify transformation parameters to WGS84 model via sub-node**Transformation**that refers via**epsg**or**alias**attributes to a sub-node**Transformation**in the**Transformations**section.

Attribute name	Description	Units
alias	Unique id. Example: "DHDN/3". It is related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "6314".	Integer
name	Human-readable name of Datum. Example: "Deutsches Hauptdreiecksnetz".	String
codeSpace	Owner of CRS. It is optional.	String

Example:

```
<Datums>
<Datum epsg="6314" codeSpace="OGP" name="Deutsches Hauptdreiecksnetz ">
<Datum id="DHDN/3">
<Transformation epsg="1777" alias="DHDN/3_to_WGS84" />
</Datum>
```

```
<Datum id="DHDN/2" name="Deprecated - Replaced by DHDN/3">
<Transformation epsg="1777" alias="DHDN/2_to_WGS84" />
</Datum>
<Datum id="DHDN" name="Deprecated - Replaced by DHDN/2">
<Transformation epsg="1673" alias="DHDN_to_WGS84" />
</Datum>
<Ellipsoid epsg="7004" alias="BESSEL" />
</Datum>
â€¦
</Datums>
```

6.3.3 Ellipsoids

The Ellipsoids section is the main section where models of the Earth are defined. Each**Ellipsoid**node should define an identification by a unique name (**alias**) and a semi-major axis (**a**) and one of the following attributes at least: flattering (**f**), semi-minor axis (**b**) or eccentricity (**e**).

Attribute name	Description	Units
alias	Unique id. Example: "WGS84". It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "7030".	Integer
name	Human-readable name of Datum, Example: "WGS 84".	String
codeSpace	Owner of CRS. It is optional.	String
a	Length of the semi-major axis of the ellipsoid, the radius of the equator.	Float-Point value, strongly in Meter
b	Length of the semi-minor axis of the ellipsoid, the distance along the ellipsoid axis between equator and pole.	Float-Point value, strongly in Meter
f	Flattening	Float-Point value, unity
e	Eccentricity	Float-Point value, unity

Example:

```
<Ellipsoid epsg="7008" alias="CLRK66" name="Clarke 1866, Benoit Ratio"
 a="6378206.4000000004" b="6356583.7999999998" f="294.9786982139"
 e="0.0822718542" />
```

6.3.4 Transformations

This section is a main node where all transformation methods are defined. Supported Transformation methods are described in the **ProjectionCode** and **Methods** sections.

Attribute name	Description	Units
alias	Unique id. Example: "DHDN_to_WGS84". It related to CS-MAP, AutoCAD names.	String
epsg	Unique EPSG database id. Example: "1673".	Integer
codeSpace	Owner of Transformation. It is optional.	String
source	Source Datum.	String
target	Target Datum.	String
use	Transformation method. Supported transformations using geocentric methods: • Geocentric translations • Four/Six/Seven Parameter Transformation • Similarity Transformation • Position Vector transformation • Coordinate Frame rotation	String
method	Methods of building of rotation matrix, if applicable, "PVT" is Position Vector Transformation, "CFR" is Coordinate Frame Rotation.	String
tX	X-axis translation.	Meter
tY	Y-axis translation.	Meter
tZ	Z-axis translation.	Meter
rX	X-axis rotation.	Degree
rY	Y-axis rotation.	Degree
rZ	Z-axis rotation.	Degree
dS	Scale difference.	Unity

Attribute name	Description	Units
xp	Coordinate 1 of evaluation point.	Meter
yp	Coordinate 2 of evaluation point.	Meter
zp	Coordinate 3 of evaluation point.	Meter
dtX	Rate of change of X-axis translation.	Unity
dtY	Rate of change of Y-axis translation.	Unity
dtZ	Rate of change of Z-axis translation.	Unity
drX	Rate of change of X-axis rotation.	Unity
drY	Rate of change of Y-axis rotation.	Unity
drZ	Rate of change of Z-axis rotation.	Unity
ddS	Rate of change of scale difference.	Unity
t0	The reference epoch for time-dependent parameters.	Float point value
fallback	Alias of transformation from target datum to WGS84 datum.	String

Example:

```
<Transformation epsg="1679" alias="Pulkovo42/2_to_WGS84" src="Pulkovo42/2"
 trd="WGS84" use="Param7" method="CFR" tx="-40.595" ty="-18.55" tz="-69.339"
 ds="-4.299" rx="-2.508" ry="-1.832" rz="2.611" accuracy="9" />
```

6.3.5 ProjectionCode and Methods

These sections describe the**ProjectionCode**and**Methods**that are mapped to AutoCAD definitions of projections and transformations types. They are used to store the definitions in an XML file of coordinate reference system definition inside the**AcDbGeoData**object that represents a geographic location.

6.4 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.

Nota:

- 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.

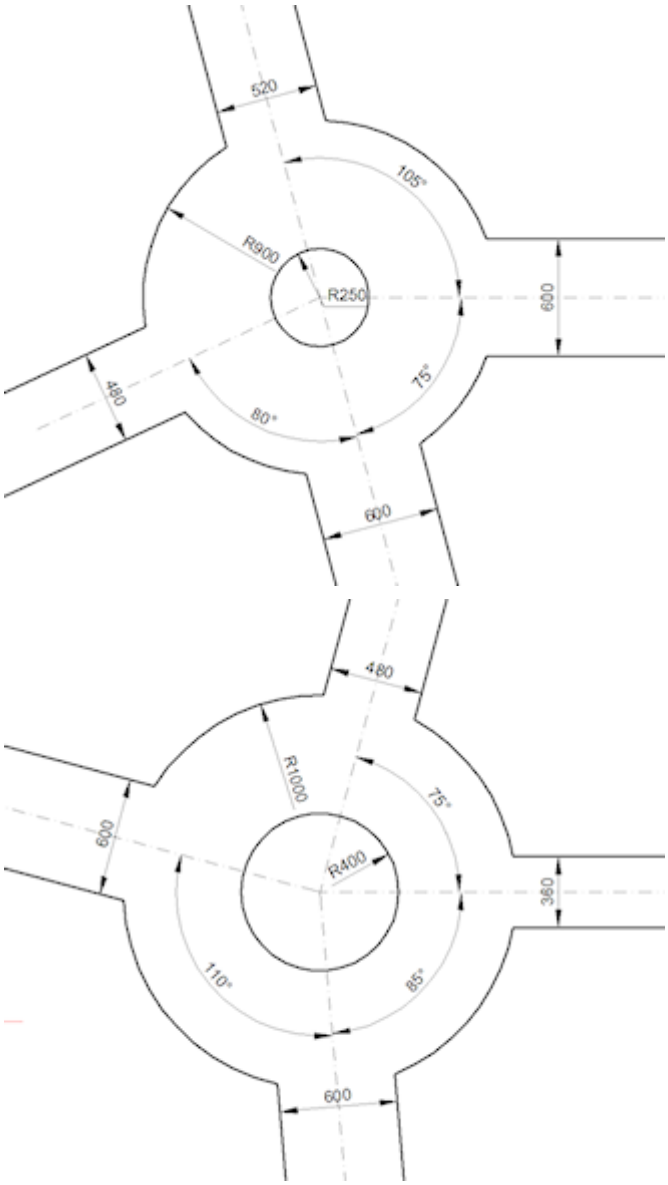


Figura 1: Two versions of the same fully constrained drawing representing a roundabout

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.

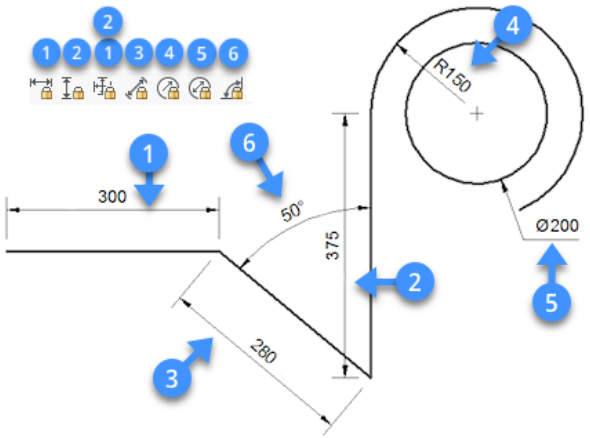
Command	Description
	Applies geometric relationships between entities and/or valid constraint points on entities.
	Constrains points and entities at a fixed position.
	Applies a coincident geometrical constraint to two points or constraints a point to an entity.
	Constrains the center points of circles, 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).



Nota: Dimensional constraints are created using the current dimension style.

6.5.1 Editing dimensional constraints

Dimensional constraints can be edited:

- In the Properties panel.
- In theMechanical browser panel.
- In the Parameters and constraints panel.
- Using theDDEDITcommand.

Editing dimensional constraints in the Properties panel

1 Select the dimensional constraint.

The properties of the entity display in the Properties panel.

2 Type a new value orexpressionin the**Expression**field, then press Enter.

The readout of the**Value**field changes accordingly.

3 (option) Edit the**Name**field.

4 (option) Edit the**Description**field.

5 (option) Edit the**Text rotation**field.

6 Press the Esc key to unselect.

Nota: Select multiple dimensional constraints simultaneously to set their value equally.

Editing dimensional constraints in the Mechanical browser panel

1 If not already open, open theMechanical Browser panel.

2 Select a name of the dimensional constraint in the Parameters list.

The properties the selected parameter display in bottom part of theMechanical browser panel.

3 (option) Select the**Name**field, then type a new name.

4 (option) Select the**Expression**field, then type a new value or expression.

5 (option) Select the**Description**field, then type a new description.

6 (option) Set the**Exposed**field, which specifies whether the parameter is exposed when the drawing is inserted in another drawing usingthe BMINSERT command.

The options are:

- **Auto** : the parameter is exposed only if it does not depend on other parameters.
- **ON** = the parameter is always exposed.
- **OFF** = the parameter is never exposed.

7 (option) Set the**Units**property: linear, planar or volume.

Select the field, then click the down-arrow button and choose an option.

Using the DDEDIT command

1 Do one of the following:
- Double click the dimensional constraint.
- Launch the DDEDIT command.

You are prompted: _DDEDIT

The **Edit Text** dialog box displays.

2 Type a new value in the**Text**field, then press Enter or click the**OK** button.

3 (option) Click another dimensional constraint and repeat the previous step.

4 Do one of the following:
- Keep editing more dimensional constraints.
- Press Enter or right click to stop.

All edited dimensional constraints are updated simultaneously.

6.5.2 Setting the visibility of dimensional constraints

Dimensional constraints are hidden when a drawing is closed and reopened.

Tools to control the display of dimensional constraints are located on:

- The **Dimensional Constraint** toolbar.
- the **Parametric ribbon** tab.
- the **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the DCDISPLAY command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.
- Hide All: hide all dimensional constraints in the drawing.

Nota:

- The DYNCONSTRAINTMODE system variable controls the visibility of hidden dimensional constraints when selecting entities:
  - ON: Display dimensional constraints when entities are selected.
  - OFF: Dimensional constraints remain hidden when entities are selected.
- When hovering over an entity with a dimensional constraint, the blue constraint glyph ( ) displays if the SELECTIONPREVIEW system 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:

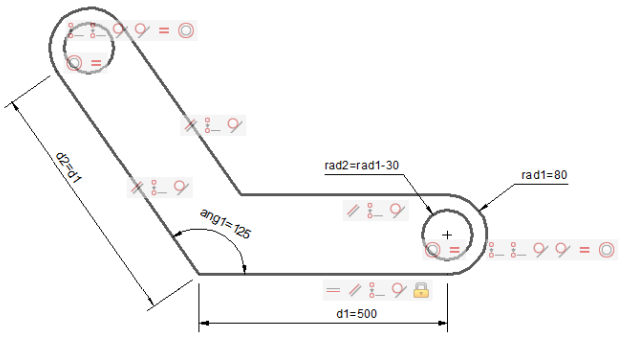
- distance: d1, d2, d3, ...
- radius: rad1, rad2, rad3, ...
- diameter: dia1, dia2, dia3, ..
- angular: ang1, ang2, ang3, ..

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 theMechanical Browser panel.

A context menu displays.

2 Select**Add new parameter**in the context menu.

A new parameter is created.

3 Edit the properties of the new parameter.

See the **Editing dimensional constraints in the Mechanical browser panel** procedure above.

Nota:

- The constants**Pi=3.14...**and**e=2.72...**can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- TheCLEANUNUSEDVARIABLEScommand purges parameters that are not used in constraint expressions or linked to dimensions.

6.6 Using geometric constraints

Geometric constraints:

- Specify a geometric relation between 2 entities (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal).
- Specify a fixed angle (horizontal, vertical).
- Specify a fixed location (fix).

When a geometric constrained is applied to an entity:

- The position of the entity is adjusted according to the applied constraint.
- An icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined in a constraint bar.

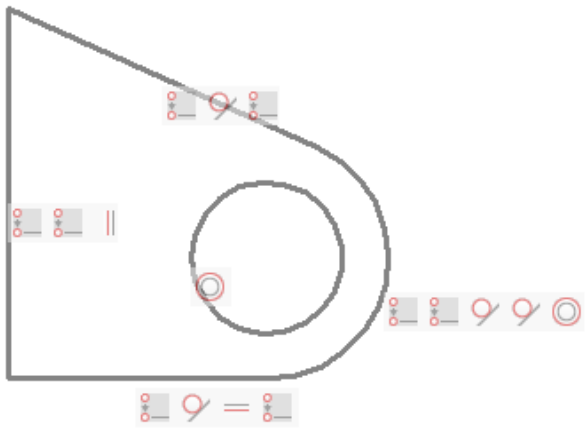
6.6.1 Creating geometric constraints

Tools to create dimensional constraints are located in:

- The **2D Constraints** toolbar.
- The **Parametric ribbon** tab.
- The **2D Constraints** fly-out menu of the **Parametric** menu.

In the image below:

- The endpoints of the three lines and the arc are joined by coincident constraints.
- The midpoint of the circle and the arc are made concentric.
- Two lines are forced to be tangent to the arc.
- One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system).



6.6.2 Displaying constraint bars

Constraint bars are hidden when a drawing is closed and reopened.

Tools to control the display of constraint bars are located on:

- The **Geometric Constraint** toolbar.
- The **Parametric** ribbon tab.
- The **Show/Hide 2D Constraints** fly-out menu of the **Parametric** menu.

These tools execute the CONSTRAINTBAR command using a specific option.

The options are:

- Show/Hide: prompts you to select the entities.
- Show All: show all dimensional constraints in the drawing.
- Hide All hide all dimensional constraints in the drawing.

Nota:

- The value of theCONSTRAINTBARDISPLAYsystem variable controls the visibility of geometric constraints:
  - 1: Display constraint bars when geometrical constraints are added.
  - 2: Display hidden dimensional constraints when the constrained entities are selected.
- When hovering over an entity with a geometric constraint, the blue constraint glyph ( ) displays if theSELECTIONPREVIEWsystem variable is set to 1 or 3 or when the Quad is active.

6.6.3 Controlling the position of a constraint bar

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the**Reset**option of theCONSTRAINTBARcommand restores the default position of the constraint bar.

Relocating a constraint bar

1 Place the cursor on the constraint bar.

2 Press and hold the left mouse button to move the constraint bar.

3 Release the left mouse button at the desired location.

Restoring the default position of constraint bars

1 Do one of the following:

- Click the**Show/Hide Geometric Constraint**tool.
- Launch the CONSTRAINTBAR command.

Prompts you:Select Entities.

2 Select the entities, then right click or press Enter.

BricsCAD® reports the number of selected entities.

Prompts you: Select option for constraints [Show/Hide/Reset] <Show>:

3 Do one of the following:

- Choose**Reset**in the context menu.
- Type**R**in the Command line, then press Enter.

6.6.4 Working with constraint bars

Controlling a constraint

1 Move the cursor over a constraint icon.

2 A tooltip displays, indicating the constraint type.

3 The associated entity (or entities) highlights.

4 The corresponding icon on the constraint bar of the associated entity highlights.

Deleting a constraint

1 Move the cursor over the constraint icon in the constraint bar.

2 Right click, then click**Delete**.

3 The constraint is deleted and the icon is removed from the constraint bar and from the constraint bar of the associated entity.

Hiding the constraint bar of an entity

Move the cursor of the constraint bar, then click the**Close**(x) button.

6.6.5 Deleting constraints

To delete all constraints from a selection set:

1 Do one of the following:

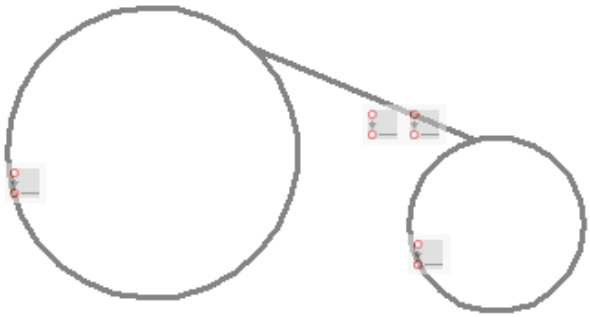
- Click the**Delete 2D Constraints**tool.
- Launch the DELCONSTRAINT command.

You are prompted: Select entities to delete all constraints:

2 Selectthe entities you want to delete the constraints from.

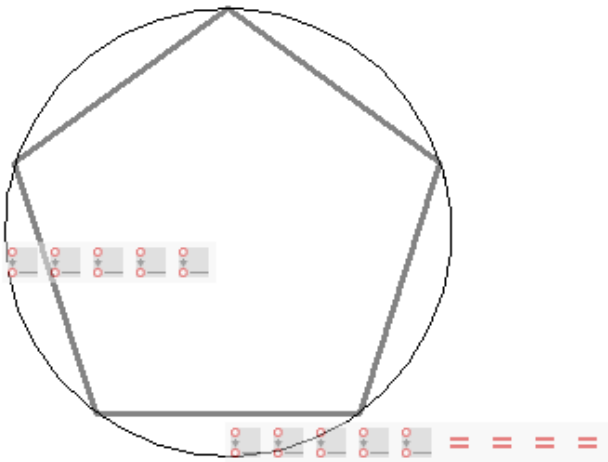
3 Right click to stop selecting entities and delete all constraints from the selection set.

6.6.6 Some examples

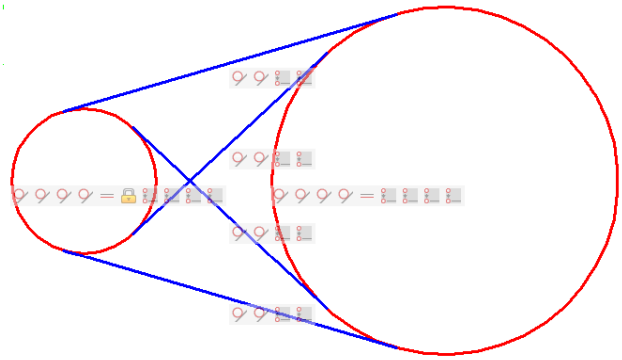


Coincident constraints between the endpoint of a line and two circles.

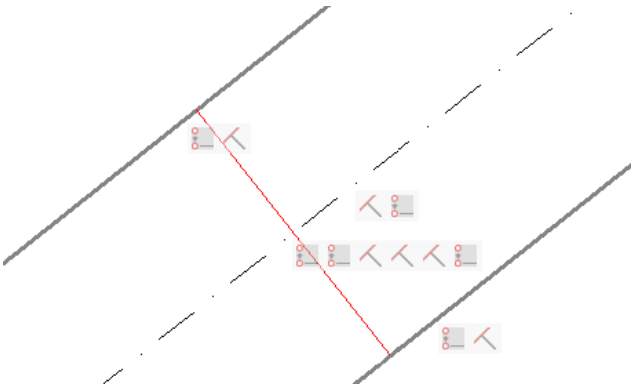
If the endpoints or center points of entities already coincide, the**Autoconstraint**option of theGCOINCIDENTcommand automatically applies coincident constraints to such points.



By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



Coincident constraints are used to:

- Force the endpoints of the red line to lie on the bold lines.
- Force the midpoint of the red line to lie on the dash-dot line.

The bold lines and the dash-dot line have a perpendicular constraint with the red line. As a result the dash-dot line always lies centered between the two bold lines.

6.7 Dynamic dimensions

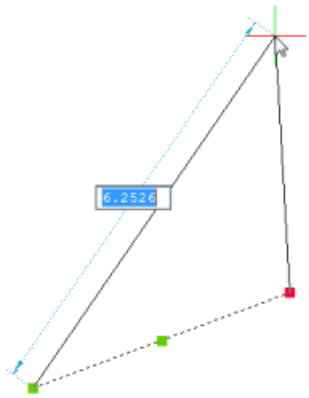
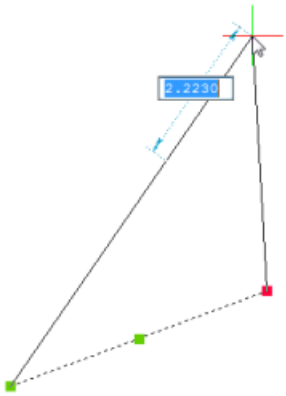
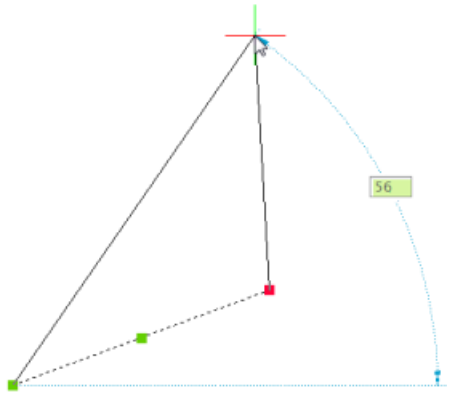
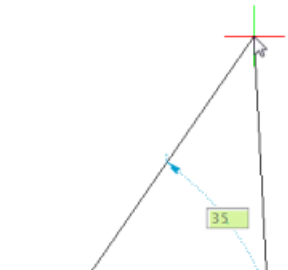
Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such, dynamic dimensions help you to focus on the drawing area.

The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

6.7.1 Defining the Dynamic Dimensions Settings

Right click the**DYN**field in theStatus Bar, then choose**Settings**in the context menu. The**Dynamic input**settings in the**Settings**dialog display.

Name	Title	Description
DYNMODE	Dynamic input mode	Controls whether and when dynamic dimensions display: • Editable dynamic dimensions: when creating or manipulating entities. • Tracking dynamic dimensions: when using entity snap tracking.

Name	Title	Description
DYNDIGRIP	Show dynamic dimensions	Defines which dynamic dimensions display when creating or grip editing entities, depending on the value of the DYNDIVIS system variable Resulting length: total length of the line or polyline segment  Extended length: incremental length of the line  Absolute angle: angle relative to the X-axis of the current UCS  Relative angle: angle relative to the original angle of the linear entity 

Name	Title	Description
DYNDIVIS	Dynamic dimension visibility	Defines the number of dynamic dimensions that display when creating or grip editing entities.
DYNPICOORDS	Default mode for dynamic coordinates input	Specifies the default mode in which coordinates are entered during dynamic input: Relative or Absolute.

Nota:

- The appearance of dynamic dimensions (color, linetype, position and transparencey) is controlled by a set of user preferences: DynDimColorHover, DynDimColorHot, DynDimDistance, DynDimLinetype, DynImputTransparence.
- If bitcode 16 (= format dynamic dimensions) of thePROPUNITSsystem variable is set, dynamic dimensions are formatted using a dedicated unit.
- Automatic unit conversion during length input can be used when bitcode 1 ofPROPUNITS(Format length properties) is set. For example, whenINSUNITSis set to **mm**, entering **2m** in a length property field will result in entering [2000.0000 mm]. Entering **1ft** gives [304.8000 mm]. If no unit is specified in the entry field, the unit as specified byINSUNITSis used.

6.7.2 Using dynamic dimensions when creating entities

1 Launch the command to create a 2D entity (e.g. LINE andDYNDIVIS= 1 or 2).

2 Specify the first point, then move the cursor to display the dynamic dimensions.

The**Length**dynamic dimension highlights.

3 Do one of the following:
   - Type the desired length of the line, then press the TAB key to jump to the**Angle**dynamic dimension.

     The**Length**dynamic dimension is locked and displays in red and the**Angle**dynamic dimension is selected.
   - Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
   - Press he TAB key to jump to the**Angle**dynamic dimension.

4 Do one of the following:
   - Type the desired angle, then press the Enter key to create the line.
   - Type the desired angle, then press the TAB key to return to the**Length**dynamic dimension.
   - Press the Enter key to create the line at the angle defined by the cursor position.
   - Press the TAB key to return to the**Length**dynamic dimension.

Nota:

- IfDYNDIVIS= 0, only one dynamic dimension displays. Press the TAB key to display the next field.
- If you type a value in the**Length**field, and add a comma (,) the content of the**Length**field is copied to the Command line and the @-character is placed in front automatically, which allows you to specify the next point usingrelative coordinateswith respect to the previous point.

6.7.3 Using dynamic dimensions to grip-edit entities

1 Select the entity or entities.

All grips on the selected entities display.

2 Click the grip you want to edit, then move the cursor.

Depending on the value of theDYNDIVISsystem variable one, two or all dynamic dimensions display simultaneously .

The first dynamic dimension highlights.

3 Do one of the following:

- Type the desired value for the currently highlighted dynamic dimension, then press the TAB key to jump to the next dynamic dimension.

  The dynamic dimension is locked and displays in red.
- Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
- Press the TAB key to jump to the next dynamic dimension.

4 Repeat step 3 or press Enter to stop.

6.7.4 Using dynamic dimensions to measure entities

The following procedure applies to the 2DWireframe visual style only.

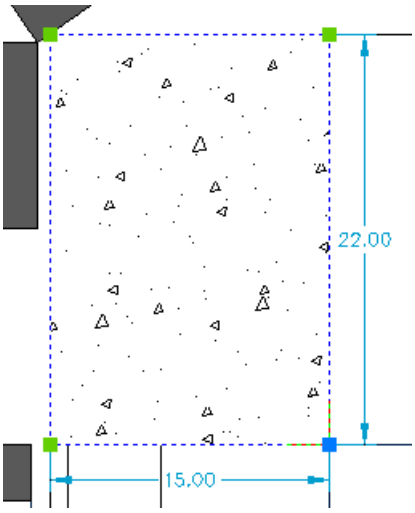
1 Select the entity or entities.

All grips on the selected entities display.

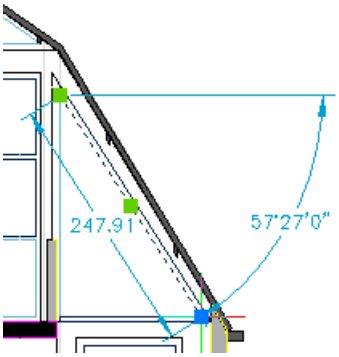
2 Hover the cursor over a grip.

The grip highlights and all dimensions related to the selected grip display.

Dynamic dimensions used to measure a rectangle



Dynamic dimensions used to measure a line



3 (option) Click to start editing the selected grip.

6.7.5 Using dynamic dimensions with tracking points

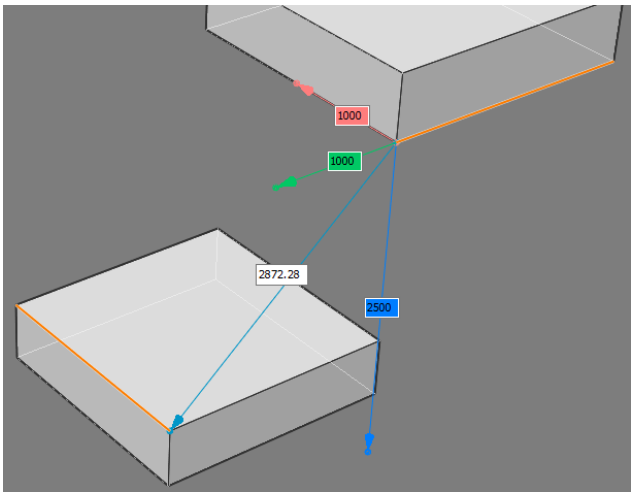
If the **Tracking dynamic dimensions** option of the DYNMODE system variable is set, dynamic dimensions display when entity snap tracking points or temporary tracking points are acquired.

6.8 Nearest distance

The Nearest Distance feature allows to dynamically view and modify the distance between two selected entities or subentities. In addition, the aligned nearest distances along the X-, Y- and Z-axes of the WCS display.

Double clicking a Nearest Distance field:

- Toggles the display of the X-, Y- and Z-axis aligned distance fields.
- Zooms out to fully display the selected entities if necessary.



TheNEARESTDISTANCEsystem variable controls the display of the Nearest Distance feature.

6.8.1 Using Nearest Distance

1 Select the first entity or subentity*.

2 Select the second entity or subentity*.

The nearest distance displays.

The arrowhead points to the second entity.

3 (option) Double click a distance field to zoom out to fully display the selected (sub)entities and/or toggle the display of the aligned nearest distances along the X-, Y- and Z-axes.

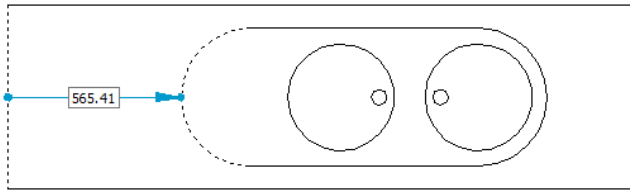
4 (option) Click a distance field and type a value.

The second entity is moved.

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

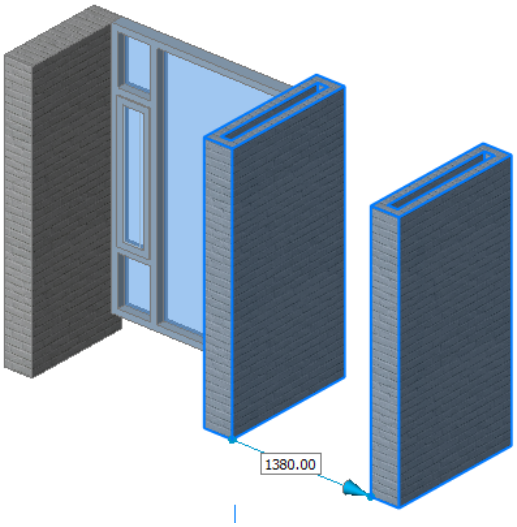
You can use the Nearest Distance feature to:

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

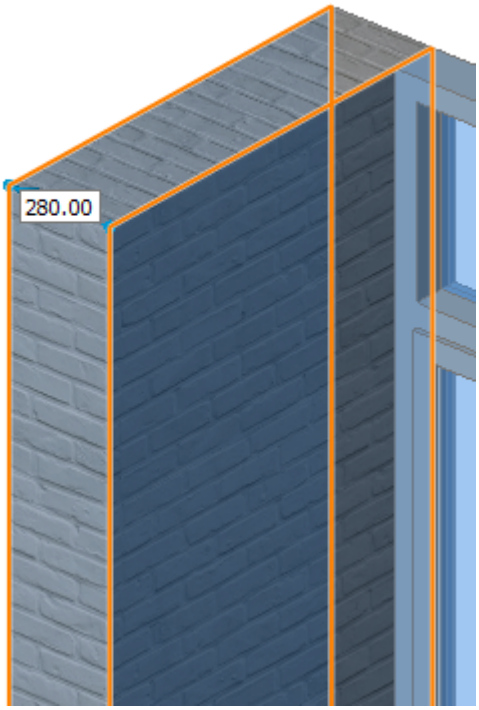


- Modify the dimensions of a rectangle.

- Adjust the distance between 3D solids.



- Edit the width of a 3D solid.



- Measure and edit the height of 3D solid.

6.9 Coordinate input

When you create entities in a drawing, they are located in relation to the underlying Cartesian coordinate system of the drawing. A drawing has a fixed coordinate system called the**World Coordinate System**(WCS).

You can also define arbitrary coordinate systems which are calleduser coordinate systems(UCS) and can be located anywhere in the WCS and oriented in any direction.

To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: R<alpha,z
- Spherical coordinates: R<alpha<beta

6.9.1 Relative coordinates

When you type the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called**Relative Coordinates**.

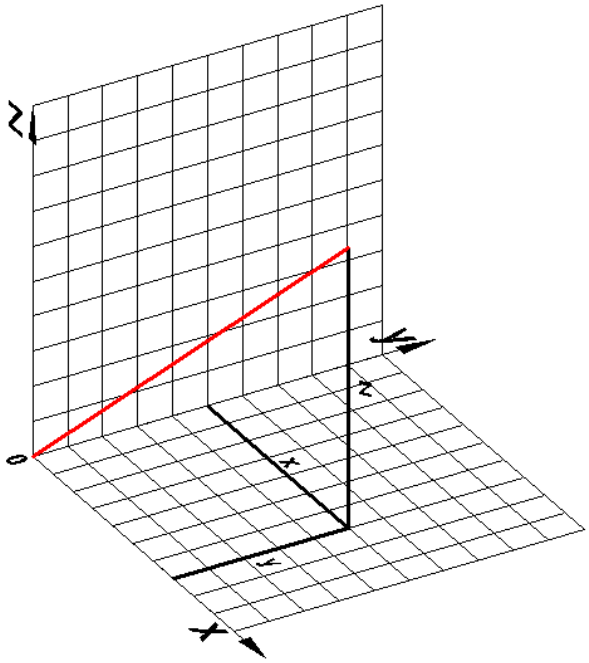
WhenDynamic Dimensionsare active and you type a value in the**Length**field and add a comma (,) the content of the**Length**field is copied to the command line and the @-character is placed in front automatically, which allows you to specify the next point using relative coordinates with respect to the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

6.9.2 Absolute coordinates

When a user defined coordinate system is active you can enter absolute coordinates (World coordinates) if you place an asterisk (*) in front. For example, *0,0 refers to the origin of the WCS (World Coordinate System).

6.9.3 Working with Cartesian coordinates

The Cartesian coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point.



- Type the x-, y- and z- coordinates separated by commas to enter the absolute Cartesian coordinates of a point.
- When the z-coordinate is omitted, the point is placed in the xy-plane (Z = 0).
- Type the @-character in front of the entry (@x,y), to specify the coordinates with respect to the previous point. This technique is called**Relative Cartesian coordinates**.

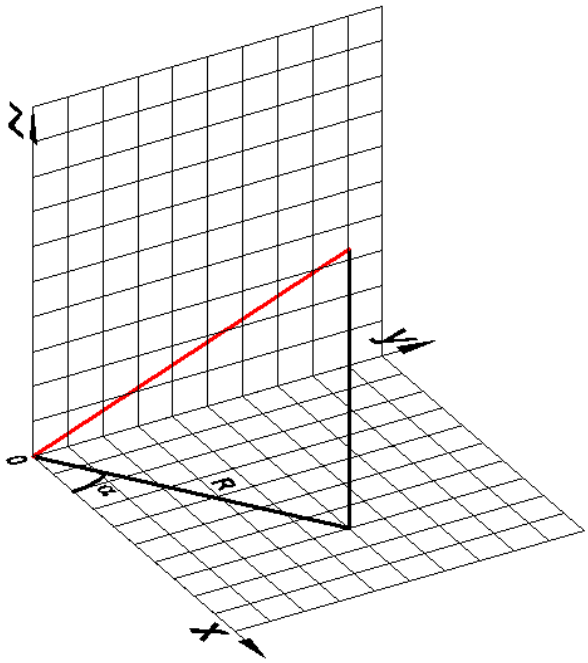
6.9.4 Working with cylindrical coordinates

A cylindrical coordinate system uses three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane
- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)

- z = the height above the xy-plane.
- When the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.
- Type the @-character in front of the entry to calculate the coordinates with respect to the previous point.

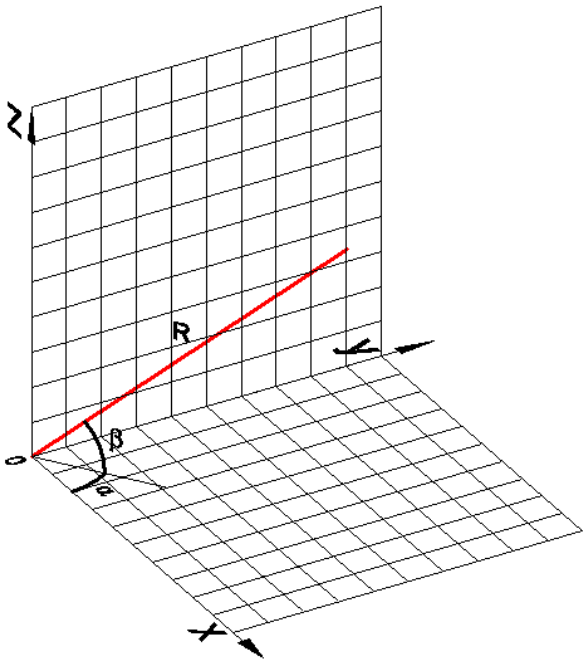


6.9.5 Working with spherical coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha<beta

- R = distance from the origin
- <alpha = angle in the xy-plane (positive angles are measured counter clockwise)
- <beta = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)

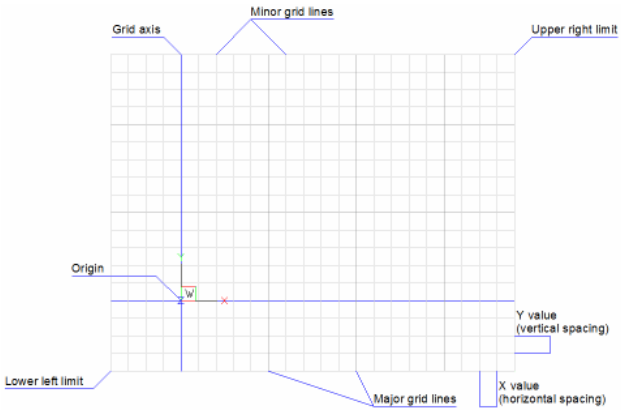


6.10 Snap and grid

Grid and snap help you to draw fast and accurately.

A grid is a set of evenly spaced lines or dots that serve as a visual distance reference.

TheGRIDSTYLEsystem variable controls whethergrid lines or dotsdisplay. TheGRIDDISPLAYsystem variable controls thedisplay of the grid. The grid can also indicate how far the drawing limits extend.



The snap feature creates a set of evenly spaced, invisible magnetic points,which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. Snap constrains the points that you can pick with the mouse.

- Click theGRIDtool ( ) on the Settings toolbar or theGRID field in theStatus Bar, or pressCtrl+Gto toggle grid display on/off.

- Click theSNAPtool ( ) on the Settings toolbar or theSNAP field in theStatus Bar, or pressCtrl+Bto toggle snap on/off. The setting is saved through the SNAPMODE system variable.

  If SNAPTYPE = 2,adaptive grid snapis used and SNAPMODE is denied.
- BothSNAPandGRIDcan be set differently in eachviewport.
- TheSnapoption of theGRIDcommand equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

6.10.1 Displaying the Snap and grid settings

Do one of the following:

- In theStatus Bar, right click on theSNAPor GRID field, then chooseSettingsin the context menu.
- In theSettingsdialog go to: **Drawing** > **Drafting** > **Coordinate Input** > **Snap** > **Grid**.

6.10.2 Synchronizing snap and grid spacing

1 Launch the GRID command.

  Prompts you:

  Grid is on. Enter grid spacing or [grid OFf/Snap/Aspect] <10>:

2 Type**s**, then press Enter.

  The grid spacing matches the snap spacing.

3 (Option) Use the Aspect option of the SNAP command to adjust the spacing.

Nota: The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using theGRID command or in the Settings dialog box.

6.10.3 Using drawing limits

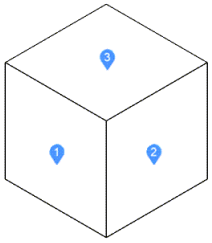
- Execute the LIMITScommand to set the drawing limits.
- Click the**Drawing Limits**item ( ) in the Settings menu to toggle theLIMCHECKsystem variable on/off.

  If LIMCHECK = ON, it is not possible to draw outside the drawing limits.
- The**Display beyond LIMITS area**option of theGRIDDISPLAYsystem variable defines whether or not the grid displays beyond the drawing limits.

6.10.4 Using isometric snap

Isometric grid allows to create two-dimensional isometric drawings, Which simulate a three-dimensional view on a two-dimensional plane. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as**Left** (1),**Right** (2)and**Top** (3). You cannot alter the arrangement of these planes. The three isometric axes are 30 degrees, 90 degrees, and 150 degrees.



Do the following:

- Set SNAPSTYL = 1 (isometric Snap).
- Set GRIDSTYLE = 1 (dotted grid).
- Set SNAPISOPAIR: 0 = Left, 1 = Top, 2 = Right.

  Press the F5 function key to cycle the isometric planes.
- Set ORTHOMODE = On to constrain the movement of the crosshairs to the current isometric plane.
- Make sure SNAPANG = 0.

6.10.5 Setting the snap angle

You can rotate a dotted grid using the SNAPANG system variable (ifSNAPANG is different from 0, grid lines do not display).

1 Display the**Snap/Grid**settings in theSettingsdialog.

2 Select SNAPANG system variable.

3 Type a new value in the setting field.

4 Close the Settings dialog.

Nota: You can also set theSnap Angleby typing**snapang**in the Command line. When in a command, type'**snapang**(with an apostrophe in front), to set theSnap Anglevariable transparently (= without interrupting the running command).

6.11 Adaptive grid snap

The adaptive grid snap feature lets you create, edit,nudgeandmanipulateentities using concise values without keyboard entry.

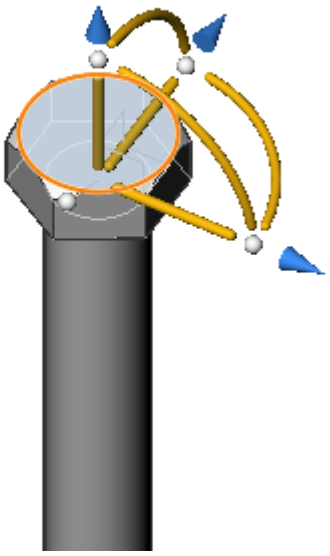
The snap step size is adapted according to the current screen zoom factor. The adaptive grid snap is active ifSNAPTYPE = 2. This system variable overrides theSNAPMODE system variable.

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

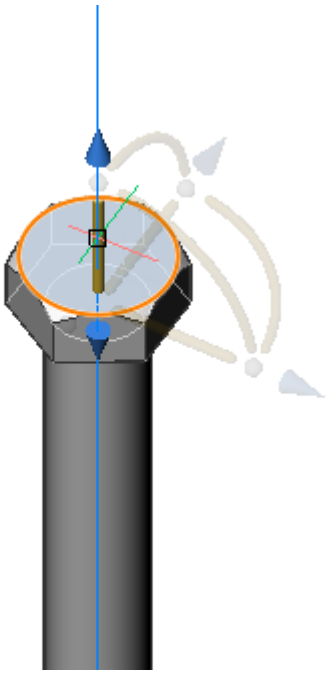
6.11.1 Using adaptive grid snap with the Manipulator

1 Move the cursor over an entity or subentity, then hold down the left mouse button a little bit longer to select the entity (= long left click).

  The Manipulator widget displays:



2 Move the cursor over the axis in the direction you want to move the entity. The selected axis highlights; the other parts of the Manipulator are faded.



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

The Ruler displays.

The current displacement distance displays in a dynamic field.

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

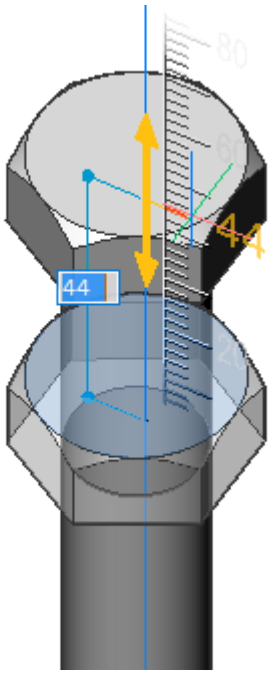
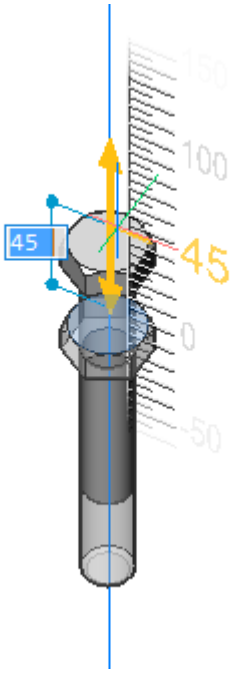


Figura 2: Ruler divisions at 2 mm

Figura 3: Ruler divisions at 5 mm



4 (option) Zoom in or out to adjust the dynamic grid.

5 Do one of the following:
   - Click to accept the current value in the Ruler.
   - Type a value in the dynamic field, then press Enter.

6.12 Nudge

The Nudge drawing aid allows you to move selected entities or subentities in small increments along the X, Y, or Z axis using the keyboard.

6.12.1 Using Nudge

1 Select one or more entities or subentities (for example polyline segments or 3D solid faces).

2 Do the following to move the selection:

- Along the X axis of the current coordinate system:

  Hold down the Ctrl key, then use the left/right arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Y axis of the current coordinate system:

  Hold down the Ctrl key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the X axis of the UCS: xx*
- Along the Z-axis of the current coordinate system:

  Hold down the Shift key, then use the up/down arrow keys.

  The Command line reads: NUDGE: offset along the Z axis of the UCS: xx*

* The step size used by the NUDGE functionality is based on theadaptive grid snapresolution. Zoom in/out to decrease/increase the step size.

6.13 2D entity snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points.

With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways:

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

If theAPBOXsystem variable is on, the entity Snap Aperture Box is added to the crosshairs when one or more entity snaps are active.

When you move the cross hairs, the program snaps to the snap point closest to the center of the Snap Aperture Box. The**Snap Marker**indicates the current snap point.

The DRAGSNAP system 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.

The OSOPTIONS system variable suppresses entity snaps on certain entity types. The value is stored as a bitcode using the sum of the values of the selected options.

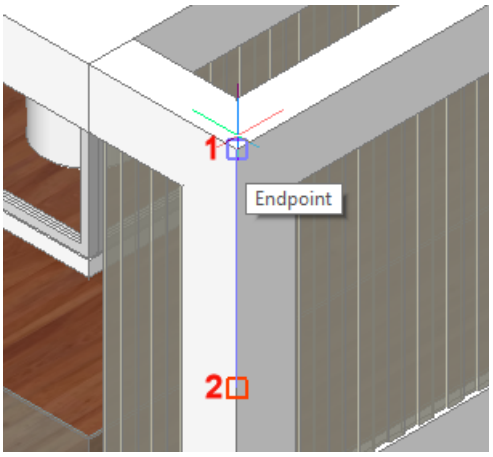
- 1: Entity snapping is disabled for hatch entities.
- 2: Entity snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- 4: Entity snapping is disabled for endpoints of dimension extension lines.

Nota: Press the TAB key, without moving the mouse, to cycle through all possible entity snaps available at the current cursor position.

6.13.1 Defining the 2D Entity Snap system variables

Right click the ESNAP field in the Status bar then choose settings in the context menu. The Settings dialog box displays with the 2D and 3D entity snaps settings expanded.

6.13.2 2D Entity Snap options

Icon	Description
	Sets the OSNAPZ system variable to Ignore entity snap elevation, replacing the Z coordinate of the snapped point with the current value of the ELEVATION system variable.  The 3d snap marker (1) displays the identified 3d point, the snap marker (2) displays at the current Elevation height.
	Disables the Ignore hatches option of the OSOPTIONS system variable, thus allowing to snap to hatches.
	Disables the Entity snap to negative-z option of the OSOPTIONS system variable, thus allowing entity snaps with a negative z-value when aDynamic UCSis active.
	Disables the Ignore endpoints of dimension extension lines option of the OSOPTIONS system variable, thus allowing entity snaps to dimension extension lines.

6.13.3 Entity Snap Modes

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
TK				tk	Use temporary tracking points. Prompts you to identify temporary points; press Enter to accept the last temporary point.
MTP			2	mtp	Snaps to a point in the middle between two points. Prompts you to identify two points.
ENDPOINT			E	end	Snaps to the nearest endpoint of an entity or polyline segment.
MIDPOINT			M	mid	Snaps to the midpoint of an entity or polyline segment.
NEAREST			N	nea	Snaps to the nearest point on an entity.
CENTER			C	cen	Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc.
GEOMETRIC CENTER			G	gce	Snaps to the centroid of any closed polylines and splines, planar 3d polylines, regions and planar faces of 3D solids.
PERPENDICULAR			P	per	Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
PARALLEL			L	par	Displays a tracking line parallel to the selected entity.
TANGENT			T	tan	Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.

Name	Icon	Marker	Context menu shortcut*	Transparent shortcut**	Description
QUADRANT			Q	qua	Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
INSERTION			I	ins	Snaps to the insertion point of an attribute, block or text entity.
NODE			O	poi nod	Snaps to a point entity.
INTERSECTION			R	int	Snaps to the intersection of any combination of entities.
APPARENT INTERSECTION			A	app	Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space. Snaps also to the intersection of any combination of entities.
EXTENSION			X	ext	Snaps to the extension of an entity or to the intersection of the extension of two entities.
NONE			Y	non	Turns off all entity snap modes.

* Type this letter to quickly select the entity snap mode in the right click context menu.

** Type the transparent shortcut in the Command line when a command prompts for a point.

Other running entity snaps are temporarily disabled.

6.13.4 Setting the Entity snaps

Do one of the following:

- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu.

  The icons of the currently active Entity Snap modes are outlined.
- Click the buttons on the Entity Snaps toolbar.

  The buttons of the currently active entity snap modes are pressed.

When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Alternatively, you can type a single letter shortcut as indicated in the table below.

To toggle theEntity Snapson/off:

- Click theESNAPfield in theStatus Bar.
- Press the**F3**function key.

6.13.5 Working with multiple Entity Snap modes

1 Move the cursor to the entity you want to snap.

One of the active Entity Snap modes markers display.

2 Do not move the mouse and press the TAB key.

The entity the snap point is on highlights.

3 Do one of the following:

- Click to accept the snap point.
- Press the TAB key.

  The next possible Entity Snap mode marker displays.

  The entity the snap point is on highlights.

4 (option) Repeat step 3 until the right snap point is found.

6.13.6 Snapping to the extension of two entities

1 If not already on, turn on the**Extension**Entity Snap mode.

2 Launch an entity creation command, for example LINE.

3 Move the cursor over the endpoint of the first entity (1).

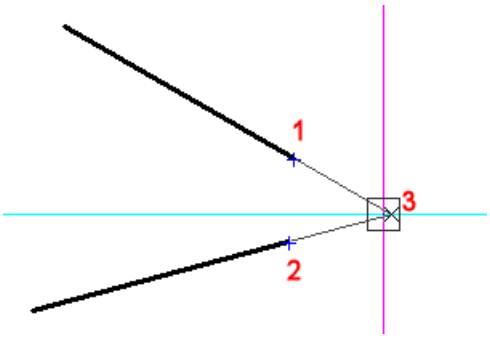
A small red cross indicates the entity is marked for extension.

4 Move the cursor over the endpoint of the second entity (2).

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

5 Move the cursor near the intersection of the extensions of the two entities.

An X indicates the intersection (3).



6 Click to accept the snap point.

Nota: You can snap to the extension of lines, polylines, arcs and elliptical arcs.

6.13.7 Using the Middle of 2 points snap mode

1 When prompted to specify a point, do one of the following:

- Activate the **Snap to the middle of 2 points** snap mode ( ).
- Type **mtp** or **m2p** in the Command line and press Enter.

You are prompted: First point of mid:

2 Specify the first point.

You are prompted: Second point of mid:

3 Specify the second point.

The point in the middle of the two points is snapped.

6.13.8 Using the Parallel entity snap mode

1 Make sure Entity Snap Tracking is 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 the 2D entity snaps, a series of 3D entity snap modes apply to points on 3D entities such as solids, surfaces, splines and meshes.

The current 3D entity snap modes are saved through the 3DOSMODE system variable as the sum of the bitcodes of the selected snap modes.

Name	Icon	Marker	Description
Center			Snaps to the center of 3D faces.
Intersection			Snaps to the intersections of linear entities, edges, and polar or entity snap tracking lines with faces.
Knot			Snaps to knots (fit points) onsplines.
Midpoint			Snaps to the midpoint of a face edge.
Nearest			Snaps to a point on the face of a 3D entity that is nearest to the cursor.
None			Disables all 3D snap modes.
Perpendicular			Snaps to a point perpendicular to a face on a 3D entity.
Vertex			Snaps to the closest vertex of a 3D entity and the control vertices ofsplines.
Point Cloud			Snaps to point cloud points.

6.14.1 Setting the 3D Entity Snap modes

- Click the buttons on the**3D Entity Snaps**toolbar.

  The buttons of the currently active entity snap modes are pressed.
- When no command is running, clicking a tool button toggles the snap mode on/off.
- When executing a command, clicking a tool button sets the snap mode for the next point, temporarily disabling all other selected snap modes.
- Run the3DSONAPor-3DOSNAPthe commands.
- Shift + Right click, hover over**3d Entity Snap**in the context menu and click the 3D entity snap option on the flyout menu.

6.15 Polar tracking

Polar tracking (POLAR) allows you to draw at exact angles:

- At fixed intervals, starting from 3 o'clock (East).
- At specific additional angles.

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as theSnap Aperture box overlaps the polar tracking line.

In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS.

Snap track lock: to lock a snap track path hit the Shift key. To unlock a track path, hit Shift again: it works as a toggle. Locked tracking paths display in thecolor defined by the SnapMarkerColoruser preference.

- The display of the tracking path line is controlled through the TRACKPATH system variable.
- The color of the tracking path line is controlled through the AutoTrackingVecColor user preference.
- Tracking paths parallel to the X-, Y- or Z-axis of the current coordinate system (WCS or UCS) display in the color of the axis they are parallel with (defined by COLORX , COLORY or COLORZ variables).
- Turning onPOLARautomatically disablesORTHO (Orthogonal Mode) and vice versa.
- The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

6.15.1 Defining the tracking settings

1 Do one of the following:
   - Right click on thePOLARor the STRACK field in theStatus Bar, then right click and select**Settings**in the context menu.
   - Open the**Settings**dialog and expand the**Coordinate input**subcategory under**Drafting**in the**Drawing**settings class.

     Expand the**Snap Tracking**settings group.

2 Adjust the settings:
   - AUTOSNAP system variable: controls the behavior of Polar Tracking and whether polar tracking and/or entity snap tracking is active.
   - POLARANG system variable: sets the polar angle increment in degrees; default = 90.
   - POLARADDANG system variable: sets additional angles; separate angles by semicolons.
   - POLARMODE system variable: controls the behavior of polar and entity snap tracking.
     - Relative: the angles are measured with respect to the direction of the snapped entity; otherwise angles are measured according to the current coordinate system (WCS or UCS).
     - Use polar angle settings for entity snap tracking also.
     - Whether additional angles (POLARADDANG) are used or not.
     - Use the Shift key to acquire entity snapping points.
   - TRACKPATH system variable: controls whether and how the track paths display: full screen or partially.

6.15.2 Locking a tracking line

Press the Shift key while a snap track marker line displays to lock a tracking path, any (snap)point that is picked will be projected perpendicularly onto the tracking path. Press the Shift key again to unlock.

A locked tracking path changes to the color defined by theSnapMarkerColoruser preference.

6.15.3 Using temporary tracking points

When**Polar Tracking** is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry. It is recommended to havedynamic dimensions(DYN) active as well.

1 Whenever a command prompts for a point, do one of the following:
   - Click the middle mouse button (mouse wheel).
   - Activate the**Temporary tracking points**entity snap mode ( ).
   - Type**TK**in the Command line and press Enter.

     The command is suspended temporarily.

     You are prompted: Specify first temporary tracking point:

2 Enter a point, for example use an entity snap to specify a point.

   You are prompted: Specify next temporary tracking point (press ENTER to end).

3 Do one of the following:
   - Enter a point.
   - Press Enter to accept the last temporary tracking.

     The command resumes.

6.16 Entity snap tracking

Entity Snap Tracking(STRACK) assists you in drawing entities with respect to other entities.

WhenEntity Snap Trackingis active, BricsCAD® displays temporary tracking lines from entity snap points.At least oneEntity Snapmode must be on if you want to useEntity Snap Tracking.

Click theSTRACKfield in theStatus barto enable/disableEntity Snap Tracking.

Entity Snap Trackinglines are parallel to the X- and Y-axis of the current UCS.

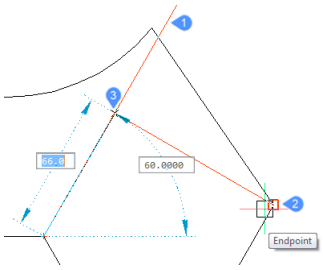
If bitcode 2 of the POLARMODE system variable is checked, the polar angles used inPolar Tracking(interval and additional angles) are also used inEntity Snap Tracking.

Alignment points are acquired when you hover over a point, using entity snaps. Acquired points display a small plus sign (+). To remove an acquired point, hover over the point again. The small plus sign (+) is then removed. If bitcode 8 of the POLARMODE is checked, press and hold the Shift key to acquire entity tracking points. Alignments relative to an acquired point are displayed as you move the cursor over their tracking paths. You can then define points either on the alignments or at the intersection of two alignments.

Nota: The ANGBASE system variable defines the base angle of the tracking lines with respect to the current coordinate system.

6.16.1 Locking a tracking line

Press the Shift key when a tracking line displays to lock the tracking line. Locked tracking lines display in the color defined by theSnapMarkerColoruser preference (1). Entity snaps (2) or specified points are projected perpendicularly onto a locked tracking line (3). Press the Shift key again to unlock a tracking line.



More options with locked tracking lines are:

- If the Intersection entity snap is active, it is possible to snap to the intersections of a locked tracking line and entities in the drawing.
- Acquire additional tracking lines and snap to the locations where these intersect with the locked tracking line.

6.16.2 Using Entity Snap Tracking

1 Launch a drawing command, for example LINE.

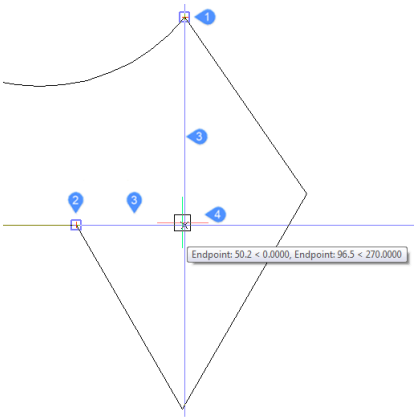
Prompts you: Start of line or [Follow] <Last point>:

2 Hover the cursor over the first tracking point (1).

A small plus sign (+) displays to indicate the tracking point is active.

3 Hover the cursor over the second tracking point (2).

A small plus sign (+) displays to indicate the tracking point is active.



4 Move the cursor close to position 4.
   - Tracking lines display from both tracking points (3).
   - A cross marker (x) indicates the intersection of the tracking lines.
   - Snap markers display at the tracking points.

- An Entity Snap Tracking tooltip at the intersection of the tracking lines shows the angles and distances from the tracking points.

5 Click to accept theEntity Snap Trackingpoint.

6.17 User coordinate systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. TheELEVATION system variable specifies the distance from the XY-plane of the current coordinate system. IfELEVATION= 0 (default), entities are created in the XY-plane of the current coordinate system.

Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

You can define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS). You can create as many UCS as you want, saving or redefining them to help you construct entities in the three-dimensional space.

When you create a new drawing, the WCS is the current coordinate system by default, which is indicated by the letter W in the UCS icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the Z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect the new viewpoint.

Nota:

- You cannot delete or modify the WCS.
- When theUCSORTHOsystem variable is On, an orthographic UCS is restored automatically when the related orthographic view is restored by the-VIEWcommand or selected on theLook Fromcontrol.

  An orthographic UCS has its origin at the WCS origin or the UCS referred to by the UCSBASE system variable.

  Orthographic views are: top, bottom, left, right, front, back.
- When multiple viewports exist, theUCSVP(UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = On) or changes to match the UCS in the current viewport (UCSVP = Off).
- When the UCSFOLLOW system variable is On, the plan view, zoomed to extents, is generated automatically whenever the coordinate system changes.
- ThePLANcommand restores the Plan view (Top view) of the current coordinate system (UCS or WCS).

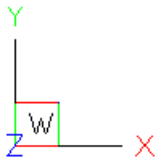
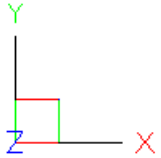
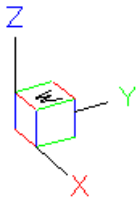
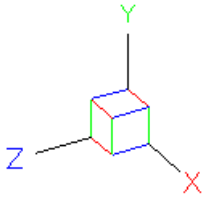
6.17.1 Controlling the display of the coordinate system icon

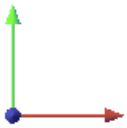
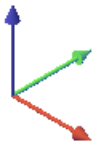
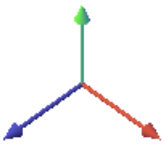
TheUCSICON and UCSICONPOS system variables control the display and location of the UCS icon:

- UCSICON:
  - **Show icon**: Controls whether the icon shows or not.
  - **At origin**: Controls the location of the UCS icon: if On, the icon indicates the origin of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport borders, the UCS icon moves to the corner of the viewport, as defined by theUCSICONPOS

system variable. When the **At Origin** option is not checked, the icon always displays in the corner of the viewport defined by UCSICONPOS.

- 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 the COLORX, COLORY and COLORZ system variables.

6.17.2 Defining a UCS

1 Type **ucs** at the command prompt.

You are prompted:

Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:

2 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

3 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

4 Specify a point to define the positive Y-axis.

The UCS is defined.

6.17.3 Defining and saving a User Coordinate System

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

2 Click the New tool button on the Drawing Explorer dialog.

The BricsCAD Explore dialog closes.

You are prompted: Current/Entity/Origin/View/X/Y/Z/ZAxis;3point

3 Press Enter to accept the default option (define the UCS using 3 points).

You are prompted: New origin <current origin>:

4 Specify the origin point of the UCS.

You are prompted: Point of positive X axis <current point>:

5 Specify a point to define the positive X-axis.

The command window reads: Point in X-Y plane with positive Y value <current point>:

6 Specify a point to define the positive Y-axis.

The UCS is defined.

The Drawing Explorer dialog reopens.

7 Click in the UCS Name field of the newly defined UCS to replace the <NewUCS> default name.

8 Close the Drawing Explorer dialog.

6.17.4 Restoring a saved UCS

1 Launch the EXPUCS command.

The Drawing Explorer - Coordinate Systems dialog box displays.

2 Click the blank tile in front of the UCS name to make it current.

3 Close the Drawing Explorer dialog box.

6.17.5 Restoring the WCS

Type **ucs** in the Command line, then press Enter twice (= choosing the default option of the UCS command).

6.18 Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with the entity under the cursor.

TheUCSDETECTsystem variable controls the supported entity types:

- 1: faces of 3d Solids (the default)

- 2 :other entity types: point, line, polyline, 2D polyline, 3D polyline, ray, xline, arc, circle, ellipse, spline, text, mtext, solid, 3Dface, trace, block insert, viewport, mline, leader, mleader, hatch, helix, camera, light, section, shape, pdf underlay and image.

Right click the DUCS field in the status bar to select the supported entity types.

This feature is controlled by user preferences.

- RedHilite_DUCSLocked_Face_Colorcontrols the highlight color for the face to which the Dynamic UCS has been locked by pressing the Shift key.
- RedHilite_DUCSLocked_Face_Alphacontrols the highlight transparency for the face to which the Dynamic UCS has been locked by pressing the Shift key.

Nota: It is recommended to set UCSFOLLOW = Off when using the dynamic UCS feature.

6.18.1 Toggling the dynamic UCS

Do one of the following:

- Press the**F6**function key.
- Click the**DUCS**field in thestatus bar.

6.18.2 Using the Dynamic UCS

1 With the Dynamic UCS feature active, launch an entity creation command, for example Draw.

2 Move the cursor over an entity or 3D solid face.

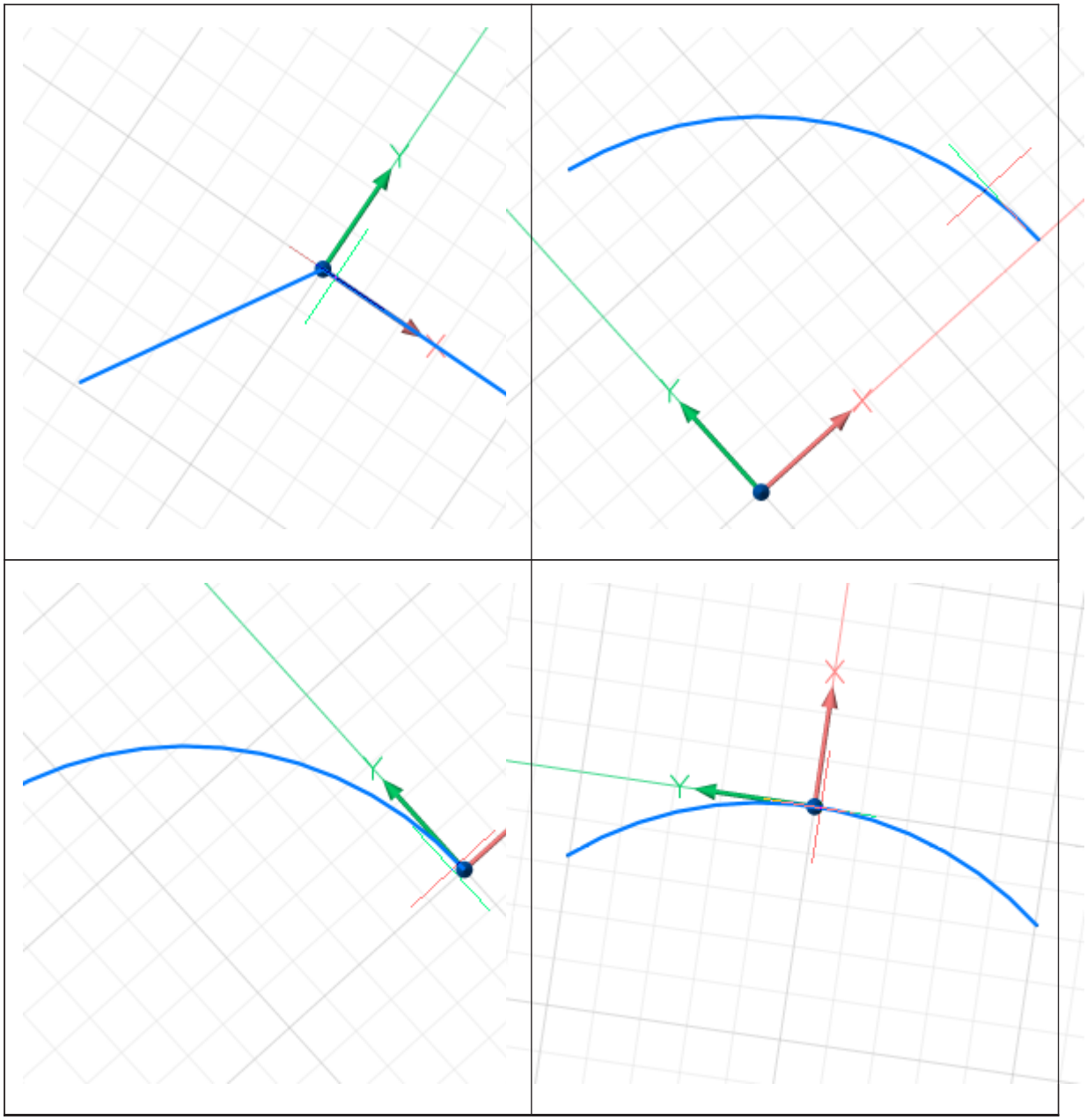
The entity or face highlights.

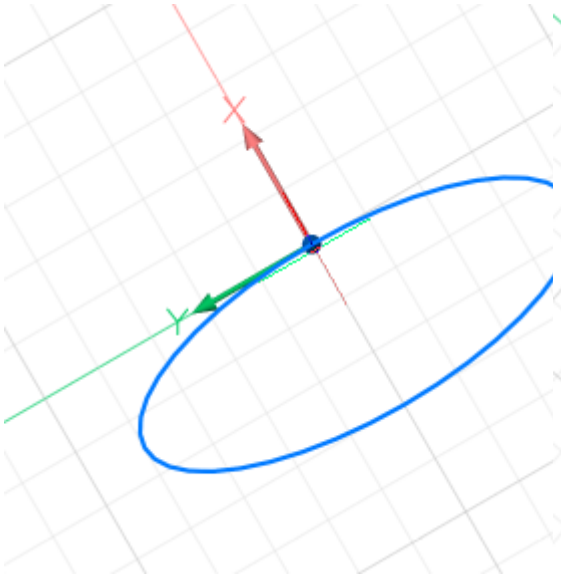
If GRIDMODE is On, the grid is aligned with the hovered entity or face.

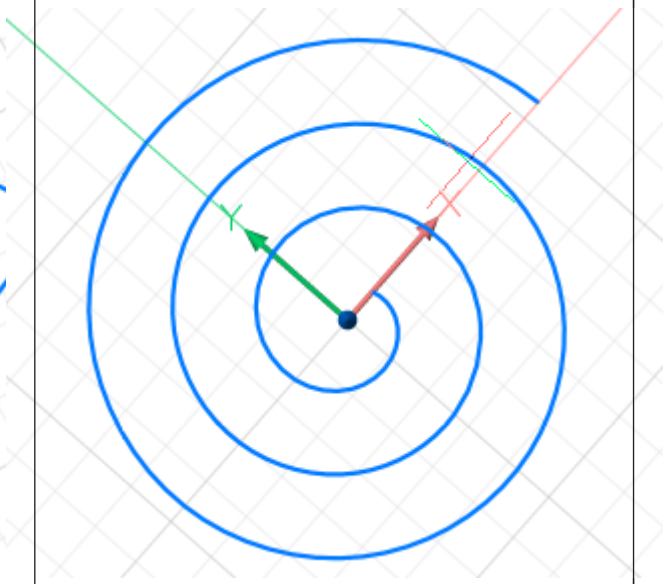
3 IfUCSICON= 3 (On at Origin), the UCS icon displays.

4

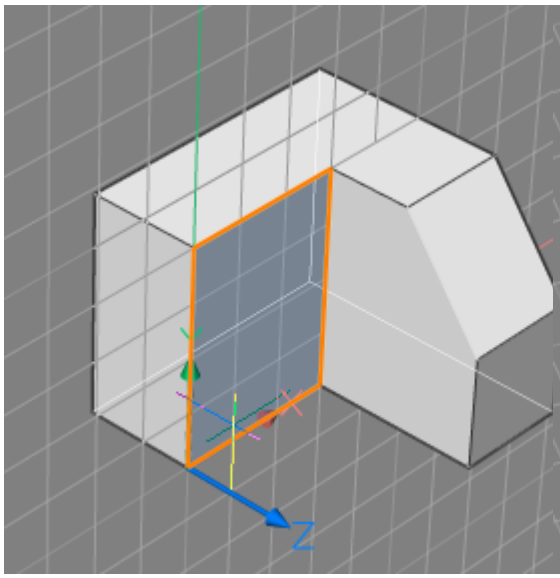
Dynamic UCS on other entities

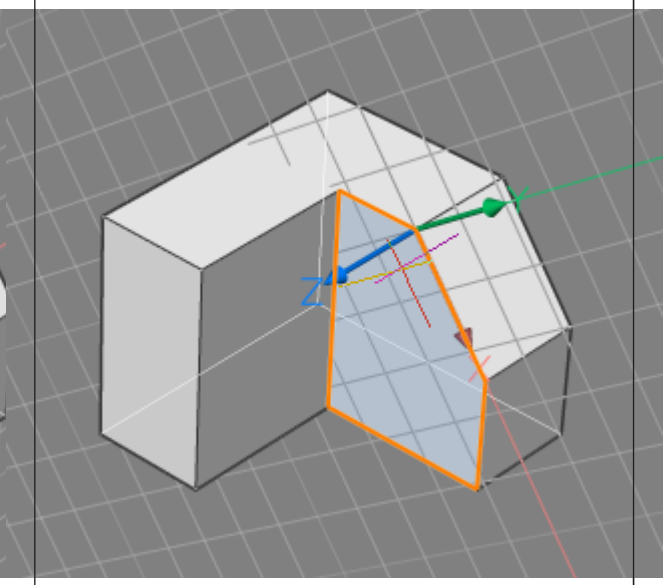


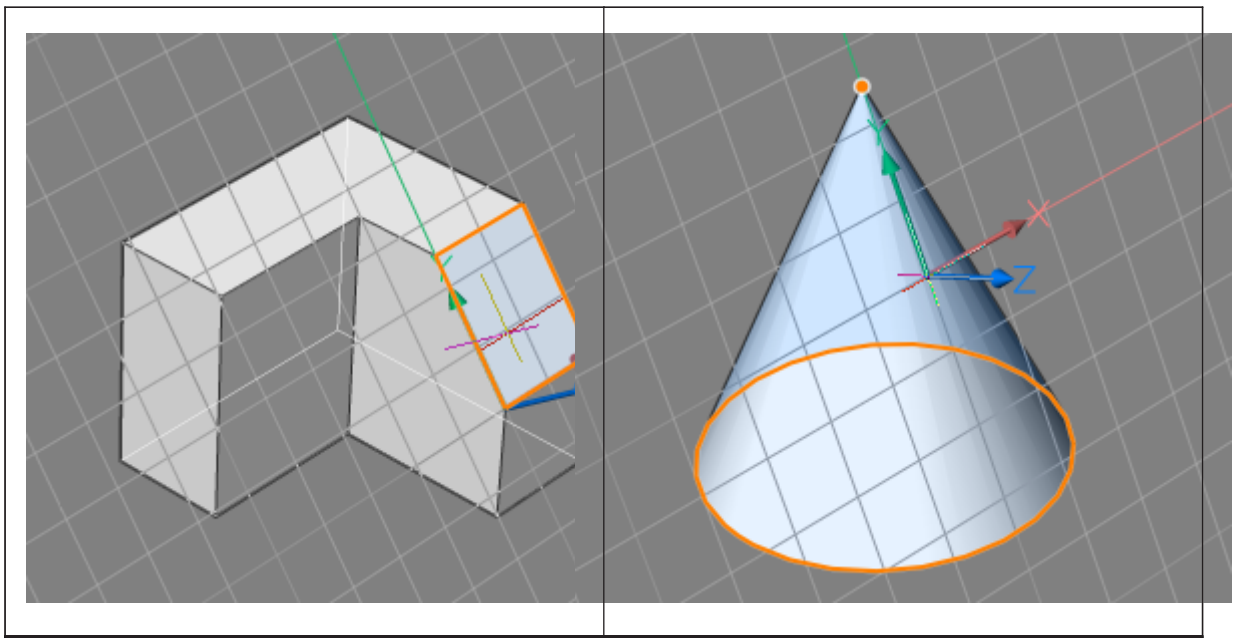




Dynamic UCS on 3D solid faces







The dynamic UCS is oriented depending on how the cursor touches the hovered entity or face, thus providing users a way to manipulate the orientation of the DUCS. On 3D solid faces the origin of the DUCS is set to the start point of the solid edge and the X-axis is chosen co-directional to the tangent at the start point of the edge. On curved faces, the X-axis is tangent to the face, the Y-axis lies in the face and the Z-axis is perpendicular to the face.

5 (option) Tap the Shift key to lock the DUCS, which allows you to start drawing outside the hovered entity or face. Tap the Shift key again to unlock the DUCS.

6 Execute the entity creation command.

Nota: When bitcode 2 of the OSOPTIONS system variable is selected, entity snap ignores negative Z-values.

7 When the command concludes, the current coordinate system (WCS or UCS) is restored.

6.19 Measuring

The DIST command reports the distance and angle between two points.

The AREA command finds the area and perimeter (or length) of 2D entities.

The MASSPROP command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for **mass properties**).

The ID command reports the x,y,z coordinates of picked points.

The LIST command lists the properties of selected entities.

Nota: BricsCAD® reports the results in the Command line or in the Prompt History window when the Command line is closed. Depending on the number of prompt history lines in the Command line, it can be necessary to press the F2 function key to open the Prompt History window. Press the F2 function key again to close the Prompt History window.

6.19.1 Measuring distances

To measure a distance between 2 points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Hover over the second point using an appropriate entity snap mode.

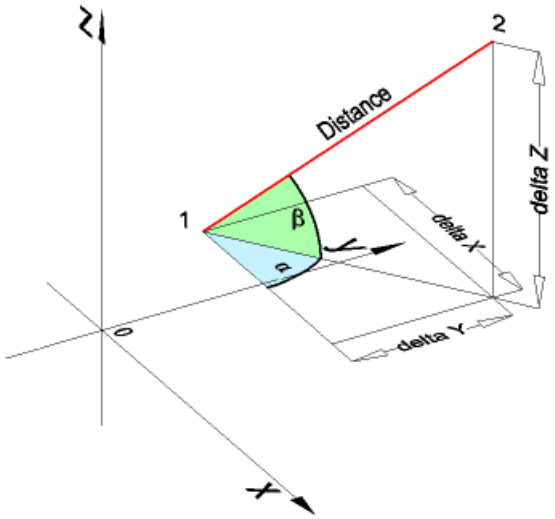
When Dynamic Input (DYN) is On, the distance displays in thedynamic dimensionfield.

4 Click to confirm the second point.

BricsCAD® prints the results in the Command line:

Distance, Angle in XY Plane (α), Angle from XY Plane (β)

Delta X, Delta Y, Delta Z



The values are in drawing units and with respect to the currentcoordinate system.

To measure a distance between multiple points:

1 Launch the DIST command.

Prompts you: Starting point for distance.

2 Identify the first point.

Prompts you: Enter second point or [Multiple points]:

3 Choose the Multiple points option.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

4 Identify the second point.

BricsCAD® displays the distance between the two points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

5 Identify the next point.

BricsCAD® displays the total distance between the points.

Prompts you: Enter next point or [Arc/Length/Undo/Total] <Total>:

6 Keep adding points, then press Enter or right click to conclude.

6.19.2 Measuring lengths

To measure the length of single linear entity, click the entity then read the **Length** field in the Properties panel.

To combine the lengths of a series of linear entities:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Choose the **Add areas together** option.

Prompts you: Draw first point to add to current area or [add Entities to area/Subtract from area]:

3 Choose the **ad Entities** option.

Prompts you: Select entities to add to area:

4 Select the first linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

5 Select the second linear entity.

BricsCAD® reports in the Command line:

Area = xx, Length = xx

Total length = xx

Total area = xx

Prompts you: Select entities to add to area:

6 Keep selecting linear entities or press Enter or right click to stop.

Nota: You can combine the lengths of lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

6.19.3 Measuring areas

To measure the area and length of single closed entity, click the entity then read the **Area** and **Length** fields in the Properties panel.

To measure areas and perimeters by specifying points in a drawing:

1 Launch the AREA command.

Prompts you:

Select first point [find area of one Entity/Add areas together/Subtract areas]:

2 Click the first point.

Prompts you: Next point.

3 Click the second point.

Prompts you: Next point.

A dashed outline displays dynamically.

4 Keep adding points, then press Enter or right click.

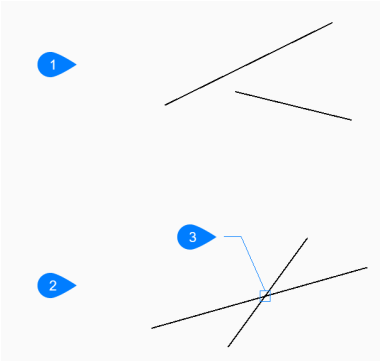
BricsCAD® reports in the command line:

Area = xx, Perimeter = xx

6.20 Entity snaps overview

6.20.1 Apparent intersection entity snap

The Apparent intersection entity snap enables the cursor to snap to the intersections of entities, even when they only appear to intersect in 3D space.



1 In one 3D viewpoint, the lines do not intersect.

2 In another 3D viewpoint, the lines appear intersect.

3 Apparent intersection.

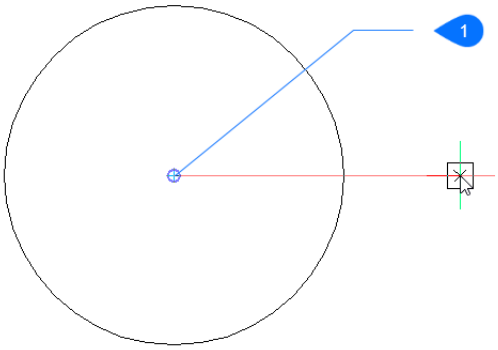
6.20.2 Center entity snap

The Center entity snap enables the cursor to snap to the center point of circular or elliptical entities.

Including:

- Circles
- Arcs
- Polyline arcs
- Ellipses
- Elliptical arcs
- Circular 3D faces

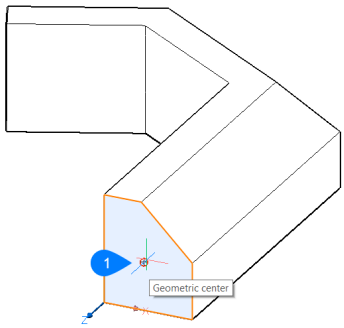
- Elliptical 3D faces



1 Center of circle.

6.20.3 Center of a Face 3D entity snap

The Center of a Face 3D entity snap enables the cursor to snap to the geometric center of a 3D face.



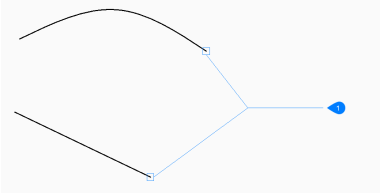
1 Geometric center entity snap.

6.20.4 Snap a entità punto finale

Lo snap ad entità punto finale consente al cursore di eseguire lo snap ai punti finali delle entità.

Incluso:

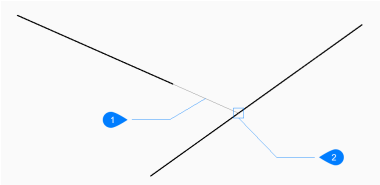
- Linee
- Archi
- Segmenti polilinee
- Spline aperte
- Spigoli 3D



1 Snap snap ad entità punto finale e centro

6.20.5 Extension entity snap

The Extension entity snap enables the cursor to snap to the intersection of one or two extended entities.



1 Extension

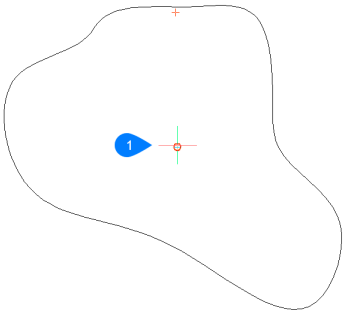
2 Extension entity snap.

6.20.6 Geometric center entity snap

The Geometric center entity snap enables the cursor to snap to the centroid of closed entities.

Including:

- Polylines
- Splines
- Regions
- Planar 3D faces



1 Geometric center entity snap.

6.20.7 Insertion entity snap

The Insertion entity snap enables the cursor to snap to the insertion points of entities.

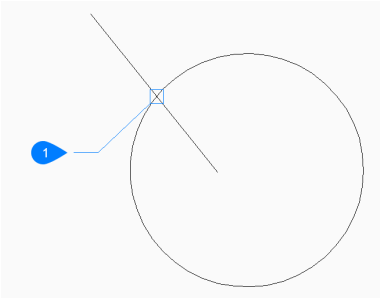
Including:

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

1 Insertion entity snap.

6.20.8 Snap a entità intersezione

Lo snap ad entità Intersezione consente al cursore di agganciarsi all'intersezione di entità che si intersecano fisicamente l'una con l'altra.



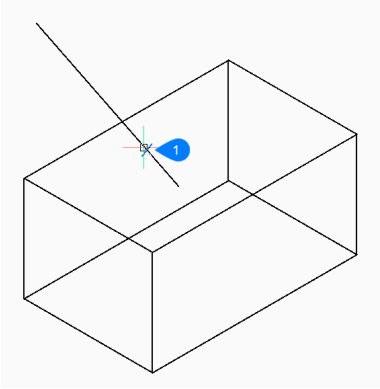
1 Snap a entità intersezione.

6.20.9 Snap a Entità 3D Intersezione

Lo Snap a Entità 3D Intersezione consente al cursore di eseguire lo snap all'intersezione delle facce con entità lineari.

Incluso:

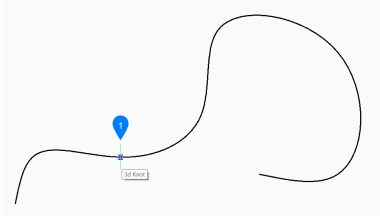
- Linee
- Segmenti polilinee
- Spigoli 3D
- Linee di puntamento polare
- Linee di tracciamento snap entità



1 Snap a Entità 3D Intersezione.

6.20.10 Knot 3D entity snap

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



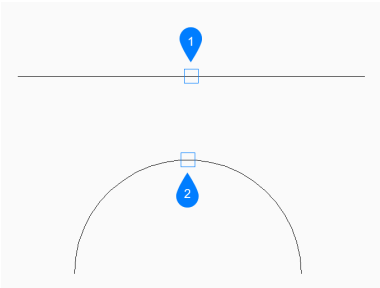
1 Knot 3D entity snap.

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

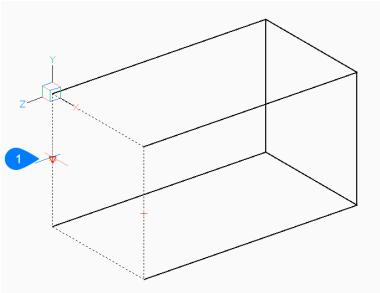


1 Midpoint (of line).

2 Midpoint (of arc).

6.20.12 Midpoint 3D entity snap

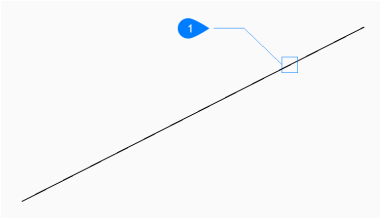
The Midpoint 3D entity snap enables the cursor to snap to the midpoint of 3D edges.



1 Midpoint 3D entity snap.

6.20.13 Nearest entity snap

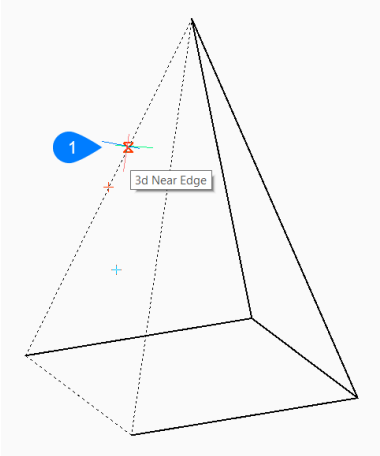
The Nearest entity snap enables the cursor to snap to 2D or 3D geometry nearest to the cursor.



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.

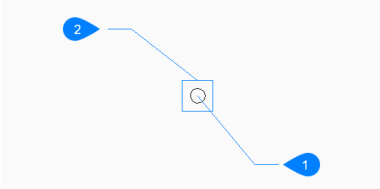


1 Nearest 3D entity snap.

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

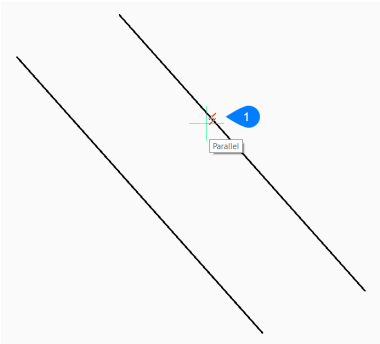


1 Point.

2 Node entity snap.

6.20.16 Parallel entity snap

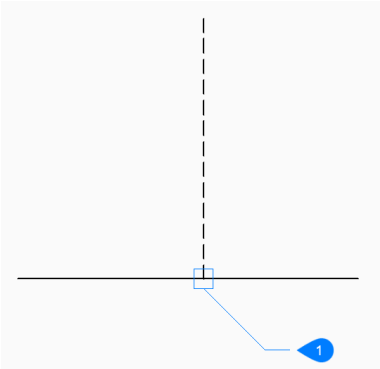
The Parallel entity snap enables the cursor to snap to a point parallel to an existing entity.



1 Parallel entity snap.

6.20.17 Perpendicular entity snap

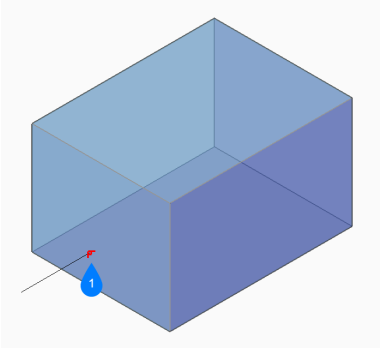
The Perpendicular entity snap enables the cursor to snap to a point perpendicular to an existing entity.



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.



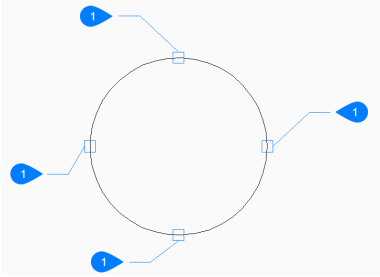
1 Perpendicular to a face 3D entity snap.

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.

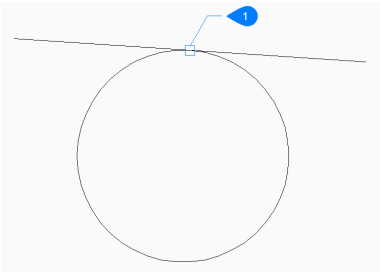
6.20.20 Tangent entity snap

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

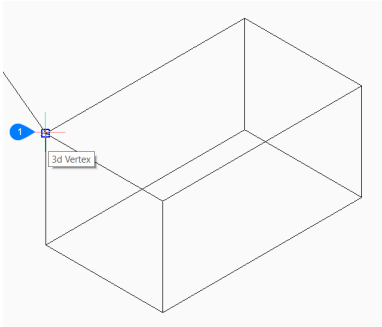
- Circular 3D faces
- Elliptical 3D faces



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. Visualizzazione del disegno

7.1 View commands overview

BricsCAD provides various tools and commands to control the display and view orientation of your drawing.

These tools can be found in the **View** menu, toolbar or ribbon tab.

Icon	Tool Name	Command	Description
	Redraw	REDRAW	Refreshes the screen display in the current viewport.
	Redraw All	REDRAWALL	Refreshes the screen display in all open viewports.
	Regenerate	REGEN	Recalculates the screen display in the current viewport.
	Regenerate All	REGENALL	Recalculates the screen display in all open viewports.
	Update Fields	UPDATEFIELD	Updates the selected fields.
	Real Time Pan	RTPAN	Pans the view dynamically.
	Pan	PAN	Pans the view.
	Real Time Zoom	RTZOOM	Zooms in/out dynamically.
	Zoom In	ZOOM + *I(*)*	Zooms in on the center of the window by a factor of 2.
	Zoom Out	ZOOM + *O(*)*	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents	ZOOM + *E(*)*	Displays all the entities in the drawing (referred to as the drawing extents).
	Zoom Window	ZOOM + *W(*)*	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous	ZOOM + *P(*)*	Restores the displayed view prior to the current one.

Icon	Tool Name	Command	Description
	Zoom All	ZOOM + *A(*)*	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	ZOOM + *L(*)*	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	ZOOM + *C(*)*	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	ZOOM + *R(*)*	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around	RTLOOK	Look around in a 3D scene.
	Look Up and Down	RTUPDOWN	Move up / down and left / right in a 3D scene.
	Walk	RTWALK	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere	RTROT	Rotates the view dynamically in 3D. Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world Z-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis.
	Real-Time Sphere Center	RTROTCTR	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.
	Real-Time Free Sphere	RTROTF	Rotates the view dynamically in 3D. Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen Y-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen X-axis.
	Real-Time X	RTROTX	Rotates the view dynamically about the screen X-axis.
	Real-Time Y	RTROTY	Rotates the view dynamically about the screen Y-axis.
	Real-Time Z	RTROTZ	Rotates the view dynamically about the screen Z-axis.

Icon	Tool Name	Command	Description
	Set View Point	DDVPOINT	Opens the **Preset Viewpoints** dialog.
	Plan View	PLAN	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	DVIEW	Defines parallel and visual perspective views.
	Save/Restore View	VIEW	Saves and restores named views.
	Viewports	VPORTS	Manages viewports in model space.
	Paper Space Views	MVIEW	Creates viewports in paper space.

(*) <command> <option>: type **zoom** followed by Enter, then type the command option letter, followed by Enter

7.2 Layer utilities

The layer utility commands help you to easily manipulate the **On/Off**, **Thaw/Freeze** and **Unlocked/Locked** layer properties by selecting entities. You can then save the result in a Layer State.

The layer utility commands sit on the **Layers** toolbar and on the **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Command	Icon	Description
LAYON		Turns on all layers that have been turned off.
LAYOFF		Turns off layers associated with selected entities.
LAYFRZ		Freezes layers associated with selected entities.
LAYTHW		Thaws all frozen layers.
LAYLCK		Locks layers associated with selected entities.
LAYULK		Unlocks layers associated with selected entities.

Command	Icon	Description
LAYISO		Isolates layers associated with selected entities. If no entity on the current layer is selected, the layer of the entity that is selected first temporarily becomes the current layer until the LAYUNISO command is executed.
LAYUNISO		Restores layers isolated by the LAYISO command.

7.3 Hide and isolate entities

Command	Icon	Description
HIDEOBJECTS		Hides the selected entities.
ISOLATEOBJECTS		Displays the selected entities only.
UNISOLATEOBJECTS		Brings back all previously hidden entities.

The tools to launch these commands can be found on:

- the **Hide/Show Entities** and **Access 3D** toolbars.
- the **Select** Quad tab.
- The **Home / Layers** ribbon panel in the **Drafting** and **Modeling** workspaces.

Show, Hide and Isolate entities is also available in the right-click context menu in the **Structure** panel.

Nota: The OBJECTISOLATIONMODE system variable controls whether hiding and isolating entities is saved or not.

7.4 Redrawing and regenerating a drawing

The screen display of a drawing is really a simplified version of the drawing database. From time to time, it is necessary to synchronize the screen display and the drawing database.

The following commands are available:

- The REGEN command (short for "regenerate") regenerates the current viewport.
- The REGENALL command (short for "regenerate all") regenerates all viewports.
- The REGENAUTO command controls when BricsCAD automatically regenerates the drawing.
- The REDRAW command redraws the current viewport to clean it up.
- The REDRAWALL command redraws all viewports to clean them up.
- The REGENMODE system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of REGENMODE through the REGENAUTO command. When REGENMODE = ON, BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might still be necessary.

Take care not to confuse the REGEN command with the REDRAW command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

7.5 Panning

The PAN and RTPAN commands move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed.

7.5.1 Using the pan command

1 Launch the PAN or RTPAN command.

1 The shape of the cursor changes to a hand.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.6 Zooming

The **ZOOM** command visually changes the size of the drawing within the current viewport.

The RT**ZOOM** command zooms the drawing in real-time.

You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass when aN RTZOOM tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

7.6.1 Real-time zooming

1 Launch the RTZOOM command.

1 The shape of the cursor changes to a magnifying glass.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.

- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.6.2 Zooming to extents

Do one of the following:

- Double-click the middle mouse button (scroll wheel).
- Launch the ZOOM command, then type E and press Enter.

7.6.3 Dynamic zooming

Pans and zooms using a rectangular view box, representing the current viewport. The command first zooms out to display the entire drawing. You can then control the size of the view box dynamically and position the view box on the part of the drawing you want to display in the current viewport.

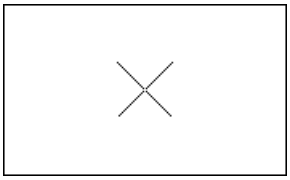
1 Launch the ZOOM command.

2 Choose the **Dynamic** option.

The drawing zooms out to extents.

A dashed rectangle indicates the area of the previous view.

A view box of the same size as the dashed rectangle is attached to the cursor.



3 (option) To resize the view box:

- Click, then move the cursor to resize.

- Click again to stop resizing.

4 Position the view box over the drawing area you want to enlarge, then right-click.

5 The drawing is zoomed in on the selected area.

7.7 View manipulation using the mouse and keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts.

In perspective mode you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse or Keyboard	Action	Result
Mouse wheel	scroll	Zoom in / out. The incremental change in zoom with each mouse-wheel action is controlled through the **Zoom Factor** variable (ZOOMFACTOR system variable). Default = 60. Values between 3 and 100 are accepted. When zooming in, the incremental step decreases gradually, allowing you to focus on a particular detail easily. In a 3D model, the entity under the cursor has the zoom focus. The ZOOMWHEEL system variable controls the zoom direction: 0 or OFF: scrolling forwards zooms in, backwards zooms out. 1 or ON: scrolling forwards zooms out, backwards zooms in.
Middle mouse button or wheel*	press and hold while moving the mouse	Real-time pan (RTPAN command)
Middle mouse button or wheel*	double-click	Zoom extents
Ctrl + Shift + Left mouse button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM command)
Ctrl + Shift + Right mouse button	press and hold while moving the mouse	Constrained real-time pan (RTPAN command). Release the Shift key for free panning.
Ctrl + Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time sphere (RTROTF command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ command)

Mouse or Keyboard	Action	Result
Walk through navigation		
Alt + Left Button	press and hold while moving the mouse	Walk left / right and forward / backward (RTWALK command)
Alt + Middle Button	press and hold while moving the mouse	Move up / down and left / right (RTUPDOWN command)
Ctrl + Middle Button	press and hold while moving the mouse	Look around (RTLOOK command) RTWALK command
Ctrl + Arrow Keys	press the arrow keys while holding the Ctrl key	Walk left / right and forward / backward ()
Ctrl + Shift + Arrow Keys	press the arrow keys while holding the Alt + Ctrl keys	Look up / down and left / right (RTUPDOWN command)
Ctrl + Home		Sets the view direction horizontal
Alt + Home		Moves target point to center of scene
Alt + plus/minus		Increase / decrease movement speed
Ctrl + plus/minus		Increase / decrease rotation speed

* On condition the **Middle Button Pan** (MBUTTONPAN) system variable is set **ON**.

7.8 View rotation

The following command are available for view rotation operations:

- The **RTROT** (Real-Time Constrained Sphere) command rotates 3D drawings in real-time.
- The **RTROTCTR** (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point.
- The **RTROTF** (Real-time Free Sphere) command freely rotates 3D drawings in real-time.

- The **RTROTX, RTROTY** and **RTROTZ** commands rotates 3D drawings in real-time about the x-, y- or z-screen axis.
- The **DDVPOINT** command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").
- The **PLAN** command restores the plan (top) view.
- The **VPOINT** command changes the 3D viewpoint.

Nota:

- If the **CONTINUOUSMOTION system** variable is set, the view rotation continues until you press the Esc key to conclude the command.
- View rotation commands should not be used when working in 2D. Use the PLAN command or the LookFrom control to restore top view.

7.8.1 Rotating a view freely

1 Launch the RTROT, RTROTF, RTROTX, RTROTY or RTROTZ command.

1 The shape of the cursor changes according to the selected command.

2 Hold down the left mouse button, then move the mouse.

3 (option) Right-click to display a context menu.

4 The options are:

- **Exit**: exits the command.
- **Pan**: switches to the RTPAN command.
- **Zoom**: switches to the RTZOOM command.
- **Constraint Sphere**: switches to the RTROT command.
- **Sphere**: switches to the RTROTF command.

5 Press the Esc key to exit the command.

7.8.2 Using Preset Viewpoints

Nota: Switch off Perspective mode before starting the command.

1 Launch the DDVPOINT command.

1 The **Set Viewpoint** dialog box appears.

2 Click the **Angle Precision** button to set the display mode for the **Set Viewpoint** dialog box.

The options are: **45°**, **15°** or **5°**.

The **Horizontal** and **Vertical** select panes are divided accordingly.

3 Click an angle in the **Horizontal** and **Vertical** select panes.

The view orientation is updated.

4 (option) Click the **Plan** button to restore the plan (top) view.

5 (option) Repeatedly click the **Previous** button to restore previous viewpoints.

6 To conclude the command:
   - Click the **OK** button to keep the current view.
   - Click the **Cancel** button to restore the original view.

7.9 The LookFrom control

The **LookFrom** control allows you to select a number of preset views such as orthographic views and isometric views. By default, the **LookFrom** control is displayed in the top right corner of the graphic screen.

The NAVVCUBEDISPLAY system variable sets the display of the LookFrom control.

7.9.1 Context menu options

When you right-click the LookFrom control, a context menu appears. The options are:

- **UCS**: sets the current coordinate system.
  - **World**: restores the WCS (World Coordinate System)
  - **New**: runs the UCS command.
  - **UCS Manager...**: runs the EXPUCS command; displays the **Drawing Explorer / Coordinate Systems** dialog box.
- **View**:
  - **New**: runs the -VIEW command.
  - **View Manager...**: runs the VIEW command; displays the **Drawing Explorer / Views** dialog box.
- **Visual Style**: Allows you to set the visual style.
- **Isometric Mode**: Sets the isometric mode.
- **Twist Mode**: Sets the Twist mode.
- **WCS**: LookFrom uses the WCS as reference coordinate system.
- **UCS**: LookFrom uses the current coordinate system (WCS or UCS) as reference coordinate system.
- **Perspective**: Sets the perspective view mode.
- **Location**: Sets the location of the LookFrom control.
- **Dismiss**: Closes the LookFrom control.

  Edit the NAVVCUBEDISPLAY system variable to restore the LookFrom control.

7.9.2 LookFrom control settings

The behavior of the **LookFrom** control depends on a number of system variables and user preferences.

To access the LookFrom settings:

1 Right-click the LookFrom control and choose **LookFrom Settings...** in the context menu.

2 Run the LOOKFROM command and choose the **Settings** option.

7.9.3 Setting the LookFrom mode

The LookFrom control runs in one of two modes:

- Isometric mode for working in 3D models.
- Twist mode for 2D drafting.

To set the LookFrom mode:

1 Right-click the LookFrom control.

2 Select **Isometric Mode** or **Twist Mode** in the context menu.

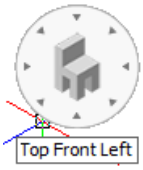
7.9.4 Using the LookFrom control in Isometric mode

1 Move the cursor to the **LookFrom** control.

The **LookFrom** control becomes active.

2 Hover the cursor over a view orientation.

3 The chair in the center of the LookFrom control rotates according to the view orientation.



4 (option) Hold down the Ctrl key to look bottom-up.

5 Click to confirm.

6 The view is rotated.

7.9.5 Using the LookFrom control in Twist mode

In **Twist** mode, the **LookFrom** control rotates the view around the screen Z-axis.

1 Move the cursor to the **LookFrom** control.

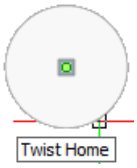
The **LookFrom** control becomes active.

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

3 Click to rotate the view.

4 (option) Without moving the mouse, click again to rotate over the same angle again.

5 (option) Move the cursor to the center of the **LookFrom** control, then click to restore an unrotated view.



7.9.6 View transition options

The following system variables control the behavior of view transitions:

- VTENABLE: determines whether animated view transitions are enabled for zoom/pan and/or for view rotation operations.
- VTDURATION: determines the duration of view transition animations in milliseconds.
- VTFPS: defines the minimum frame rate required to allow animation. That is 7 by default, which means that the redraw time should take less than 143 (=1000/7) milliseconds. If the computer is not capable to redraw the view fast enough, no animation will be available.

7.10 Layer states

Layer states let you save and restore configurations of layer properties. You might want a layer to be displayed in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

To easily create layer states, the Layer Utility commands might be useful.

The LAYERSTATE command opens the **Drawing Explorer / Layer States** dialog box.

7.10.1 Saving the current layer state

1 Click the **New** button on the **Drawing Explorer / Layer States** dialog box.

   A new layer state is created.

2 Type a name for the layer state, replacing the **NewLayerState1** default name.

3 (option) When in **Detail View**, type a description in the **Description** field.

4 Select the **Layer Properties to restore**.

   Click the tick box to select a layer property, click again to remove a property.

5 Select a **Restore option**.

6 When in paper space, tick the **Apply properties a viewport overrides** option.

Nota: In a new layer state, all layer properties are selected to be restored by default.

7.10.2 Editing layer states

1 (option) Click the **Overwrite** button to overwrite the layer state with the current layer settings.

2 (option) Rename the layer state.

3 (option) Edit the layer state description.

4 (option) Edit the **Layer Properties to restore**.

   Click the tick box to select a layer property, click again to remove a property.

7.10.3 Restoring layer states

1 Do one of the following:
   - Click the **Restore** button on the **Drawing Explorer / Layer States** dialog box.
   - Right-click, then choose **Restore** in the context menu.

2 Click the **Regen** button, unless the **Regen Auto** button is pressed.

Nota:

- The restore option of the layer state applies to layers which are added after the selected layer state was created. The options are:
  - Turn Off.
  - Freeze
  - Do nothing.
- Layer states can be restored using the **Layer State** control, which you can add to a ribbon panel or a toolbar.

7.10.4 Copying layer states between open drawings

1 Open both, the source and the target drawing.

2 Select the source drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

3 Select the layer state(s) you want to copy.

4 Press and hold the Ctrl key to select multiple layer states.

5 Hold down the left mouse button, then drag-and-drop the selection onto the target drawing in the **Open Drawings** list on the **Drawing Explorer / Layer States** dialog box.

   The selected layer states are copied to the target drawing.

7.10.5 Exporting layer states

1 Select the layer state on the **Drawing Explorer / Layer States** dialog box.

2 Click the **Export** button.

   The **Export Layer States** dialog box appears.

3 In the **File Name** field, type a name for the layer state file.

4 Select a destination folder.

5 Click the **Save** button.

7.10.6 Importing layer states

1 Click the **Import** button on the **Drawing Explorer / Layer States** dialog box.

   The **Import Layer States** dialog box appears.

2 Select the layer state file to import.

3 Click the **Open** button.

   The layer state is imported.

7.11 Named views

You can save the view in the current window as a named view. Saved views can be restored at any time.

A Background can be assigned to a named view, which will be used when the view is rendered.

7.11.1 Exploring views

1 Launch the VIEW command.

1 The **Drawing Explorer / Views** dialog box appears.

2 (option) Click the blank tile in front of the **View Name** to restore the view.

The current view is marked.

3 (option) Click the **New** button in the **Drawing Explorer** toolbar to save the current view (launches the -VIEW command).

4 (option) Click the **Delete** button in the **Drawing Explorer** toolbar to delete the selected view.

5 (option) Click the **View Name** , then right-click and choose **Rename** in the context menu.

Type a new name for the saved view.

6 (option) Check the **Perspective** property to make the view a visual perspective view.

If not checked the view is a parallel perspective view.

7 (option) Click the **Background** column of a view to choose a background. The background definition of the view is applied during rendering.

7.11.2 Saving a view

1 Do one of the following:

- Launch the -VIEW command.
- Click the **New** button in the **Drawing Explorer / Views** toolbar.

2 Choose the **Save** option.

-VIEW command workflow:

- **You are prompted**: Save view as:
- Type a name for the view in the Command line, then press Enter.

Drawing Explorer workflow:

- The **Drawing Explorer / Views** dialog box appears.
- Type a name to replace the NewView1 default name.

Nota: If you type a name in the Command line that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

7.11.3 Restoring a view

In the **Drawing Explorer / Views** dialog box, click the **Current** field of the view.

7.11.4 Editing the view background

1 Select the view in the **Drawing Explorer - Views** dialog box.

2 Select the **Background** property.

3 Click the **Browse** button at the right hand side of the settings field.

The **Background** dialog box appears.

4 Click the **Type** list button, then select the background type.

The options are **None**, **Solid**, **Gradient** and **Image**.

5 Select a background type.

6 The settings for the selected type are displayed in the **Background** dialog box.

7 Specify the background settings.

8 Click the **OK** button on the **Background** dialog box.

9 (option) Click the **Current** field.

Nota: The GRADIENTMODE system variable sets the default for gradient backgrounds: 2 or 3 colors.

7.12 Visual styles

A visual style is a collection of settings that control the display of 3D solids and surfaces.

The VISUALSTYLES command opens **Drawing Explorer – Visual Styles**.

Here you can:

- Edit existing visual styles.
- Create new visual styles.
- Delete visual styles.
- Copy and paste visual styles between open drawings.

7.12.1 Setting the current visual style

The current visual style can be set in the **View / Visuals** style field in the **Properties** panel and in the right-click context menu of the **LookFrom** widget.

Click the desired visual style in the list.

The default options are:

- 2D Wireframe
- Conceptual
- Hidden
- Modeling
- Realistic
- Shaded
- Shaded with edges
- Shades of gray
- Sketchy
- Wireframe
- X-Ray

- BIM (BIM templates only)
- Mechanical (Mechanical templates only)

7.13 Model space and paper space

When you start a drawing session, your initial working area is called **Model Space**. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the **Model** tab.

In general, model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The VPORTS command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called **Paper Space**. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one **Layout** in which you can have one or more **Layout Viewports**. Such layout viewports are to be considered as a view window on the entities in model space. Layout viewports can be placed anywhere in a paper space layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The MVIEW command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

7.13.1 Switching between model space and paper space

- To switch between Model Space and Paper Space layouts, use the **Model** and **Layout** tabs at the bottom of the graphic window.
- When working in a layout, you can access model space through a paperspace viewport:
  - Double-click inside a viewport to access model space. Double-click outside a viewport to go back to paper space.
  - Click the **Paper/Model** field in the **Status** bar to toggle between the workspaces. This method has to be used when it is not possible to click outside the viewport.
  - Use the **MSPACE** and **PSPACE** commands. These commands are available in the right-click context menu of the **Paper/Model** field in the **Status** bar.

7.13.2 Using the layout tabs

- The name of the active tab is displayed in bold.
- Click the navigation buttons at the left hand side to switch.
- Drag layout tabs to change their order.
- Click the button between the navigation buttons and the **Model** tab to open the **Layout Manager** dialog box.
- Right-click a layout tab to display a context menu.
  - **New**: Create a new layout.
  - **From Template**: Imports a layout from another drawing.
  - **Rename**: Renames the layout.
  - **Copy**: Inserts a copy of the layout.
  - **Move Right / Move Left**: Moves the layout.
  - **Export...**: Exports the layout to a new drawing.

7.14 Model space viewports

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active, click in a viewport to make it the current viewport. You can print the current viewport only.

The coordinate system (WCS or UCS), grid display and snap can be set per viewport.

The VPORTS command creates and manages model space viewports.

7.14.1 Working with multiple viewports

- Click in a viewport to make it current.
- The border of the current viewport is highlighted.
- The cursor is shown in the current viewport only.
- You can start a command in one viewport, and finish the operation in a different viewport.

7.14.2 Joining adjacent viewports

You can join viewports which share an edge of equal length.

1 Launch the **VPORTS**command.

2 Select the **Join** command option.

You are prompted: Select inside dominant viewport <Current>:

3 Click in the dominant viewport or press Enter to choose the current viewport.

4 The border of the selected viewport is highlighted.

You are prompted: Select inside viewport to join.

5 Click the viewport to join again to confirm.

6 The two selected viewports are joined.

7.15 Paper space viewports

The **ALIGNSPACE** command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

In a layout, you can create multiple viewports, each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

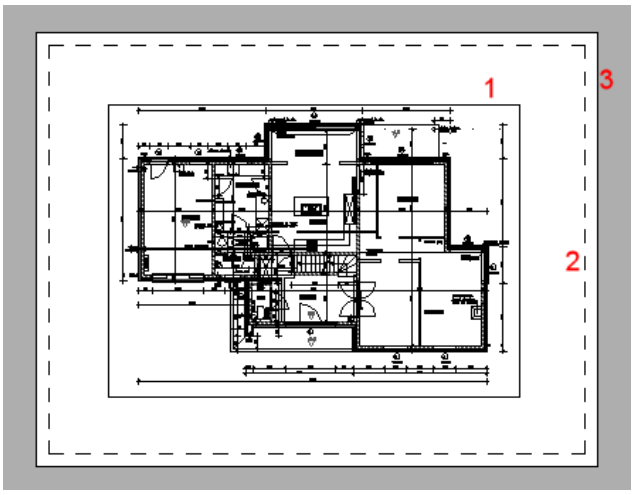
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space in layout, click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport.

To preserve the scaling of a viewport, you can lock the display. It is no longer possible to zoom or pan in a locked viewport.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

7.15.1 Displaying the paper sheet and the printable area



1 Viewport.

2 Printable area.

3 The display of the printable area is controlled through the DISPPAPERMARGINS system variable.

4 Paper sheet.

5 The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

7.16 Creating paper space viewports

The MVIEW and VPORT commands create one or more viewports in layout tabs.

- A new viewport shows all entities on the layers which are currently visible in model space.
- It is recommended to use a dedicated layer for viewport entities. If you don't want the viewport outlines to be printed, set the **Do Not Plot** property of the viewport layer.

7.16.1 Creating an array of layout viewports

The MVSETUP command (short for "multiple viewports setup") creates multiple paper space viewports, and then aligns, rotates and scales them.

7.16.2 Defining the viewport properties

1 Select the viewport.

1 The viewport properties appear in the **Properties** panel.

2 Click the settings field of a property to modify.

Name	Description
Center	XYZ coordinates of the center point of the viewport with respect to the bottom left corner of the printable area of the layout.
Height (*)	Height of the viewport in **drawing units**.
Width (*)	Width of the viewport in **drawing units**.
On	Controls the display of the content of the viewport.
Clipped	Indicates whether the viewport is clipped or not.
Display locked	Locks the scaling of the viewport content to preserve the scaling factor. The VPLOCK field in the **Status** bar displays the Display Locked status of the selected viewport(s). Click the field to toggle the **Display Locked** property of the selection.

Name	Description
Scale	The SCALELISTEDIT command allows you to edit the available scales in the current drawing. **Annotation scale**: Sets the annotation scale for the viewport. Editing the Annotation scale automatically sets the Standard scale. If the ANNOALLVISIBLE system variable is OFF, annotative entities which do not support the annotation scale are not displayed. **Standard scale**: By default, the Standard scale matches the Annotation scale, but you can adjust it manually. The **Standard scale** field reads '**Custom**' if the value in the **Custom scale** field is not available in the drawings scale list. **Custom scale**: Displays the viewport scale factor. Allows you to set the viewport scale to a scale which is not available in the drawings scale list.
UCS per viewport	If **Yes**, lets you define a UCS for this viewport.
Visual Style	Sets the visual style for the viewport. This option only applies to 3D models.
Shade Plot	Allows you to adjust the plotted shading from the visual style. Default is '**As Displayed**'.
Default Lighting	Allows you to switch off the lights in the model and use the default lighting.

7.17 Editing paper space viewports

7.17.1 Setting the layer properties in a paper space viewport

The layer visibility can be controlled per paper space viewport. When working in a paper space layout, a number of VP* (Viewport) properties are available, which affect the current viewport only and can be adjusted from the corresponding default properties:

- VP Freeze
- VP Color
- VP Linetype
- VP Lineweight
- VP Transparency
- VP Plot

1 Double-click inside the paper space viewport.

2 Launch the EXPLAYERS command.

3 The **Drawing Explorer / Layers** dialog box appears.

4 (option) Click the icon in the **VP Freeze** column to toggle the display of a layer.

5 (option) Click the icon in the **VP Color** column, then select a color in the **Select Color** dialog box.

6 (option) Click the **VP Linetype** column, then select a linetype in the drop-down list.

7 (option) Click the **VP Lineweight** column, then select a lineweight in the drop-down list.

8 (option) Click the **VP Transparency** column, then select a transparency value in the drop-down list.

9 (option) Click the **VP Plot** column, then select a plot style in the **Select Plot Style** dialog box. This option applies to STB (Style Tables) drawings only.

Nota:

- When the **Indicate layers in use** option in the **View** menu of the **Drawing Explorer** dialog box is set, the icon in the **Current** column indicates whether VP property overrides exist.
- Use the **Viewports / Vpfreeze** option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

7.17.2 Clipping a viewport

The **VPCLIP** command (short for "viewport clipping") clips viewports in layouts.

Only circles and closed polylines can be used to create a clipped viewport. Therefore, use the **Create polyline representations of an ellipse** option of the PELLIPS system variable when creating an elliptical viewport.

7.17.3 Removing the clipping boundary of a viewport

The **Delete** option of the VPCLIP command removes the clipping boundary.

7.17.4 Rotating the display in a paper space viewport

The display of a paper space viewport can be rotated only if the display of the viewport is not locked.

The **Align** > **Rotate** option of the MVSETUP command defines the angle of the content of a viewport relative to the X-axis of the world coordinate system (WCS).

If the VPROTATEASSOC system variable is ON, the ROTATE and MANIPULATE commands rotate the content of a viewport over a specified angle. The viewport is clipped by a rectangle of the dimensions of the original viewport. Use the **Delete** option of the VPCLIP command to remove the clipping.

In order to undo the rotation:

- Set the rotation angle = 0 (zero) in the **Align** > **Rotate** option of the MVSETUP command.
- Double-click inside the viewport, then launch the PLAN command and press Enter twice (= set plan view of the WCS).

7.17.5 Aligning viewports

1 Launch the MVSETUP command.

2 Choose **Align** in the prompt menu.

You are prompted: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:

3 (option) Choose the **Horizontal** alignment option.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved vertically to align the two points horizontally.

4 (option) Choose the **Vertical alignment option**.

You are prompted: Specify basepoint:

- 1. Pick a point in the reference viewport.
- 2. **You are prompted**: Specify point in viewport to be panned:
- 3. Pick a point in the viewport to be aligned.
- 4. The display in the viewport is moved horizontally to align the two points vertically.

7.17.6 Aligning model space and paper space points

The ALIGNSPACE command adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space.

Aligning 1 point

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Right-click or press Enter.

5 **You are prompted**: Alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Activate the viewport which will be used for alignment.

8 Click inside the viewport you want to align.

9 Right-click or press Enter to confirm.

10 The display in the selected viewport is moved to make the two points coincident.

Aligning 2 points

1 Launch the ALIGNSPACE command.

1 **You are prompted**: First alignment point in Model space:

2 Click inside a layout viewport, then pick a point.

3 **You are prompted**: Second alignment point in Model space or <Return> for none:

4 Pick a second point in the layout viewport.

5 **You are prompted**: First alignment point in Paper space:

6 Pick a point in the layout.

7 **You are prompted**: Second alignment point in Paper space:

8 Pick a second point in the layout.

9 **Prompts you:** Activate the viewport which will be used for alignment.

10 Click inside the viewport you want to align.

11 Right-click or press Enter to confirm.

12 The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

7.18 Layouts

In BricsCAD, you can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the paper size, print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. If the CREATEVIEWPORTS system variable is ON, a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts, you can either define them yourself or you can import layouts from another drawing: template file (.dwt), drawing file (.dwg) or drawing interchange file (.dxf).

Viewports can display all or part of the drawing's model space entities.

The CACHELAYOUT system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced.

7.18.1 Creating a new layout

1 Launch the LAYOUT command, then select the **New** option.

2 **You are prompted**: Enter a layout name. <Layout1>:

3 Type a unique name for the layout, then press Enter.

4 The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

5 Right-click or press Enter to accept the default name (e.g. Layout1).

6 The new layout tab is added.

Nota: Right-click a layout tab, then choose **New** in the context menu or click the plus (+) button after the last layout tab to add a new layout using a default name. See step 3 in the above procedure to rename the layout.

7.18.2 Copying layouts

1 Right-click the layout tab you want to copy.

1 A context menu appears.

2 Choose **Copy** in the context menu.

You are prompted: Enter new layout name:

3 Type a new unique name in the Command line, then press Enter.

4 A copy of the selected layout is added.

7.18.3 Arranging the layout tabs

Do one of the following:

- Drag the layout tab to the desired position.
- Right-click the layout tab, then choose a **Move** option in the context menu.

7.18.4 Importing layouts

1 Right-click the **Model** tab or one of the layout tabs.

A context menu appears.

2 Choose **From Template...** in the context menu.

The **Select Template From File** window opens.

3 Browse to the drawing that you want to import layouts from.

4 Click the **Open** button on the **Select Template From File** window.

The **Insert Layout(s)** window opens.

5 Select the layout(s) you want to import.

6 Hold down the Ctrl key to select multiple layouts.

7 Click the **OK** button.

The selected layouts are imported.

7.18.5 Renaming a layout

1 Double-click the layout tab you want to rename.

1 The **Rename Layout** dialog box opens.

2 Type a new unique name in the **Name** field.

3 Click the **OK** button.

7.18.6 Deleting a layout

1 Right-click the layout tab you want to delete.

1 A context menu appears.

2 Choose **Delete** in the context menu.

3 An **Alert** box appears.

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

7.18.7 Using the Layout Manager

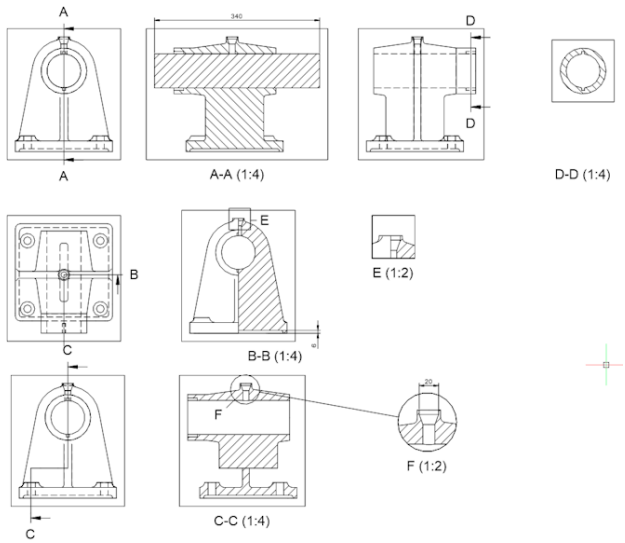
1 Do one of the following:
   - Launch the LAYOUTMANAGER command.
   - Click the button between the navigation buttons and the **Model** tab.

2 In the **Layout Manager** dialog box you can:
   - Create layouts.
   - Copy layouts.
   - Delete layouts.
   - Change the order of the layout tabs.
   - Publish layouts.

7.19 Generated drawing views

The Generated Drawing Views functionality allows you to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FLATSHOT command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the VIEWUPDATE command.



The VIEWBASE command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The VIEWSECTION command creates a cross section view based on a standard drawing view generated by the VIEWBASE command in a paper space layout.

The VIEWSECTIONSTYLE command allows you to modify the appearance of the section views created by the VIEWSECTION command.

The VIEWPROJ command generates additional projected views from an existing generated drawing view.

The VIEWDETAIL command creates a detail view of a portion of a standard generated drawing, at a larger scale.

The VIEWDETAILSTYLE command allows you to modify the appearance of the detail views created by the VIEWDETAIL command.

The VIEWUPDATE command updates a selection of drawing views obtained by VIEWBASE and VIEWSECTION when VIEWUPDATEAUTO = 0.

The VIEWEXPORT command exports the content of drawing views obtained by VIEWBASE and VIEWSECTION to the Model Space of the drawing. This command can be used in Paper Space only.

The VIEWEDIT command allows you to change the scale and the hidden line visibility of drawing views.

Tools to generate drawing views of your 3D solid model are available on:

- The **Drawing Views** toolbar, which is a flyout of the **View** toolbar.
- The **Section Types** and **Detail Section Types** toolbars.
- The **View | Drawing Views** menu.
- The **Drawing Views** panel of the **Annotate Ribbon** tab (Mechanical Workspace).

7.19.1 Standard drawing views

Standard views include multiview orthographic and isometric projections.

In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object.

Isometric projection is a method to represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees.

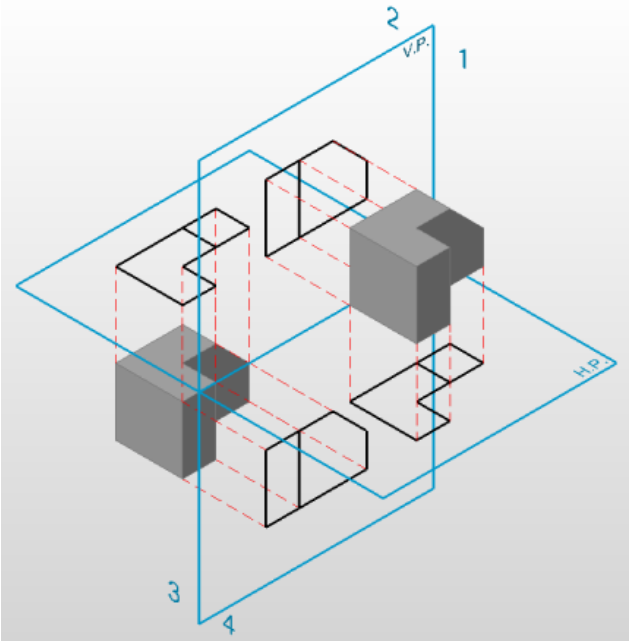
To generate the standard views of your 3D solid model, run the ViewBase command from Model Space.

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

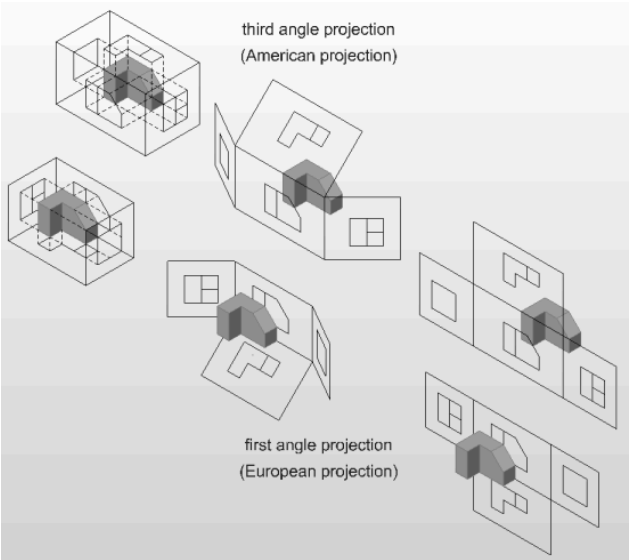
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections.

Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide 3D space in four quarters.

- In the first angle projection system, the model is placed in the first quarter space (first angle).
- In the third angle projection system, the model is placed in the third quarter (third angle).



With multiview orthographic projections, up to six views of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other, according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto the 6 faces of a box around the model. When unfolding the box, all views are organized in a 2D drawing.



The projection type is set by the **Projection Type** option of the VIEWBASE command or by editing the PROJECTIONTYPE system variable:

- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE system variable is saved in the drawing.

Generated drawing view lines are created on default layers:

- BM_Isometric_Hidden;
- BM_Isometric_Visible;
- BM_Ortho_Hidden;
- BM_Ortho_Visible.

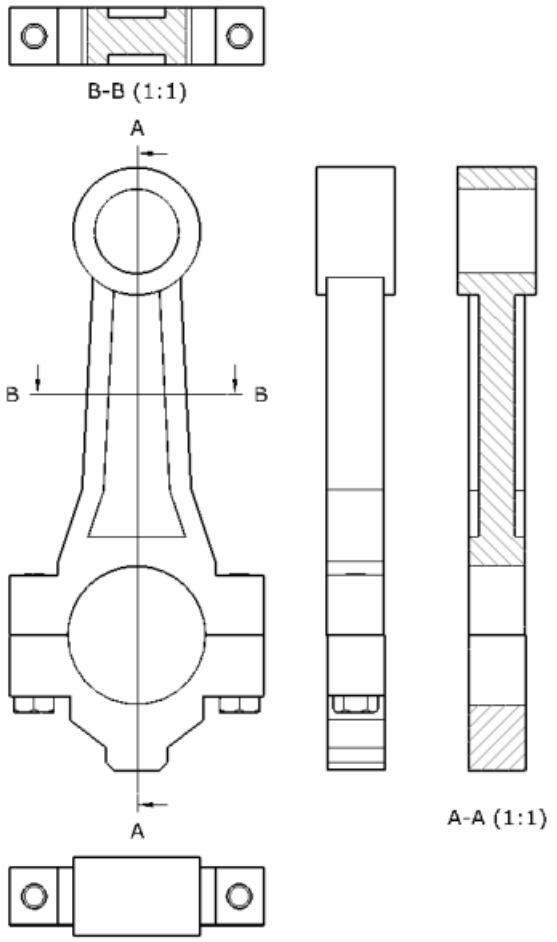
The display of the BM_Isometric_Hidden layer is turned off by default.

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

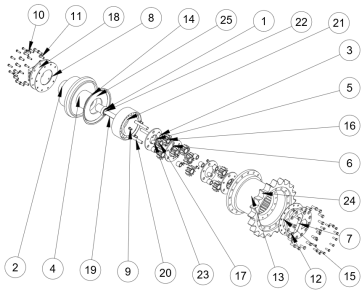
7.19.2 Generating section views

A section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections.

To generate section views of a 3D solid model, run the VIEWSECTION command from the paper space layout where the standard drawing views are located.



7.19.3 Special views



Bill of Materials Drive_Assembly		
No.	Component	Quantity
1	BBRG-03	2
2	BBRG-03	1
3	BBRG-60	9
4	Body	1
5	Carrier_1	2
6	Carrier_2	2
7	Cover	1
8	Cover_Interal_Seal	1
9	DIN 933	12
10	DIN 933	24
11	DIN 933	16
12	DIN 933	16
13	Drum	1
14	External_Medium_Seal	1
15	External_Seal	1
16	Gear_Planet	9
17	Gear_Sun	3
18	Internal_Cover	1
19	Key	1
20	Pin	3
21	RLR BRG-32	1
22	Shaft	1
23	Sleeve	6
24	Sprocket	1
25	Washer	1

It is possible to generate drawing views for a selected exploded representation. Normally, an orthogonal projection is used for that; however, BricsCAD allows you to create any standard projection for exploded representations. Sections and detailed views are also supported.

To create a view for an exploded representation, run the VIEWBASE command with the **Special** views option, select the **Exploded view** option and then select the required exploded representation from the dialog box.

Generated views for exploded representation can be associative with the original model, so annotations and balloons can be placed. To do that, the views should be created or updated with GENERATEASSOCVIEWS = ON.

7.19.4 Updating drawing views

By default BricsCAD checks whether the source 3D solid model or the associated exploded representation was modified and automatically recalculates the drawing when:

- opening a paper space layout containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- running the VIEWUPDATE command.
- running the BMUPDATE command.
- 3D geometry.

Nota:

- Hidden line removal calculation might take some time for complex 3D models. In such cases, it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.
- If GENERATEASSOCVIEWS = ON, associative dimensions will be updated according to the modified 3D geometry.

7.19.5 Exporting views to model space

The associativity mechanism of drawing views in a paper space layout allows for limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry, the ViewExport command allows you to move or copy drawing views to

model space or to a separate drawing. Exported drawing views lose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc.

7.19.6 Customizing drawing views

The first time the VIEWBASE command is launched, four new layers are created:

- BM_Ortho_Hidden: hidden lines in drawing views;
- BM_Ortho_Visible: visible lines in drawing views;
- BM_Tangent_Hidden: hidden tangent lines in drawing views;
- BM_Tangent_Visible: visible tangent lines in drawing views.

Use the VIEWEDIT command to change the visibility of these layers for all views or for a selection of views only.

- Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.
- Use ViewSectionStyle command to adjust different properties of section views such as style, color, and height of identifier text;
- start and end symbols, their colors, size and direction of arrows scale, patter, color, and angle of hatches;
- color, type, and weight of a line representing a cutting plane.

Use the VIEWDETAILSTYLE command to adjust different properties of details views such as:

- style, color, height of identified text, and the type of identifier
- symbol, and its color and size
- color, type, and weight of detail view boundary

8. Disegno 2D

8.1 Creare entità

In un disegno è possibile creare una varietà di differenti tipologie di entità. Le entità del disegno possono essere veramente semplice così come le linee, cerchi, archi, punti e raggi oppure complesse come le polilinee, spline o piani. Oltre alle entità 2D, in BricsCAD è possibile creare entità superficie 3D e solidi 3D.

Per completare ulteriormente il proprio disegno, BricsCAD dispone di una serie di strumenti per aggiungere testo, tratteggi e quote.

8.1.1 Procedura generale per creare nuove entità

Le nuove entità vengono create utilizzando le impostazioni di creazione delle entità correnti sul layer corrente nel piano XY del sistema di coordinate corrente, per impostazione predefinita questo è il sistema WCS (World Coordinate System).

1 (opzionale) Impostare il layer corrente.

2 (opzionale) Regola le proprietà di creazione dell'entità nella sezione **Generale** del pannello **Proprietà**.

3 Avviare il comando.

4 Eseguire il comando in base alle istruzioni dei comandi.

8.2 Impostazioni creazione entità

Le impostazioni e le preferenze predefinite per creare entità di disegno sono specificate da variabili di sistema.

È possibile visualizzare e modificare queste variabili:

- Nella finestra di dialogo Impostazioni. Espandere **Disegno** > **Disegnare** > **Creazione entità**. Nella parte superiore dell'elenco vengono visualizzate le proprietà generali come colore, tipolinea e layer. Di seguito vengono visualizzate le impostazioni per le proprietà specifiche dell'entità.
- Nel nodo **Generale** del pannello Proprietà quando non viene selezionato nulla.

Le impostazioni **Generali** sono:

- **Colore**: Imposta il colore corrente. Predefinito = DaLayer.
- **Layer**: Imposta il layer corrente.
- **Tipolinea**: Imposta il tipolinea corrente. Predefinito = DaLayer.
- **Scala tipo linea**: imposta la scala del tipolinea corrente. Predefinito = 1.
- **Spessorelinea**: Imposta lo spessorelinea corrente. Predefinito = DaLayer.
- **Trasparenza**: imposta la trasparenza corrente. Predefinito = DaLayer.
- **Elevazione**: specifica l'altezza (valore Z) sopra il piano XY del sistema di coordinate corrente (WCS o UCS). Predefinito = 0.

Nota: La maggior parte delle impostazioni di creazione entità vengono salvate nel disegno, quindi vengono applicate solo al disegno corrente. Se si desidera regolare queste impostazioni per tutti i disegni futuri, modificare i template dei disegni e regolare le impostazioni. I template dei disegno sono salvati

nella cartella memorizzata nell'impostazione TEMPLATEPATH. La variabile di sistema BASEFILE specifica il percorso e il nome del file del template predefinito.

8.3 Lavorare con i layer

È possibile creare e gestire layer nella sezione **Layer** della finestra di dialogo Esplora Disegno (comando EXPLAYERS) o nel pannello Layer (comando LAYERSPANELOPEN).

La finestra di dialogo Esplora Disegno non può essere aperta mentre si utilizzano altri comandi, mentre il pannello Layer può farlo. È possibile spostarlo e ridimensionarlo come desiderato oppure spostarlo su un secondo monitor.

8.3.1 Layer corrente

Le nuove entità vengono disegnate nel layer corrente. Per disegnare nuove entità su un layer diverso, occorre prima renderlo il layer corrente.

- Il layer corrente viene salvato tramite la variabile di sistema CLAYER.
- La variabile di sistema CENTERLAYER* controlla il layer in cui vengono posizionati i marcatori centro e le linee d'asse creati dal comando DIMCENTRO.
- La variabile di sistema DIMLAYER* controlla il layer per le entità di quota create dai comandi per la quotatura.
- Le entità create dal comando INTERFERENZA vengono create nel layer che è specificato dalla variabile di sistema INTERFERELAYER.

* Il valore predefinito è <Usa Corrente>

Per impostare il layer corrente, eseguire una delle seguenti operazioni:

- Senza alcuna entità selezionata, selezionare **Layer** nel pannello Proprietà, quindi scegliere il nuovo layer corrente nell'elenco a discesa.
- Sulla barra degli strumenti **Proprietà Entità**, cliccare sulla freccia giù accanto al campo **Layer**, quindi selezionare un layer nell'elenco a discesa.
- Nel pannello **Layer** della barra multifunzione, cliccare sulla freccia giù e scegliete un layer nell'elenco a discesa.
- Cliccare il pulsante destro del mouse sul campo **Layer** nella Barra di Stato, quindi scegliere un layer nel menu contestuale.

Nota:

- È possibile spegnere la visualizzazione del layer corrente ( ). Di conseguenza, tutto il contenuto sarà invisibile fino a quando non si riaccende la visualizzazione del layer corrente ( ). Un messaggio di avviso viene visualizzato se si tenta di spegnere il layer corrente.
- Non è possibile congelare ( ) il layer corrente.

8.3.2 Stati Layer

Comando STATOLAY.

Gli Stati Layer sono elementi memorizzati in un disegno.

Gli Stati Layer consentono di salvare e ripristinare le configurazioni delle proprietà dei layer. Si potrebbe desiderare di visualizzare un layer in blu ed altre volte in verde, oppure si ha necessità che alcuni layer siano spenti o congelati o bloccati durante la modifica di una parte specifica di un disegno. Dopo aver speso un sacco di tempo a regolare tutte le impostazioni dei layer si possono usare gli stati layer per salvarle. È possibile salvare più stati layer in ogni disegno. I Stati Layer possono essere copiati tra i disegni ed è possibile, inoltre, salvare uno stato layer in un file esterno, che può essere successivamente importato in un altro disegno.

8.4 Tipo linea entità

È possibile utilizzare diversi tipilinea per differenziare lo scopo di una linea da un'altra. Un tipolinea è costituito da uno schema ripetuto di punti, trattini o spazi vuoti. I tipilinea determinano l'aspetto delle entità sia sullo schermo che al momento della stampa.

Per impostazione predefinita, ogni disegno ha almeno tre tipilinea: Continua, DaLayer e DaBlocco. Il disegno può contenere anche un numero illimitato di tipilinea denominati.

Le definizioni dei tipilinea vengono salvate nel disegno. I nuovi tipilinea vengono importati da altri disegni o caricati da un file tipolinea (*.lin). Il file di origine dei tipilinea, utilizzato per impostazione predefinita è controllato dalla variabile di sistema MEASUREMENT:

- MEASUREMENT = ON utilizza **Iso.lin**: Le definizioni dei tipilinea sono espresse in mm.
- MEASUREMENT = OFF: **Default.lin**: Le definizioni dei tipilinea sono espresse in pollici.

Le nuove entità vengono disegnate utilizzando il tipolinea corrente impostato dalla variabile di sistema CELTYPE.

Le opzioni per impostare i tipilinea sono:

- DaLayer: le entità adottano il tipilinea del loro layer. Questo consente di modificare il tipolinea di tutte le entità regolando il tipolinea del layer.
- DaBlocco: Entità disegnate come linee continue finché non vengono incluse in un blocco. Le entità ereditano quindi l'impostazione del tipilinea dell'entità blocco quando si inserisce il blocco in un disegno.
- Tipolinea denominato: continuo o uno dei tipilinea salvati nel disegno.

Scala tipo linea

I tipilinea non continui sono costituiti da linee, spazi e punti riuniti in una varietà di modelli. Nei tipilinea complessi è possibile includere anche testo o forme.

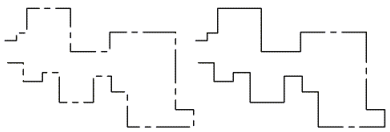
Nella definizione del tipolinea, le lunghezze delle linee e degli spazi vuoti sono definite in pollici (default.lin) o mm (iso.lin). Se le unità di disegno sono diverse da pollici o millimetri, i tipilinea vengono scalati di conseguenza. La scala del tipolinea è controllata dalla variabile di sistema LTSCALE, che deve essere impostata rispetto le unità di disegno (UD):

Unità di Disegno	LTSCALE
Imperiale (MEASUREMENT = OFF)	
Pollice	1
Piede	1/12
Metrico (MEASUREMENT = ON)	
mm	1
cm	0.1
m	0.001

BricsCAD metterà sempre un trattino all'inizio e alla fine di una linea, polilinea, arco o spline. Anche il modello del tipolinea risulta ben centrato, in modo che entrambe le estremità dell'entità abbiano lo stesso aspetto.

Generazione Polilinea

I tipilinea vengono normalmente generati da vertice a vertice. Le polilinee i cui vertici sono molto vicini l'uno all'altro, possono essere rappresentate come una linea continua se il tipolinea non si adatta tra i due vertici successivi. La variabile di sistema PLINEGEN risolve questo problema: quando è impostata su 1, il tipolinea viene disegnato da un'estremità della polilinea fino all'altra estremità invece che da un vertice all'altro.



La stessa polilinea disegnata con PLINEGEN attivo=on (sinistra) e non attivo=off (destra).

Scala Tipo di linea spazio carta

La variabile di sistema PSLTSCALE controlla il ridimensionamento del tipolinea nei layout dello spazio carta.

Se è ON, il ridimensionamento del tipolinea è indipendente dalla scala della finestra della vista.

In generale, è meglio impostare PSLTSCALE su ON. Notare che questa variabile può essere impostata per ogni layout.

Scala tipi di linea spazio modello

La variabile di sistema MSLTSCALE controlla il ridimensionamento del tipolinea nello spazio modello.

Se attivo, la scala annotazione corrente (variabile di sistema CANNOSCALE) viene applicata ai tipolinea nello spazio modello.

E' preferibile impostare MSLTSCALE=OFF. Potrebbe essere utile attivarlo quando si stampa dallo spazio modello.

8.4.1 Impostare il tipolinea corrente

Eseguire una delle seguenti operazioni:

- Nel pannello Proprietà, selezionare **Tipolinea**.

  Assicurati che non sia selezionata alcuna entità.

  Scegliere un tipolinea dall'elenco a discesa.
- Sulla barra degli strumenti **Proprietà Entità**, cliccare sulla freccia giù accanto al campo **Controllo Tipo di Linea**, quindi scegliere un tipolinea.
- Cliccare il pulsante destro del mouse sul campo **Tipo di Linea Corrente** nella Barra di Stato, quindi scegliere Proprietà nel menu contestuale.

8.4.2 Aggiungere un nuovo tipolinea

1 Eseguire una delle seguenti operazioni:

   - Avviare il comando TLINEA, quindi scegliere l'opzione **Nuovo**.
   - Selezionare **Tipolinea** nel pannello Proprietà, quindi cliccare sulla freccia verso il basso e scegliere **Carica...**

   Viene visualizzata la finestra di dialogo Carica Tipi linea.

2 (opzionale) Cliccare sul pulsante **File...** per caricare un diverso file sorgente di tipolinea.

3 Selezionare uno o più tipilinea nella finestra di dialogo Carica Tipi linea.

   Tenere premuto il tasto CTRL per aggiungere o rimuovere tipi di linea.

4 Cliccare sul pulsante **OK**.

8.4.3 Impostare la scala del tipolinea dell'entità

1 Nel pannello Proprietà, selezionate **Scala Tipo linea**.

   Assicurarsi che non sia selezionata alcuna entità.

2 Digitare un nuovo valore nel campo scala del tipolinea.

Il valore della scala del tipolinea viene salvato tramite la variabile di sistema CELTSCALE. E' preferibile impostare questa variabile su 1, ad eccezione dei tipi di linea speciali, come BATTING o GAS_LINE, che molto probabilmente devono essere ridimensionate.

Nota: La scala del tipolinea delle entità si applica solo alle nuove entità.

8.5 Colore entità

Il colore determina il modo in cui un'entità viene visualizzata sullo schermo e il modo in cui viene stampata. Le entità vengono create nel colore corrente, come definito dalla variabile di sistema CECOLOR.

E' possibile scegliere tra 255 **colori indice** o definire un **true color**.

8.5.1 Colore indice

Il colore indice è la specificazione del colore di un pixel sullo schermo utilizzando un valore di colore ad 8 bit, permettendo di avere fino a 256 colori possibili.

Ogni colore indice possiede un unico numero da 1 a 255. Sette di questi colori possono inoltre avere un nome: rosso (1), giallo (2), verde (3), ciano (4), blu (5), magenta (6) e bianco/nero (7). Il colore 7 viene visualizzato bianco su uno schermo a sfondo nero e nero su uno schermo a sfondo bianco. Il colore 7 viene stampato in nero.

Le due proprietà di colore aggiuntive sono DaLayer e DaBlocco. Queste proprietà del colore fanno in modo che un'entità assuma il colore del layer o del blocco di cui fanno parte. DaLayer è il colore numero 256, e DaBlocco è il colore numero 0. In tutti i comandi dove si vuole utilizzare un colore, è possibile indicare DaLayer e DaBlocco così come i numeri 256 e 0, rispettivamente.

Le opzioni per impostare il colore sono:

- DaLayer: le entità adottano il colore del loro layer. Questo consente di modificare il colore di tutte le entità regolando il colore del layer.
- DaBlocco: le entità vengono disegnate con il colore dell'indice 7 (nero o bianco, a seconda del colore di sfondo dello schermo). Se incluse in una definizione di blocco, tali entità adottano il colore del blocco quando si inserisce il blocco in un disegno.
- Colore Indice
- True Color

Nota: Se le entità che utilizzano un colore indice verranno stampate con questo colore, dipende dalla **Tabella Stili di Stampa dipendenti da colore** (CTB) oppure dalla **Tabella Stile di Stampa dipendenti da nome** (STB) utilizzata per la stampa. Solo se l'impostazione **Colore** nel file di definizione CTB o STB è impostata su **Usa colore oggetto**, il colore stampato corrisponde al colore dell'entità.

8.5.2 True Color

I colori true color sono la specificazione del colore di un pixel sul display di uno schermo che utilizza un valore di colore a 24 bit che permette di avere più di 16 777 216 possibili colori.

Il numero di bit utilizzati per definire il colore dell'ombra di un pixel è la sua profondità in bit. I True Color sono anche conosciuti come colori a 24-bit. Alcuni nuovi sistemi di visualizzazione a colori offrono una modalità a 32-bit. Il byte aggiuntivo, denominato canale alpha, è utilizzato per il controllo delle informazioni degli effetti speciali.

I true color utilizzano una definizione di colore **RGB**(**R**ed,**G**reen,**Bl**ue). Ognuno di questi parametri ha un intervallo da 0 a 255. La definizione RGB per il nero è (0.0.0), la definizione RGB per il bianco è (255.255.255).

8.5.3 Impostazione del colore dell'entità corrente

1 Eseguire una delle seguenti operazioni:

- Avviare il comando COLORE.
- Cliccare sull'icona **Seleziona Colore** ( ) della barra degli strumenti Impostazioni.
- Nel pannello Proprietà, selezionare **Colore**, quindi scegliere **Seleziona colore** nell'elenco a discesa.
- Doppio clic sul **Campo Colore** nella Barra di Stato.

Viene visualizzata la finestra di dialogo Seleziona Colore.

2 (opzionale) Cliccare sulla scheda **Colore indice**, quindi eseguire una delle seguenti operazioni:

- Cliccare su uno dei quadrati colorati.
- Digitare il numero del colore nel campo **Colore**.

3 (opzionale) Cliccare sulla scheda **True Color**, quindi eseguire una delle seguenti operazioni:

- Cliccare sul pannello **Selezione Colore**.
- Impostare i parametri del colore nei campi **Parametri Colore**.

4 Cliccare il pulsante **OK** per confermare.

8.6 Spessorelinea

Gli spessori di linea determinano lo spessore con cui le entità appaiono sullo schermo e/o nella stampa.

Lo spessore di linea può essere impostato DaLayer, DaBlocco, Predefinito o su un valore specifico.

Gli spessori di linea sono definiti in millimetri o pollici, a seconda della variabile di sistema LWUNITS.

Non è possibile assegnare spessori di linea a piani, punti, caratteri TrueType ed immagini raster.

Le opzioni per impostare la Trasparenza sono:

- DaLayer: lo spessore di linea è determinato dalla proprietà Spessorelinea del layer corrente. La modifica della proprietà Spessorelinea di un layer avrà effetto su tutte le entità che hanno il loro Spessorelinea impostato DaLayer e si trovano su questo layer.
- DaBlocco: le entità create utilizzando lo spessore di linea DaBlocco, ottengono lo spessore di linea predefinito fino a quando non vengono incluse in una definizione di blocco. Le entità ereditano quindi lo spessorelinea del blocco quando si inserisce il disegno in un blocco.
- Predefinito: lo spessore di linea predefinito viene salvato tramite la variabile di sistema LWDEFAULT. La modifica dello spessore di linea predefinito influisce su tutte le entità dell'intero disegno che hanno lo spessore di linea impostato su Predefinito.

8.6.1 Variabili di sistema

La variabile di sistema CELWEIGHT definisce lo spessore di linea per le nuove entità.

La variabile di sistema LWDEFAULT imposta il valore dello spessore di linea predefinito.

La variabile di sistema LWDISPLAY controlla la visualizzazione degli spessori di linea. Cliccare sul campo SLN nella Barra di Stato per attivare/disattivare la visualizzazione degli spessori.

La variabile di sistema LWDISPLAYSCALE controlla la scala di visualizzazione degli spessori di linea nella scheda Modello. Sono accettati valori compresi tra 0 e 20.

8.6.2 Stampa spessorelinee

Le entità che utilizzano un colore indice utilizzeranno lo spessore di linea quando vengono stampate solo se la **Tabella Stili di Stampa dipendente da Colori** (CTB) o lo **Stile di Stampa dipendete da Nome** (STB) è definito come **Usa lo spessorelinea dell'oggetto**. In caso contrario, lo spessore della linea è definito dal colore dell'entità o dallo stile di stampa con nome.

Le entità che utilizzano un colore truecolor utilizzano sempre lo spessorelinea dell'oggetto quando vengono stampate.

8.7 Modalità Riempimento

Il comando PIENO consente di attivare/disattivare la visualizzazione di aree riempite e tratteggiate e di specificare se riempire polilinee con larghezza, tracce, solidi 2D, tratteggi (incluso il riempimento solido). Attiva o disattiva il valore della variabile di sistema FILLMODE.

Quando FILLMODE è disattivato, tutte le entità riempite vengono visualizzate e stampate come contorni e si riduce il tempo necessario per visualizzare o stampare un disegno.

	Polilinea	Anello	Traccia	Piano (Solido)
Riempimento Attivo				
Riempimento Non Attivo				

8.7.1 Impostare la Modalità Riempimento

Eseguire una delle seguenti operazioni:

- Eseguire il comando PIENO.
- Impostare il valore della variabile di sistema FILLMODE.

Eseguire il comando RIGEN o RIGENT per rigenerare le finestre.

8.8 Trasparenza

La proprietà Trasparenza permette di controllare la trasparenza delle entità del disegno. È possibile impostare la trasparenza per una selezione di entità o per tutte le entità sullo stesso layer. La trasparenza predefinita per le nuove entità è controllata dalla variabile di sistema CETRANSPARENCY.

La trasparenza predefinita per i tratteggi ed i riempimenti sfumati è controllata dalla variabile di sistema HPTRANSPARENCY.

La Trasparenza può essere utilizzata per migliorare la visualizzazione dei disegni, ad esempio per attenuare dettagli meno importanti o entità che vengono utilizzate solo come riferimento.

Le opzioni per impostare la Trasparenza sono:

- DaLayer - applicare il valore della proprietà trasparenza del layer.
- DaBlocco: le entità che utilizzano la trasparenza DaBlocco ereditano la trasparenza del blocco quando si inserisce il blocco in un disegno.
- Valore: sono accettati valori compresi tra 0 (completamente opaco) e 90.

  Il livello di trasparenza è limitato al 90% per evitare confusione con i layer congelati o disattivati.

8.8.1 Stampare le trasparenze

Le trasparenze verranno stampate solo se è spuntata l'opzione Stampa Trasparenze sulle finestre di dialogo Stampa o Imposta Pagina.

8.9 Modificare le proprietà delle entità

Le proprietà dell'entità possono essere modificate nel pannello **Proprietà** oppure utilizzando la caratteristica **Proprietà Rapide**.

8.9.1 Modificare le entità nel Pannello delle Proprietà

BricsCAD offre un pannello **Proprietà** come la tavolozza **Proprietà** in AutoCAD. È possibile accedervi con il comando PROPRIETA (alias PR) e con altri metodi come il doppio clic su un oggetto.

Le proprietà visualizzate nel pannello dipendono dal tipo o dai tipi di entità selezionate. Quando vengono selezionate più entità, nella visualizzazione delle proprietà condivise il valore delle proprietà condivise che variano nel gruppo di selezione, viene visualizzato con ***Vari***.

A differenza di AutoCAD, il pannello **Proprietà** in BricsCAD mostra il **Gestore** (in sola lettura) dell'entità selezionata, che può essere utile da memorizzare e richiamare a scopo di programmazione.

8.9.2 Modificare le entità con le Proprietà Rapide

Quando le Proprietà Rapide sono attive (**PR** nella **Barra di Stato**), vicino al cursore vengono visualizzate alcune delle proprietà dell'entità. Cliccare su una proprietà per modificarla.

Quando il Quad è attivo, cliccare sulla barra del titolo del **Quad** per passare dalle Proprietà Rapide al menu del cursore Quad.

Cliccare il pulsante destro del mouse sul campo **PR** nella **Barra di Stato** e scegliere **Personalizza…** nel menu contestuale per aprire la finestra di dialogo **Personalizza / Proprietà**). Qui è possibile definire quali proprietà vengono visualizzate nelle **Proprietà Rapide** per ogni tipo di entità.

8.10 Disegnare entità lineari

I comandi per disegnare entità lineari sono:

	Disegna polilinee con coordinate x,y,z.
	Disegna archi.
	Crea una polilinea che forma un contorno attorno alle aree chiuse.
	Disegna cerchi.
	Disegna cerchi larghi da polilinee.
	Disegnare ellissi e archi ellittici.
	Disegna spirali 2D o eliche 3D.
	Disegna segmenti di linea.
	Disegna multilinee.
	Disegna linee e archi collegati.
	Disegna poligoni regolari come polilinee.
	Disegna semi-linee di costruzione.
	Disegna rettangoli e quadrati.
	Disegna nuvole di revisione.
	Disegna schizzi a mano libera.
	Disegna spline aperte e chiuse in 2D e 3D.
	Disegna segmenti di linea larghi.
	Disegna linea di costruzione.

8.11 Disegnare linee

Una linea è composta da due punti: un punto di partenza e un punto finale. Utilizzando il comando LINEA è possibile disegnare singole linee o una serie di linee collegate, ma ciascuna linea è considerata un'entità linea separata.

Lo strumento Linea è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

8.11.1 Procedura Generale

1 Avviare il comando LINEA.

Viene richiesto: Inizio della linea o [Continua] <Ultimo punto>:

2 Specificare il punto di partenza.

La linea viene visualizzata in modo dinamico.

Viene richiesto: Impostare il punto finale o [ANgolo/Lunghezza/Continua/Annulla]:

3 Specificate il punto finale.

La linea è stata creata. Una nuova linea viene visualizzata dinamicamente.

Viene richiesto: Impostare il punto finale o [ANgolo/Lunghezza/Continua/Annulla]:

4 Specificate il punto finale.

La linea è stata creata. Una nuova linea viene visualizzata dinamicamente.

Viene richiesto: Impostare il punto finale o [ANgolo/Lunghezza/Continua/CHiudi/Annulla]:

5 Eseguire una delle seguenti operazioni:

- Continuare a specificare i punti finali per ogni nuovo segmento di linea.
- Scegliere l'opzione **Chiudi** per connettersi al punto di partenza della prima linea.
- Premere Invio o cliccare il pulsante destro del mouse per interrompere.

8.11.2 Specificare l'angolo e la lunghezza

Le opzioni sono:

- Usare le opzioni Angolo e Lunghezza del comando.
- Usare le quote dinamiche (DYNDIVIS = 1 o 2).

  Premere il tasto TAB per passare tra i campi Angolo e Lunghezza.

8.11.3 Specificare l'angolo a un'altra entità lineare

Cliccare il pulsante destro del mouse sul campo UCSD nella barra di stato, quindi scegliere "UCSD su altre entità" per impostare l'opzione **Abilita per entità 2D** della variabile di sistema UCSDETECT.

1 Avviare il comando LINEA.

2 Passare il mouse sulla linea di riferimento o sul segmento di polilinea lineare.

Un UCS temporaneo si allinea con un segmento di linea o polilinea.

L'icona UCS viene visualizzata nel punto intermedio o punto finale più vicino dell'entità sotto il cursore.

3 Premere il tasto Maiusc per bloccare l'UCS temporaneo.

4 Specificare il punto di partenza della linea.

Gli angoli sono ora relativi alla linea o al segmento di polilinea lineare selezionato.

Nota: Il WCS viene ripristinato quando si conclude il comando LINEA.

8.12 Disegnare raggi

Un raggio è una linea che inizia in un punto e si estende all'infinito. Poiché i raggi si estendono all'infinito, non vengono calcolati come parte delle estensioni del disegno.

Il metodo predefinito per disegnare un raggio è quello di selezionare il punto iniziale del raggio, e quindi specificare la sua direzione.

Poiché vengono spesso utilizzate come riferimento per la creazione di altre entità, Raggi e Linee di Costruzione(XLINEA) vengono talvolta definite linee temporanee.

Lo strumento Raggio è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.

8.12.1 Procedura Generale

1 Avviare il comando RAGGIO.

Viene richiesto: Raggio infinito: inizio del raggio o [ORizzontale/Verticale/Angolo/Bisettrice/Parallelo]:

2 Specificare il punto di partenza del raggio.

Il raggio viene visualizzato dinamicamente.

3 Specificare la direzione del raggio.

Il raggio successivo viene visualizzato dinamicamente.

4 (opzionale) Indicare direzioni specifiche per disegnare una serie di raggi, iniziando dallo stesso punto.

5 Cliccare il pulsante destro del mouse per interrompere il disegno di raggi.

8.13 Disegnare Linee di Costruzione

Una linea infinita (XLINEA) è una linea che attraversa un dato punto, orientata a un angolo specificato e che si estende all'infinito in entrambe le direzioni. Poiché le linee di costruzione si estendono all'infinito, non vengono calcolate nelle estensioni del disegno

Il metodo predefinito per disegnare una linea di costruzione è selezionare un punto e quindi specificarne la direzione.

Poiché vengono spesso utilizzati come riferimento per la creazione di altre entità, Raggi e Linee di Costruzione vengono talvolta definite linee temporanee.

Lo strumento Linea di Costruzione è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.

8.13.1 Procedura Generale

1 Avviare il comando XLINEA.

Ti chiede:

Linea di costruzione: Specificare punto lungo la linea o [ORizzontale/Verticale/Angolo/Bisettrice/Parallelo]:

2 Specificare il punto iniziale della linea di costruzione.

La linea di costruzione viene visualizzata in modo dinamico.

3 Specificare la direzione della linea di costruzione.

La prossima linea di costruzione viene visualizzata dinamicamente.

4 (opzionale) Continuare a specificare le direzioni per disegnare una serie di linee di costruzione, a partire dallo stesso punto.

5 Cliccare il pulsante destro del mouse per interrompere l'operazione.

8.14 Disegnare polilinee

Una polilinea è una sequenza aperta o chiusa di linee e/o segmenti di arco connessi, che vengono considerati come un'unica entità. Ogni segmento di una polilinea può avere una larghezza costante o rastremata per tutta la lunghezza del segmento.

Il comando PLINEA crea polilinee aperte e chiuse mediante l'immissione dei punti.

I comandi CONTORNI e -CONTORNI creano polilinee chiuse da un'area racchiusa, definita da altre entità.

Il comando UNISCI crea polilinee da una serie di linee, polilinee e archi connessi.

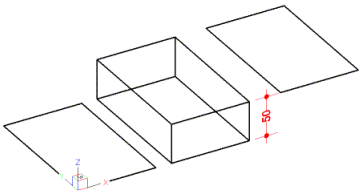
Lo strumento **Polilinea** è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

La proprietà **Spessore** specifica l'altezza di una polilinea. Se Spessore = 0 (zero) non c'è differenza visiva tra polilinee e una serie di linee e/o archi.

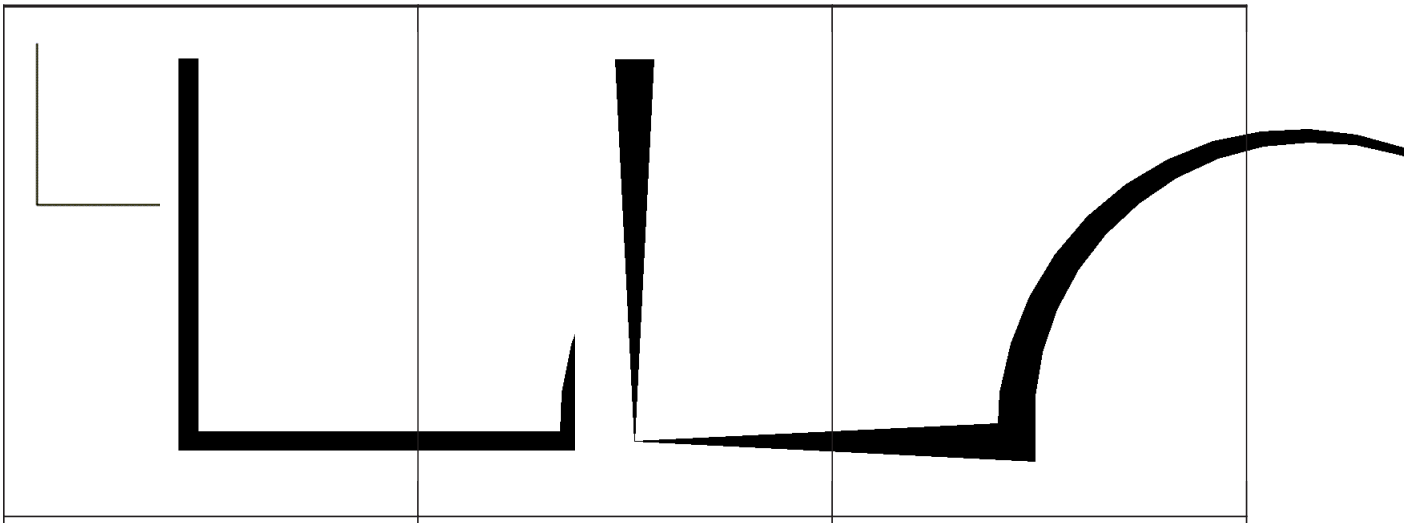
Lo spessore predefinito viene salvato tramite la variabile di sistema THICKNESS.

La proprietà **Elevazione** specifica la distanza tra il piano della polilinea e il piano XY del WCS (sistema di coordinate globale).



THICKNESS = 0 (sinistra), **THICKNESS** = 50 (in mezzo), **ELEVATION** = 50 (destra)

Quando una polilinea viene modificata, è possibile modificare l'intera polilinea o modificare solo i singoli segmenti. Si possono, inoltre aggiungere o rimuovere i vertici.



Polilinea con larghezza zero	Polilinea con larghezza costante	Polilinea con rastremazione della larghezza

Nota:

- Quando FILLMODE = 0, viene visualizzato solo il contorno di tutte le entità riempite, come polilinee e solidi 2D.
- A seconda del valore della variabile di sistema GRIPS, nella polilinea verranno mostrati i grip sui punti finali e vertici (GRIPS = 1) o punti finali, vertici e punti medi (GRIPS=2)
- Le polilinee hanno una direzione, che può essere modificata dall'opzione **Inverti direzione** del comando EDITPL (edita polilinea).

8.14.1 Procedura Generale

1 Avviare il comando PLINEA.

Viene richiesto: Selezionare inizio della polilinea o [daPrecedente] <Precedente>:

2 Specificare un punto.

Viene richiesto: Specificare punto successivo o [Arco/Distanza/daPrecedente/Mezza-larghezza/Larghezza]:

3 Specificare un punto.

Viene richiesto: Specificare punto successivo o [Arco/Distanza/daPrecedente/Mezza-larghezza/Larghezza/ANnulla]:

4 Specificare un punto.

Ti chiede:

Specificare punto successivo o [Arco/CHiudi/Distanza/daPrecedente/Mezza-larghezza/LArghezza/ANnulla]:

5 Continuare specificando i punti.

6 Eseguire una delle seguenti operazioni:

- Scegliere l'opzione **Chiudi** per connettersi al punto iniziale e creare una polilinea chiusa.
- Scegliere l'opzione **Annulla** per rimuovere il vertice precedente.
- Cliccare il pulsante destro del mouse oppure premere Invio per interrompere.

8.14.2 Creare contorni polilinee

1 Avviare il comando CONTORNI.

Viene visualizzata la finestra di dialogo **Contorno**.

2 Cliccare sul pulsante **Seleziona punti nei contorni**.

La finestra di dialogo si chiude.

3 Cliccare nell'area in cui si desidera creare la polilinea di contorno.

La polilinea di contorno si evidenzia.

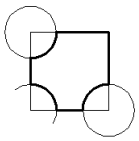
4 (opzionale) Continuare a scegliere punti per creare ulteriori contorni polilinea.

5 Cliccare il pulsante destro del mouse per accettare le polilinee.

Viene nuovamente aperta la finestra di dialogo **Contorno**.

6 Cliccare sul pulsante **OK**.

La polilinea o le polilinee vengono create.



8.15 Disegnare multilinee

Le multilinea sono costituite da due o più linee parallele, in base allo stile creato con il comando STILEML.

Lo strumento Multilinea è disponibile su:

- menu **Disegna**
- barra degli strumenti **Draw**
- pannello **Disegna** della barra multifunzione

8.15.1 Procedura Generale

1 Avviare il comando LINEAM.

1 Impostazioni Correnti: Giustificazione = Alto, Scala = 20.00, Stile = Standard

Viene richiesto: Punto iniziale o [Giustificazione/SCala/STile/Continua] <Ultimo punto>:

2 Specificare un punto.

3 **Viene richiesto**: Impostare il punto finale o [ANgolo/Lunghezza/Annulla]:

4 Specificare un punto.

5 **Viene richiesto**: Impostare il punto finale o [ANgolo/Lunghezza/Continua/Annulla]:

6 Specificare un punto.

7 **Viene richiesto**: Impostare il punto finale o [ANgolo/Lunghezza/Continua/CHiudi/Annulla]:

8 Eseguire una delle seguenti operazioni:
- Specificare altri punti.
- Scegliere l'opzione **Chiudi** per creare un loop chiuso.
- Cliccare il pulsante destro del mouse per concludere.

Note:

Le multilinee vengono create utilizzando lo stile multilinea corrente come definito dalla variabile di sistema CMLSTYLE.

La variabile di sistema CMLJUST imposta la giustificazione della multilinea rispetto ai punti specificati. Le opzioni sono:

- **Alto**: nella linea superiore dello stile multilinea.
- **Zero**: all'offset zero della definizione dello stile multilinea.
- **Basso**: nella parte inferiore dello stile multilinea.

La variabile di sistema CMLSCALE imposta la scala delle multilinee.

Le proprietà di una multilinea possono essere modificate nel pannello **Proprietà**.

8.16 Disegnare spline

Una spline è una curva morbida aperta o chiusa definita da un insieme di punti. È possibile utilizzare le spline per disegnare linee curve, che non possono essere disegnate come una polilinea.

Lo strumento Spline è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

8.16.1 Procedura Generale

1 Avviare il comando SPLINE.

Viene richiesto: Primo punto per spline:

2 Specificare un punto.

Viene richiesto: Secondo punto:

3 Specificare un punto.

Viene richiesto: Impostare il punto successivo o [Chiudi/ADatta tolleranza/Annulla]:

4 (opzione) Ripetere il passaggio 3 per selezionare più punti.

5 Cliccare il pulsante destro del mouse per smettere di aggiungere altri punti.

Viene richiesto: Seleziona punto tangente iniziale:

Una linea tangente alla spline viene visualizzata dinamicamente dal punto iniziale della spline.

6 Cliccare per definire la curva iniziale della spline.

Viene richiesto: Inserisci punto tangente finale:

Una linea tangente alla spline viene visualizzata dinamicamente dal punto finale della spline.e

7 Cliccare per creare la spline.

8.17 Disegno di schizzi a mano libera

Uno schizzo a mano libera è costituito da una serie di segmenti di linea, creati sia come entità linea individuali che come polilinea.

Uno schizzo a mano libera non viene aggiunto al disegno fino a quando l'utente non "scrive" lo schizzo nel disegno.

La variabile di sistema SKPOLY definisce se creare catene di linee o una polilinea.

La variabile di sistema SKETCHINC imposta la lunghezza o l'incremento di ciascuna linea o segmento polilinea. Più piccolo è l'incremento, più accurato è lo schizzo, ma un valore ridotto aumenta la dimensione del file.

Lo strumento Schizzo è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.

8.17.1 Procedura Generale

1 Avviare il comando SCHIZZO.

Viene richiesto: Lunghezza dei segmenti di linea per lo schizzo <lunghezza corrente>:

2 Eseguire una delle seguenti operazioni:

- Premere Invio per accettare la lunghezza corrente.
- Digitare un nuovo valore nella barra dei comandi.
- Cliccare due punti per definire la lunghezza graficamente.

3 Cliccare per iniziare a disegnare (penna giù).

Il cursore è stato sostituito dallo strumento Matita ( )

Ti chiede:

Premere INVIO o fare clic per attivare o disattivare [Cancella/COllega/salva ed ESci/CHiudi senza salvare/Record (salva)]:

4 Cliccare per avviare lo schizzo: mentre si sposta il cursore, le linee dello schizzo vengono visualizzate in verde chiaro.

5 (opzionale) Cliccare per interrompere il disegno a mano libera (penna su), quindi cliccare ancora (penna giù) per riprendere.

6 (opzione) Scegliere l'opzione **Cancella**.

Il cursore cambia con lo strumento **Cancella** ( ).

Spostare il cursore alla fine di una linea dello schizzo per iniziare la cancellazione.

Cliccare per interrompere la cancellazione e riprendere lo schizzo.

7 (opzione) Scegliere l'opzione **Record (salva)** per salvare lo schizzo e continuare a disegnare.

8 Scegliere l'opzione **salva ed ESci** per salvare lo schizzo e uscire dal comando di schizzo.

8.18 Disegnare cerchi

Il metodo predefinito per disegnare un cerchio è quello di specificare il punto centrale ed il raggio.

Altri metodi per disegnare cerchi possono essere trovati nel sottomenu Cerchio del menu Disegna e nella barra degli strumenti Cerchi.

Il comando AI_CIRCTAN disegna cerchi tangenti a tre entità lineari.

Lo strumento Cerchio è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

8.18.1 Procedura Generale

1 Avviare il comando CERCHIO.

Ti chiede:

Selezionare centro del cerchio o [2 Punti/3 Punti/Tangente-Tangente-Raggio/trasformare Arco in cerchio/cerchi Multipli]:

2 Specificare un punto.

Viene richiesto: Impostare Raggio o [Diametro] <raggio corrente>:

3 Eseguire una delle seguenti operazioni:

- Premere Invio per accettare il raggio corrente.
- Digitare un nuovo raggio e premere Invio.
- Cliccare per definire il raggio graficamente.

8.18.2 Disegnare un cerchio tangente a tre entità

1 Avviare il comando AI_CIRCTAN.

Viene richiesto: Specifica tangente:

2 Cliccare su un'entità lineare.

Sono accettate linee, archi, cerchi, segmenti di polilinea, xlinea e raggi.

Viene richiesto: Immettere la seconda tangente:

3 Cliccare sulla seconda entità lineare.

Viene richiesto: Immettere la terza tangente:

4 Cliccare sulla terza entità lineare.

Viene creato un cerchio tangente alle tre entità selezionate.

8.18.3 Modificare un cerchio

1 Selezionare il cerchio.

Le proprietà del cerchio vengono visualizzate nel pannello Proprietà.

2 Digitare un nuovo valore nei campi **CEntro**,**Raggio**,**Diametro**,**Circonferenza**o**Area** quindi premere Invio.

3 Premere Esc per interrompere le modifiche al cerchio.

8.19 Disegnare archi

Il metodo standard per disegnare archi di cerchio è quello di specificare tre punti: il punto d'inizio, un secondo punto ed il punto finale.

Altri metodi per disegnare archi sono disponibili nel sottomenu Arco del menu Disegna e nella barra degli strumenti Archi.

Lo strumento Arco è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.
- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

8.19.1 Procedura Generale

1 Avviare il comando ARCO.

Viene richiesto: Inserire inizio dell'arco o [Centro/COntinua dall'ultimo punto] <COntinua dall'ultimo punto>:

2 Specificare un punto.

Viene richiesto: Inserire secondo punto o [Angolo/Centro/Direzione/Fine/Raggio]:

3 Specificare un punto.

Viene richiesto: Punto finale:

4 Specificare un punto.

8.19.2 Modificare un arco

1 Selezionare l'arco.

Le proprietà dell'arco vengono visualizzate nel pannello Proprietà.

2 Digitare un nuovo valore nel campo **Centro**, **Raggio**, **Angolo iniziale** o **Angolo finale**, quindi premere Invio.

3 Premere il tasto Esc per bloccare la modifica dell'arco.

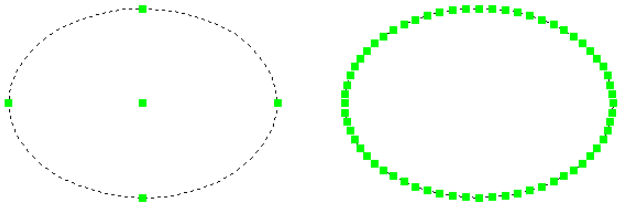
8.20 Disegnare ellissi

Il metodo standard per disegnare un'ellisse è quello di specificare i punti finali di un'asse dell'ellisse e quindi specificare una distanza che rappresenta metà della lunghezza del secondo asse. I punti finali del primo asse determinano l'orientamento dell'ellisse. L'asse più lungo dell'ellisse è denominato asse maggiore, il più corto è denominato asse minore. L'ordine con il quale si definiscono gli assi non è importante. Il programma determina l'asse maggiore e minore basandosi sulle loro dimensioni. Metà dell'asse maggiore viene chiamato raggio maggiore, metà dell'asse minore viene chiamato raggio minore. Il rapporto (raggio minore diviso per il raggio maggiore) definisce l'eccentricità dell'ellisse. Un cerchio è un'ellisse con un'eccentricità uguale a 1.

Altri metodi per disegnare archi sono disponibili nel sottomenu Ellisse del menu Disegna e nella barra degli strumenti Ellissi.

La variabile di sistema PELLIPSE controlla se vengono create ellissi reali o una rappresentazione polilinea.

La differenza tra un'ellisse reale e una rappresentazione polilinea di un'ellisse è visibile quando si seleziona l'ellisse. Un'ellisse reale ha cinque grips: il punto centrale ed i punti finali degli assi. Una rappresentazione polilinea di un'ellisse è una polilinea chiusa composta da segmenti di arco.



ellisse reale (sinistra) e rappresentazione di un'ellisse mediante polilinea (destra)

Lo strumento Ellisse è disponibile su:

- Menu Disegna.
- Barra degli strumenti Disegna.

- Pannello Disegna della barra multifunzione.
- Scheda Disegna del Quad con Nessuna Selezione.

8.20.1 Procedura generale per creare un'ellisse

1 Avviare il comando ELLISSE.

Viene richiesto: Punto finale del primo asse dell'ellisse [Arco/Centro]:

2 Specificare un punto:

Viene richiesto: Seconda fine asse:

3 Specificare un punto:

Viene richiesto: Impostare altro asse o [Rotazione]:

4 Specificare un punto.

8.20.2 Procedura generale per disegnare un arco ellittico

Un arco ellittico è una porzione di un'ellisse.

1 Avviare il comando ELLISSE.

Viene richiesto: Punto finale del primo asse dell'ellisse [Arco/Centro]:

2 Selezionare l'opzione Arco.

Ti chiede:

3 Specificare un punto:

Viene richiesto: Seconda fine asse:

4 Specificare un punto:

Viene richiesto: Impostare altro asse o [Rotazione]:

5 Specificare un punto.

Viene richiesto: Impostare angolo iniziale dell'arco o [Parametro]:

6 Eseguire una delle seguenti operazioni:

- Digitare un valore nel campo dinamico.
- Cliccare per specificare graficamente l'arco.

Viene richiesto: [Parametro/angolo Inscritto] <angolo Finale>:

7 Cliccare un punto.

8.20.3 Modificare un'ellisse o un arco ellittico

1 Selezionare l'ellisse o l'arco ellittico.

Le proprietà dell'ellisse vengono visualizzate nel pannello Proprietà.

2 Digitare un nuovo valore nei campi **CEntro**,**Raggio maggiore**,**Raggio minore**,**Rapporto del raggio**,**Angolo iniziale**o**Angolo finale** quindi premere Invio.

3 Premere Esc per interrompere le modifiche.

Nota: La modifica dei campi angolo iniziale o finale di un'ellisse trasformano l'ellisse in un arco ellittico. L'impostazione di entrambe le proprietà su 0 (zero) trasforma un arco ellittico in un'ellisse.

8.21 Disegnare rettangoli

Un rettangolo è creato come polilinea chiusa.

Un rettangolo è disegnato specificando due angoli opposti. Le quote dinamiche consentono di specificare la larghezza e l'altezza del rettangolo.

A meno che non si scelga l'opzione **Ruota**, i lati di un rettangolo sono sempre paralleli all'asse x e all'asse y del sistema di coordinate corrente (WCS o UCS).

Lo strumento Rettangolo è disponibile su:

- menu **Disegna**
- barra degli strumenti **Draw**
- pannello **Disegna** della barra multifunzione
- scheda **Disegna** del **Quad su Nessuna Selezione**

8.21.1 Procedura Generale

1 Avvia il comando RETTANGOLO.

1 **Ti chiede**:

Selezionare primo angolo del rettangolo [Cima/Raccordo/RUota/Quadrato/Elevazione/Spessore/Larghezza linea/Area/Dimensioni]:

2 Specificare un punto.

3 Il rettangolo viene visualizzato in modo dinamico.

Viene richiesto: Angolo opposto del rettangolo:

4 Eseguire una delle seguenti operazioni:
- Specificare un punto.
- Immettere un valore nel campo quota dinamica.
- Premere il tasto TAB per passare tra i campi. Premere Invio per confermare.

Nota:

Dato che un rettangolo viene creato come una polilinea chiusa, i rettangoli hanno una direzione. La direzione di una polilinea chiusa è positiva se questa viene disegnata in senso antiorario ed è negativa se viene disegnata in direzione oraria. I rettangoli hanno una direzione antioraria se entrambe le coordinate X ed Y del secondo punto sono maggiori o minori delle coordinate del primo punto (= movimento del cursore è positivo o negativo lungo le assi X ed Y). I rettangoli hanno una direzione oraria se le coordinate X od Y sono maggiori o minori delle coordinate X od Y del primo punto (= movimento del cursore è positivo lungo l'asse X e negativo lungo l'asse Y o viceversa).

L'opzione Inverti del comando EDITPL (Edita Polilinea) modifica la direzione di una polilinea.

8.22 Disegnare poligoni

I poligoni sono delle polilinee chiuse che contengono da un minimo di tre ad un massimo di 1.024 lati della stessa lunghezza.

È possibile disegnare poligoni utilizzando i seguenti metodi:

- **Centro - Vertice**: in primo luogo definire il punto centrale, quindi il vertice (= metodo del raggio circoscritto).
- **Centro - Lato**: in primo luogo definire il punto centrale, quindi il punto medio di un lato (= metodo del raggio del cerchio inscritto).
- **Spigolo**: definire la lunghezza del lato del poligono.

Lo strumento Poligono è disponibile su:

- menu **Disegna**
- barra degli strumenti **Draw**
- pannello **Disegna** della barra multifunzione
- scheda **Disegna** del **Quad su Nessuna Selezione**

8.22.1 Procedura Generale

1 Avviare il comando POLIGONO.

1 **Viene richiesto**: Poligono: Digitare il numero di lati o [Larghezza linea/poligoni Multipli] <4>:

2 Eseguire una delle seguenti operazioni:
   - Premere Invio o cliccare il pulsante destro del mouse per accettare il numero attuale di lati.
   - Digitare un valore, quindi premere Invio.

   Viene richiesto: Specificare da: Specificare centro del poligono o [definire da Spigolo]:

3 Specificare un punto.

4 **Viene richiesto**: Specificare da: Selezionare punto medio del lato o [definire da Vertice]:

5 Specificare un punto.

Nota:

- L'opzione scelta nel passaggio 3 (Selezionare punto medio del lato o definire vertice) viene ricordata la prossima volta.
- Il numero di lati è impostato dalla variabile di sistema POLYSIDES, ma il valore non viene salvato tra le sessioni. La prima volta che si lancia il comando POLIGONO in una sessione BricsCAD, POLYSIDES = 4.

8.23 Disegnare tracce

Le tracce sono entità con lati, piene, create con due lati paralleli. Quando si crea una serie di tracce, quest'ultime sono collegate l'una all'altra.

Lo strumento Traccia è disponibile su

- menu **Disegna**.
- pannello **Disegna** della barra multifunzione

La larghezza predefinita della traccia viene salvata tramite la variabile di sistema TRACEWID.

8.23.1 Procedura Generale

1 Avviare il comando TRACCIA.

1 **Viene richiesto:** Larghezza traccia <larghezza corrente>:

2 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse per accettare la larghezza corrente.
   - Digitare un valore, quindi premere Invio.
   - Definire una nuova larghezza corrente graficamente cliccando due punti.

   Viene richiesto: Inizio traccia:

3 Specificare il punto iniziale della traccia.

4 **Viene richiesto:** Punto successivo:

5 Specificare il punto finale della prima traccia.

6 Una linea indica la direzione della prima traccia.

   La direzione della seconda traccia viene visualizzata in modo dinamico.

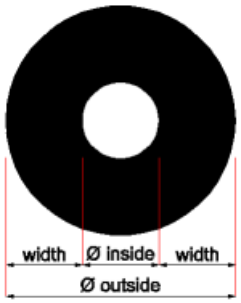
7 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse per creare una traccia.
   - Specificare il punto finale della traccia successiva.

8 Eseguire una delle seguenti operazioni:
   - Ripetere il passaggio 5 per continuare ad aggiungere tracce.
   - Cliccare il pulsante destro del mouse per interrompere l'operazione.

Nota: Quando la variabile di sistema FILLMODE è disattivata, tutte le entità riempite, come ad es. le tracce, vengono visualizzate e stampate come linee guida.

8.24 Disegnare anelli

Gli anelli sono cerchi o anelli pieni creati come polilinee chiuse, larghe a 2 segmenti.



Il metodo predefinito per disegnare anelli è specificare i suoi diametri interni ed esterni, quindi specificarne il centro. Se il diametro interno è zero, viene creato un cerchio pieno.

Lo strumento Anello è disponibile su:

- menu **Disegna**
- pannello **Disegna** della barra multifunzione

8.24.1 Procedura Generale

1 Avviare il comando ANELLO.

1 **Viene richiesto**: Diametro interno dell'anello [2 Punti/3 Punti/TangenTe tangente Raggio] <diametro interno corrente>:

2 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse per accettare il diametro interno corrente.
   - Digitare un nuovo valore, quindi premere Invio.
   - Definire graficamente un nuovo diametro interno cliccando due punti.

   Viene richiesto: Diametro esterno dell'anello <diametro esterno corrente>:

3 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse per accettare il diametro esterno corrente.
   - Digitare un nuovo valore, quindi premere Invio.
   - Definire graficamente un nuovo diametro esterno cliccando due punti.

   Viene richiesto: Centro dell'anello:

   Specificare un punto.

   Viene richiesto: Centro dell'anello:

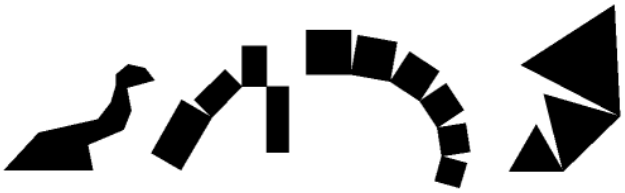
4 Eseguire una delle seguenti operazioni:
   - Continuare a creare anelli.
   - Cliccare il pulsante destro del mouse per interrompere l'operazione.

Nota:

- Quando la variabile di sistema FILLMODE è disattivata, tutte le entità riempite, come ad es. gli anelli, vengono visualizzate e stampate come linee guida.
- Il valore attuale del diametro interno ed esterno viene salvato attraverso le variabili di sistema DONUTID e DONUTOD. Questi valori non vengono salvati tra le sessioni. La prima volta che si lancia il comando ANELLO in una sessione BricsCAD, DONUTID=0.5 e DONUTOD=1.

8.25 Disegnare solidi 2D

Solidi 2D sono aree rettangolari, quadrate, triangolari o quadrangolari, riempite con un colore a tinta unita.



Da sinistra a destra: solido quadrilatero, rettangolo, quadrato e triangolo

Lo strumento Solid è disponibile su:

- menu **Disegna**
- barra degli strumenti **Disegna**
- pannello **Disegna** della barra multifunzione

8.25.1 Procedura Generale

1 Avviare il comando POLIG.

1 **Viene richiesto**: Impostare il primo punto del piano o [Rettangolo/Quadrato/Triangolo]:

2 Specificare un punto.

3 **Viene richiesto**: Secondo punto:

4 Specificare un punto.

5 **Viene richiesto**: Terzo punto del piano:

6 Specificare un punto.

7 **Viene richiesto**: Quarto punto:

8 Specificare un punto.

9 Il solido viene creato.

Viene richiesto: Terzo punto del piano:

10 Eseguire una delle seguenti operazioni:
   - Continuare a specificare i punti per creare più solidi.
   - Cliccare il pulsante destro del mouse per interrompere l'operazione.

Nota: Quando la variabile di sistema FILLMODE è disattivata, tutte le entità riempite, come ad es. i solidi 2D, vengono visualizzate e stampate come linee guida.

8.26 Disegnare fumetti di revisione

Un Fumetto di Revisione è una polilinea aperta o chiusa composta da archi. I fumetti di revisione vengono utilizzati per rivedere o ridefinire gli scopi, per indicare le parti del disegno che devono essere corrette o annotate.

Fumetti di Revisione: 1. Normale 2. Calligrafia 3. Calligrafia invertita (basata su un rettangolo).

Nota: L'opzione **Inverti** è disponibile solo in modalità **Entità** o quando si crea un fumetto di revisione aperto utilizzando la modalità a **Mano libera**.

REVCLOUDCREATEMODE: imposta la modalità predefinita.

REVCLOUDMINARCLENGTH: imposta la lunghezza minima dell'arco.

REVCLOUDMAXARCLLENGTH: imposta la lunghezza massima dell'arco.

REVCLOUDARCSTYLE: imposta lo stile predefinito (normale o calligrafia).

REVCLOUDGRIPS: quando è On, la modifica mediante grips di un fumetto di revisione influisce anche sui segmenti adiacenti.

Lo strumento Fumetto di Revisione è disponibile su:

- menu **Disegna**
- barra degli strumenti **Disegno**
- pannello **Disegna** della barra multifunzione
- scheda **Disegna** del **Quad su Nessuna Selezione**

8.26.1 Procedura Generale

1 Avviare il comando FUMETTOREV.

1 **Ti chiede**:

Specificare primo angolo o [lunghezza Arco/Entità/Rettangolare/Poligonale/MAno libera/Stile] <Entità>:

2 Specificare un punto per iniziare a creare un fumetto di revisione in modalità a mano libera.

3 Quando si sposta il cursore, viene creato il fumetto di revisione.

4 Spostare il cursore sul punto iniziale per terminare.

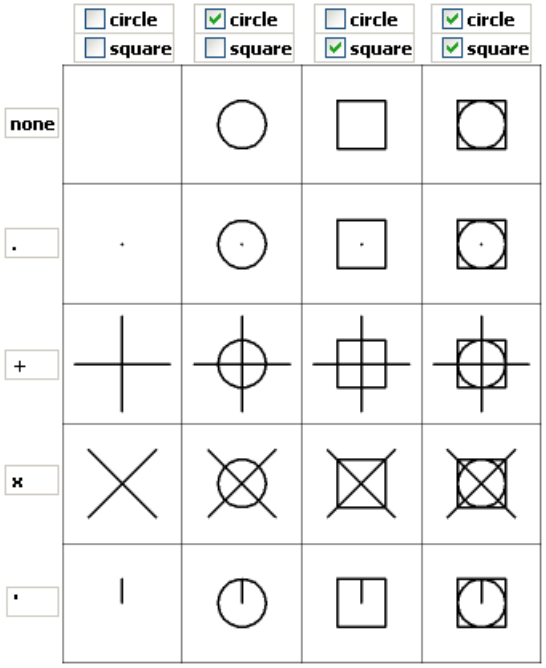
5 BricsCAD riporta nella barra dei comandi: Fumetto di revisione terminato.

Nota:

- Cliccando il pulsante destro del mouse alla prima richiesta, si inizia a creare un fumetto di revisione nella modalità predefinita (= valore corrente della variabile di sistema REVCLOUDCREATEMODE).
- Cliccando il pulsante destro del mouse prima del punto iniziale, viene creato un fumetto di revisione aperto.

8.27 Disegnare punti

Poiché i punti non hanno dimensioni, un singolo pixel dovrebbe essere la rappresentazione (quasi) corretta sullo schermo. Ma questo è difficilmente visibile, specialmente in un disegno complesso. Pertanto, è possibile scegliere tra diversi stili di visualizzazione possibili, impostati dalla variabile di sistema PDMODE (Modallità Visualizzazione Punto):



La dimensione dello stile di rappresentazione del punto è controllata tramite la variabile di sistema PDSIZE (Dimensione Visualizzazione Punto). Quando PDSIZE=0 la dimensione di visualizzazione del punto è pari al 5% della dimensione dello schermo.

Nota: Se PDSIZE=0 (zero), utilizzare il comando RIGEN per ridimensionare la visualizzazione dei punti al 5% delle dimensioni dello schermo dopo lo zoom avanti o indietro.

Lo strumento Punto è disponibile su:

- menu **Disegna**
- barra degli strumenti **Disegno**
- pannello **Disegna** della barra multifunzione

8.27.1 Procedura Generale

1 Avviare il comando PUNTO.

1 **Viene richiesto**: Posizione del punto o [Impostazioni punto.../punti Multipli]:

2 Specificare un punto.

8.28 Disegnare entità coprente

Un'entità coprente è un'area poligonale che nasconde le entità sottostanti usando il colore di sfondo corrente. Più comunemente le entità coprenti sono usate per mascherare parte del disegno per aumentarne la chiarezza. Ad esempio, il tuo obbiettivo potrebbe essere quello di aggiungere un testo ad una parte complicata del disegno. Un'entità coprente potrebbe essere usata per mascherare l'area dietro il testo per aumentarne la leggibilità del testo stesso.

L'opzione **Frame** del comando ENTCOPR imposta il valore della variabile di sistema WIPEOUTFRAME, che controlla se i contorni dell'entità coprente vengono visualizzati e/o stampati.

Nota: Le entità coprenti nascondono solo le entità esistenti. Usare l'opzione **Porta in primo piano** del comando ORDINEDIS per nascondere tutte le entità.

Lo strumento Entità Comprente è disponibile su:

- menu **Disegna**
- barra degli strumenti **Disegno**
- pannello **Disegna** della barra multifunzione
- scheda **Disegna** del **Quad su Nessuna Selezione**

8.28.1 Procedura Generale

1 Avviare il comando ENTCOPR.

1 **Viene richiesto**: Specificare punto iniziale o [Polilinea/Cornici]:

2 Specificare un punto.

3 **Viene richiesto**: Specificare punto successivo:

4 Specificare un punto.

5 **Viene richiesto**: Specificare punto successivo o [Annulla]:

6 Specificare un punto.

7 **Viene richiesto**: Specificare punto successivo o [Annulla/Chiudi]:

8 Eseguire una delle seguenti operazioni:
   - Continuare specificando i punti.
   - Cliccare il pulsante destro del mouse o scegliere **Chiudi** per interrompere.

8.28.2 Creare un'entità coprente da una polilinea

1 Avviare il comando ENTCOPR.

1 **Viene richiesto**: Specificare punto iniziale o [Polilinea/Cornici]:

2 Scegliere l'opzione **Polilinea**.

Viene richiesto: Seleziona una polilinea chiusa [opzioni di selezione (?)]:

3 Selezionare una polilinea chiusa, composta solo da segmenti di linea.

4 **Viene richiesto**: Eliminare la polilinea selezionata? [Sì/No]< No>:

5 Eseguire una delle seguenti operazioni:
   - Premere Invio per mantenere la polilinea.
   - Scegliere **Sì** per eliminare la polilinea.

8.29 Disegnare eliche

Un'elica è una curva che ruota attorno a un asse sulla superficie di un cilindro (elica cilindrica) o cono (elica conica) mentre si alza con un angolo costante verso l'alto a partire da una base.

Una spirale è una curva planare che si snoda attorno a un punto centrale fisso a una distanza crescente da quel punto.

Il comando ELICA può creare sia eliche 3D che spirali 2D.

Lo strumento Elica è disponibile su:

- menu **Disegna**
- barra degli strumenti **Polilinee**
- pannello **Disegna** della barra multifunzione

8.29.1 Procedura generale per disegnare una spirale

1 Avviare il comando ELICA.

1 Numero di spire = 3 Torsione = Antiorario

Viene richiesto: Specificare centro della base:

2 Specificare un punto.

3 Specificare il raggio della base o [Diametro] <1>:

4 Eseguire una delle seguenti operazioni:
   - Specificare un punto.
   - Digitare un valore, quindi premere Invio.

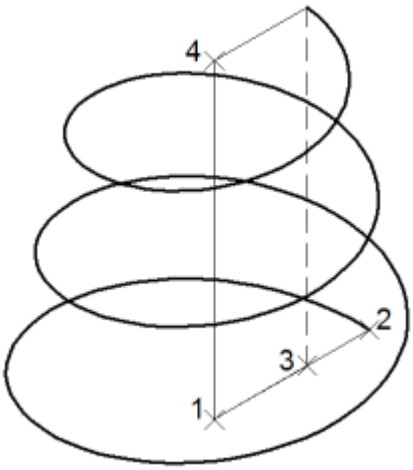
Viene richiesto: Specificare raggio superiore o [Diametro] <raggio base>:

5 Eseguire una delle seguenti operazioni:
   - Specificare un punto.
   - Digitare un valore, quindi premere Invio.

Viene richiesto: Specificare altezza elica o [punto finale Asse/Spire/alteZza spira/Torsione] <1>:

6 Digitare 0 (zero), quindi premere Invio

8.29.2 Procedura generale per disegnare un'elica



1 Avviare il comando ELICA.

1 Numero di spire = 3 Torsione = Antiorario

Viene richiesto: Specificare centro della base:

2 Specificare un punto (1).

3 **Viene richiesto**: Specificare il raggio della base o [Diametro] <1>:

4 Eseguire una delle seguenti operazioni (2):
- Specificare un punto.
- Digitare un valore, quindi premere Invio.

5 Eseguire una delle seguenti operazioni (3):
- Premere Invio per impostare il raggio superiore uguale al raggio della base, creando così un elica cilindrica.
- Specificare un punto.
- Digitare un valore, quindi premere Invio.

Viene richiesto: Specificare altezza elica o [punto finale Asse/Spire/alteZza spira/Torsione] <1>:

6 Eseguire una delle seguenti operazioni (4):
- Specificare un punto.
- Digitare un valore, quindi premere Invio.

8.29.3 Modificare un'elica

1 Selezionare l'elica.

1 Le proprietà dell'elica vengono visualizzate nella categoria **Geometria** nel pannello **Proprietà**.

2 Modificare le proprietà:
- **Vincola**: vincola l'altezza, l'altezza della spira o il numero di spire. Le proprietà non vincolate vengono regolate automaticamente.
- **Altezza**: altezza totale dell'elica. altezza = altezza spira * spire.

- **Spire**: numero di spire.
- **Altezza spira**: altezza di una spira.
- **Raggio base**: raggio iniziale.
- **Raggio superiore**: raggio finale.
- **Torsione**: direzione CW (Clockwise=senso orario) o CCW (Counterclockwise=senso antiorario).

Nota:

- Impostare il **Raggio base superiore** = **Raggio base** per eliche cilindriche.
- Impostare **Altezza** o **Altezza spira** su zero per le spirali.
- Il comando ESPLODI trasforma un'elica in una spline.

8.30 Disegnare polilinee 3D

Una polilinea 3D è composta da segmenti lineari nello spazio 3D.

Lo strumento 3DPOLI è disponibile su:

- menu **Disegna**
- barra degli strumenti **Polilinee**
- pannello **Disegna** della barra multifunzione

8.30.1 Procedura Generale

1 Avviare il comando 3DPOLI.

1 **Viene richiesto**: Selezionare inizio della polilinea o [daPrecedente] <Precedente>:

2 Specificare un punto.

3 **Viene richiesto**: Punto successivo:

4 Specificare un punto.

5 **Viene richiesto**: Punto successivo o [Chiudi/Annulla]:

6 Eseguire una delle seguenti operazioni:
   - Scegliere l'opzione **Chiudi** per connettersi al punto di partenza e creare una polilinea 3D triangolare planare.
   - Specificare un punto.

   Viene richiesto: Specificare punto successivo o [CHiudi/ANnulla]:

7 Eseguire una delle seguenti operazioni:
   - Scegliere l'opzione **Chiudi** per connettersi al punto iniziale.
   - Specificare un punto.

   Viene richiesto: Specificare punto successivo o [CHiudi/ANnulla]:

8 Continuare a specificare i punti o premere Invio per interrompere.

8.30.2 Modificare una polilinea 3D

1 Selezionare la polilinea 3D.

1 Le proprietà vengono visualizzate nel pannello Proprietà.

2 Sotto **Varie**:

- **Adatta/Curva**: selezionare **Nessuno**, **Cubica** o **Quadratica**.

  Le opzioni **Cubica** e **Quadratica** creano una rappresentazione spline della polilinea 3D.

- **Chiusa**: quando l'opzione è impostata su **No**, rimuove il segmento di chiusura.

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 Creare tratteggi e riempimenti sfumati

Il comando TRATTEGGIO riempie un'area chiusa nel disegno con una trama.

Il comando SFUMAT crea riempimenti solidi in nove diversi modelli con uno o due colori.

L'area può essere determinata con una singola entità, come ad es. un cerchio o una polilinea chiusa, oppure con una zona delimitata da una selezione di entità.

Per creare il tratteggio è possibile cliccare all'interno del perimetro chiuso oppure è possibile selezionare le entità.

Nota:

- A seconda del modello, i tratteggi fanno un utilizzo intensivo della memoria e potrebbero richiedere una considerevole quantità di tempo per essere disegnati e visualizzati. Per migliorare le prestazioni, aggiungere i tratteggi alla fine del disegno, o inserire i tratteggi su un layer separato in modo da poterlo congelare, e poter continuare a lavorare sul disegno. Oppure utilizzare la variabile di sistema FILLMODE per disattivare la visualizzazione dei tratteggi, riempimenti sfumati ed entità riempite.
- Il numero massimo di trattini è controllato dalla variabile di impostazione MAXHATCH. Quando il numero di trattini supera il valore corrente di MAXHATCH, il tratteggio non può essere creato. Una finestra di avviso viene visualizzata offrendo alcune scelte per le azioni successive. Per impostazione predefinita, il numero massimo di trattini è 100 000.

Gli strumenti Tratteggio e Sfumatura sono disponibili:

- menu **Disegna**
- barra degli strumenti **Disegno**
- pannello **Disegna** della barra multifunzione
- sulla scheda **Disegno** del **Quad su Nessuna Selezione** (solo Tratteggio)

8.32.1 Procedura Generale

1 Avviare il comando TRATTEGGIO o SFUMAT.

1 Viene visualizzata la finestra di dialogo **Tratteggio e Sfumatura**.

2 Eseguire una delle seguenti operazioni:

3 (opzionale) Cliccare sulla scheda **Tratteggio** per creare un tratteggio, quindi definire le proprietà del tratteggio.

4 (opzionale) Cliccare sulla scheda **Sfumatura** per creare un riempimento sfumato, quindi definire le proprietà del riempimento sfumato.

5 Definire le proprietà del tratteggio o del riempimento sfumato.

6 Scegliere il contorno del tratteggio o del riempimento sfumato.

7 Impostare le opzioni.

8 Cliccare sul pulsante **OK** per creare il tratteggio o il riempimento sfumato.

I tratteggi e i riempimenti sfumati vengono creati sul layer definito dalla variabile di sistema HPLAYER. Predefinito = layer corrente.

8.32.2 Definire le proprietà del tratteggio

1 Selezionate la scheda **Tratteggio** nella finestra di dialogo **Tratteggio e Sfumatura**.

2 (opzionale) Cliccare sul pulsante **Acquisisci proprietà**.

La finestra di dialogo **Tratteggio e Sfumatura** si chiude temporaneamente.

Viene richiesto: Seleziona entità tratteggio da cui copiare le proprietà:

Selezionare entità tratteggio.

Viene riaperta la finestra di dialogo **Tratteggio e Sfumatura**, in cui sono visualizzate le proprietà dell'entità di tratteggio selezionata.

3 (opzionale) Selezionare il **Tipo** di modello:
- **Definito da utente**: utilizza il modello definito in questa finestra di dialogo con le opzioni **Angolo**, **Spaziatura** e **Tratteggio Doppio**.
- **Predefinito**: utilizza modelli definiti in:
  - iso.pat (unità metriche -MEASUREMENT= ON)
  - default.pat (unità imperiali - MEASUREMENT = OFF).
- **Personalizzato** - utilizza modelli definiti nei file *.pat (una singola definizione di modello per ogni file).

  BricsCAD ricerca i files *.pat nelle cartelle definite nella variabile di sistema SRCHPATH (Percorsi di Ricerca File di Supporto).

4 Se il **Tipo** di motivo è **Predefinito** o **Personalizzato**:
- Doppio clic sul campo **Campione**,
- Selezionate un **Nome** di tratteggio nella **Tavolozza Modelli Tratteggio**, quindi doppio clic sul modello o cliccare sul pulsante **OK**.
- Digitare una scala nel campo **Scala** o scegliere la scala dall'elenco a discesa.
- Digitare un angolo nel campo **Angolo** o scegliere l'angolo dall'elenco a discesa.

5 Se il **Tipo** di modello è **Definito da utente**:
- Specificare la **Spaziatura** del modello nelle unità di disegno.
- Facoltativamente, impostare l'opzione **Tratteggio Doppio**.

6 Definire l'**Origine del Tratteggio**.

Le opzioni sono:
- **Utilizza origine corrente**: viene riutilizzata l'origine come definita dalla variabile di sistema HPORIGIN.
- **Origine specificata**: cliccare sul pulsante **Scegli nuova origine**, quindi scegliere un punto nel disegno. La nuova origine rimane memorizzata nella variabile di sistema HPORIGIN.

7 Impostare il **Colore** per il modello di tratteggio.

8 Impostare il colore di **Sfondo**.

8.32.3 Definire le proprietà del riempimento sfumato

1 Selezionare la scheda **Sfumatura** nella finestra di dialogo **Tratteggio e Sfumatura**.

2 (opzionale) Cliccare sul pulsante di opzione **Un colore** per creare un 'colore dal bianco' o un 'colore dal nero' per il riempimento sfumato.
- Cliccare sul riquadro colorato per aprire la finestra di dialogo **Seleziona Colore**, per scegliere un colore di base.

- Il cursore del colore imposta la transizione dal colore al bianco o al nero.
- Tenere premuto il pulsante sinistro del mouse per spostare il cursore del colore.

  Spostare il cursore all'estrema destra per creare un riempimento sfumato da colore a bianco.

  Spostare il cursore all'estrema sinistra per creare un riempimento sfumato «da colore a nero.

3 (opzionale) Cliccare sul pulsante di opzione **Due colori** per creare un riempimento sfumato "due colori".
- Cliccare sui riquadri colorati per aprire la finestra di dialogo **Seleziona Colore** per selezionare i colori di partenza.
- Selezionare uno dei 9 modelli di sfumatura.
- Le opzioni sono: Lineare, Cilindrica, Cilindrica Invertita, Sferica, Emisferica, Curva, Sferica Invertita, Emisferica Invertita e Curva Invertita.

4 (opzionale) **Centrato**: Se centrato, il riempimento sfumato viene creato simmetrico, se non centrato, il riempimento sfumato viene costruito partendo dalla sinistra del contorno.

5 **Angolo**: Specifica l'angolo del riempimento sfumato rispetto all'UCS corrente.

8.32.4 Definire il contorno

1 Selezionare un'opzione **Mantieni contorni**.

2 Selezionare un'opzione di rilevamento **Isola**.

Questo vale per le aree chiuse all'interno dei contorni del tratteggio.

3 Impostare la **Tolleranza spazio**. Questo valore definisce la dimensione massima di eventuali spazi vuoti nel contorno. Quando è impostato su 0 (zero), BricsCAD imposta la tolleranza rispetto alla dimensione di visualizzazione corrente. Eseguire lo zoom indietro per aumentare la tolleranza.

Questo valore viene salvato tramite la variabile di sistema HPGAPTOL.

4 (opzionale) Cliccare sul pulsante **Seleziona** per selezionare un **gruppo di Contorni**.

La finestra di dialogo **Tratteggio e Sfumatura** si chiude temporaneamente per consentire di selezionare le entità.

Se un gruppo di selezione era attivo durante l'avvio del comando TRATTEGGIO o SFUMAT, questo gruppo di selezione verrà utilizzato come **gruppo di contorni**. In questo caso, la finestra di dialogo **Tratteggio e Sfumatura** si chiude e si riapre immediatamente quando si clicca sul pulsante **Seleziona**.

5 Eseguire una delle seguenti operazioni:
- Cliccare sul pulsante **Scegli punti**.

  La finestra di dialogo **Tratteggio e Sfumatura** si chiude.

  Viene richiesto: Selezionare un punto per definire un'area di contorno o di tratteggio o [Seleziona entità/Annulla]:
- Specificare un punto in ciascuna delle aree da tratteggiare.
- **Viene richiesto**: Selezionare un punto per definire un'area di contorno o di tratteggio o [Seleziona entità/Annulla]:
- Cliccare il pulsante destro del mouse o premere Invio per interrompere l'aggiunta di contorni.

- Viene nuovamente visualizzata la finestra di dialogo **Tratteggio e Sfumatura**.
- Cliccare sul pulsante **Seleziona le entità del contorno**.

  La finestra di dialogo **Tratteggio e Sfumatura** si chiude.

  Viene richiesto Selezionare entità:
- Selezionare le entità che si desidera tratteggiare.
- Cliccare il pulsante destro del mouse o premere Invio per interrompere la selezione delle entità.
- Viene nuovamente visualizzata la finestra di dialogo **Tratteggio e Sfumatura**.
- Cliccare sul pulsante **Rimuovi le entità del contorno**.

  La finestra di dialogo **Tratteggio e Sfumatura** si chiude.

  Viene richiesto Selezionare entità:
- Selezionare le entità contorno correnti da rimuovere dalla selezione.
- Cliccare il pulsante destro del mouse o premere Invio per interrompere la selezione delle entità.
- Viene nuovamente visualizzata la finestra di dialogo **Tratteggio e Sfumatura**.

8.32.5 Impostare le opzioni

- L'opzione **Annotativo** crea dei contorni annotativi.
- Selezionare l'opzione **Associativo** per associare il tratteggio al contorno.
- Selezionare l'opzione **Crea tratteggi separati**. Quando vengono rilevati più contorni o vengono selezionate più entità, vengono creati tratteggi separati, anziché un unico tratteggio.
- Scegliere un'opzione su **Ordine di visualizzazione**.

  Questa opzione viene reimpostato su **Porta dietro al contorno** quando il disegno viene chiuso e riaperto.
- **Layer**: scegliere un layer esistente o **Usa Corrente** (impostazione predefinita).
- **Trasparenza**: impostare la trasparenza; vengono accettati valori compresi tra 0 e 90.

8.33 Editing hatches and gradient fills

Hatches and gradient fills can be edited:

- In the **Properties** panel.
- Using the HATCHEDIT command, in a dialog box.
- Using tools in the Edit command group in the Quad when a hatch or gradient fill entity is selected.

8.33.1 To edit a hatch or gradient fill in the properties panel

1 Select the hatch or gradient fill in the drawing.

1 The properties of the selected entity display in the **Properties** panel.

2 Select the property you want to edit in the **Properties** bar.

  The selected property field is activated.

3 Edit the selected property.

4 Press the Esc key to stop editing the hatch.

Nota:

- When multiple hatches or gradient fills are selected properties which are different display as ***varies***. If you edit such property, the changes apply to entire selection.
- Linetype, Linetype scale and Lineweight properties apply to **User Defined** pattern types only.

8.33.2 To edit a hatch or gradient fill in a dialog box

The Edit Hatch tool is available on:

- the **Modify** menu
- the **Modify** toolbar
- the **Modify** ribbon panel

1 Do one of the following:
   - Launch the HATCHEDIT command.
   - Select a hatch or gradient entity.
   - Double click a hatch or gradient fill entity.

   The **Hatch Edit** dialog box displays.

2 Edit the properties and settings.

3 The changes are applied in real time.

4 Click the **OK** button to accept the changes.

8.33.3 To edit a hatch or gradient fill using the Quad

1 Pause the cursor on the hatch or gradient fill.

1 The Quad displays.

2 Move the cursor to the Quad.

3 (option) In the **Edit** tab, click the **Add Loop** button.

   You are prompted: specify internal point or [Select entities/Undo]:
   - Click inside a closed area,
   - or

     Type **S** , then press Enter to select entities.
   - Right click or press Enter to conclude.

4 (option) In the **Edit** tab, click the **Remove Loop** button.

   The boundaries of the selected hatch highlight.

   You are prompted: select entities:
   - Click the boundaries you want to remove.
   - Right click or press Enter to conclude.

5 (option) In the **Edit** tab, click the **Send to back** button.

All hatches in the drawing are sent behind other entities.

6 (option) In the **Edit** tab, click the **Match** button.

You are prompted: select entities to copy properties to or [Settings/selection options (?)]:

- Select the hatch entities to copy the properties to.
- Right click or press Enter to conclude.

7 (option) In the **Draw** tab, click the **Generate Boundary** button.

The boundary of the hatch is generated.

9. Modellazione 3D

9.1 Direct modeling overview

Direct modeling is much like sculpting on-screen: just like a sculptor starts with a lump of clay or a piece of stone or wood, you start with a base volume, which is then further detailed by adding, removing and deforming geometry. All operations are executed dynamically, and you decide whether to define dimensions intuitively or by specifying values. Unlike history-based modeling, the order in which features are created is not important. All operations are performed in interactive mode: the intermediate result of an operation shows dynamically, according to user input defined by mouse movement.

A key feature of direct editing capabilities provided by BricsCAD consists in preserving the design intent. 3D dimensional constraints (**Distance**, **Radius**, and **Angle**) imposed for faces and edges give you full parametric control over the solid model. Different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometrical constraints (**Coincident**, **Tangent**, **Parallel** and others) define how these changes influence the geometry of the model. However, there is no need to specify all geometrical constraints manually since most of them can be recognized automatically by setting **Design Intent Recognition** options.

Another feature of **Direct Editing** operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations based on the currently selected or previewed entity or selection set. At the same time the Quad provides the other basic solid editing operations such as copy, union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported. When the separately licensed BricsCAD Communicator is installed, the IMPORT command supports all major 3D model formats, both industry standard and proprietary formats.

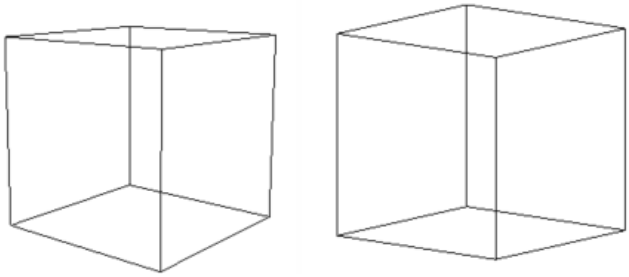
Depending on the value of the DELOBJ (Delete Entity) system variable, the defining entities are either retained or deleted or you are prompted whether to keep or delete the source entities (see table below).

Value	Result
0	Retain defining entities
1	Delete profile entities; retain paths and guide curves
2	Delete all defining entities
-1	Prompt to delete profile entities; retain paths and guide curves
-2	Prompt to delete all defining entities

9.2 Navigate in 3D

As you navigate around and through your 3D model, the location of the model in space remains constant. It is your current view (viewpoint), of the model that is changing. You can see the current view properties in the **Properties** panel when nothing is selected. The **Camera** (X,Y,Z) represents your location as you view the model. Pretend you are viewing it through a camera lens. The **Target** (X,Y,Z) is where the camera lens is pointed. As you navigate around the model, the **Camera** and **Target** coordinates update accordingly. Those coordinates are read-only in the **Properties** panel. There are other **View** properties that you can edit. **Perspective** is the most important control to be aware of for navigation purposes.

You can set **Perspective** to **On** or **Off** for the current view. When set to **On**, the view displays the visual perspective mode (left). When set to **Off**, it displays the parallel perspective mode (right).



9.2.1 To control the Perspective property

Do one of the following:

- Select the **View / Perspective** property in the **Properties** panel.
- Right click the **Look From** widget, then choose **Perspective** in the context menu.

9.2.2 Mouse and keyboard navigation options

One of the easiest and most common methods for navigating around a 3D model is with the mouse and keyboard. Simply press the Shift key and middle mouse button as you move the mouse to orbit the model. The cursor changes to an orbit icon indicating you're in orbit mode.

There are many other mouse and keyboard navigation methods that you're probably familiar with from your 2D CAD experience. Most of those methods are also useful for 3D modeling.

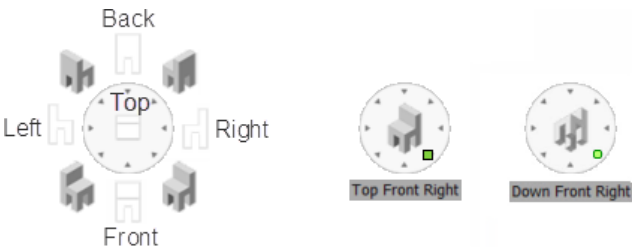
- Zoom in/out – Roll the mouse wheel. You can zoom all the way into a model and walk around inside it when in perspective mode. In isometric mode you can only zoom in very close, but not go inside.
- Zoom Extents – Double-click the mouse wheel.
- Pan – Press the mouse wheel or middle mouse button.

See the 'View Manipulation using the Mouse and Keyboard' article for a comprehensive list.

9.2.3 The LookFrom navigation tool

The **LookFrom** tool is displayed in the upper right corner of the **Drawing** area. Click on different places on the **LookFrom** tool to display the view from standard viewpoints. By default, it includes **Top**, **Left**, **Right**, **Front**, **Back**, **Top Back Left**, **Top Back Right**, **Top Front Left**, and **Top Front Right**. As you pass the cursor

over a viewpoint, the relevant icon appears at the center and a tooltip displays below. You can press the Ctrl key to access similar options for **Bottom** viewpoints.



A right-click menu offers access to additional controls and settings.

Choose **LookFrom Settings…** in the context menu to open the **Settings** dialog box and edit the **LookFrom** related system variables.

9.3 Direct modeling commands

BricsCAD direct modeling commands can create both solid and surface geometry. To distinguish between direct modeling commands and 'normal' commands, all direct modeling commands start with 'DM'.

Direct modeling commands are available in license levels Pro, BIM, Mechanical and Ultimate. Commands that create geometry are also available in BricsCAD Shape, along with basic modification commands such as DMPUSHPULL, DMCHAMFER and DMFILLET.

9.3.1 Creation commands:

DMEXTRUDE: creates 3D solids or surfaces by extruding open or closed 2D entities, faces of 3D solids, regions or closed boundaries. Optionally adds volume to solids or removes volume from solids.

DMREVOLVE: creates 3D solids or surfaces by revolution of open or closed 2D entities or regions about an axis.

DMTHICKEN: creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, faces of 3D solids, regions and wire entities.

DMCOPYFACES: copies a set of 3D solid faces forming a feature (e.g. hole, rib or protrusion) to a new position on the same or another 3D solid.

DMSTITCH: converts a set of region and surface entities that bound a watertight area to a 3d solid.

9.3.2 Modification commands:

DMCHAMFER: creates equal distance and variable chamfers between adjacent faces sharing a sharp edge.

DMDEFORMCURVE: deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

DMDEFORMMOVE: deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

DMDEFORMPOINT: deforms a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction. The selected faces are deformed as smooth as possible.

DMFILLET: creates a smooth fillet between adjacent faces sharing a sharp edge.

DMPUSHPULL: adds or removes volume from a solid by moving/offsetting a face.

DMROTATE: rotates solids, faces, edges, or vertices of a solid around an axis.

DMTWIST: modifies a 3D solid, surface or region by twisting a portion defined by two points around an axis.

9.3.3 Helper commands:

DMAUDIT: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces).

DMAUDITALL: fixes inconsistencies in 3d geometry supported by ACIS kernel (3D solids, surfaces) in drawings inserted as external references.

DMGROUP: creates group features, which are named persistent collections of faces and edges of 3D solids and surfaces.

DMSELECT: selects edges and faces of 3D solids or surfaces based on their geometric properties.

DMSELECTEDGES: selects the edges of faces and solids.

DMSIMPLIFY: simplifies the geometry and topology of 3d solid entities. The command removes unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance.

The direct modeling commands are available in:

- The **Model** tab on the Quad cursor menu when the cursor is over an entity or subentity or when a selection set exists.
- The **Direct Modeling** Menu.
- The **Direct Modeling**, **Fillet**, **Chamfer** and **Deform** toolbars.
- The **Home / Direct Modeling** ribbon panel (**Modeling** workspace).

Icon	Command	Applies to
		Any face of a 3D solid or surface.
		Cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface. Linear and circular edges of a 3D solid or surface. Whole 3D solids, surfaces, and other entities.
		Planar, cylindrical, spherical, conical, and toroidal faces of a 3D solid or surface. Whole 3D solids, surfaces, and other entities.

Icon	Command	Applies to
		3D solids, surfaces, and their faces.
		Surfaces.
		3D solids and surfaces.
		Planar faces of 3D solids and surfaces. This tool launches the UCS command with the Face option selected. ***Nota:*** When the Dynamic UCS feature is ON, the UCS aligns automatically to the solid face under the cursor and the UCS-icon changes accordingly.
	Solid mode	Closed 2D entities and detected closed boundaries. If the **Select detected boundaries** option of the SELECTIONMODES system variable is switched ON, boundaries enclosed by 2D linear entities in the XY-plane of the current UCS are detected automatically. Planar faces of 3D solids and surfaces. Closed 2D entities and detected closed boundaries on a planar face of a solid.
	Solid mode	Closed 2D entities and detected closed boundaries.* Planar faces of solids and surfaces. Closed 2D entities and detected closed boundaries on the planar face of a solid.
	Extract Faces	Surfaces and faces of 3D solids.
		A set of 3D solid faces.
		Faces of 3D solids and surfaces. Surfaces. Wire entities; lines, polylines, circles, ellipses, arcs, helices, and splines.
	Constant radius mode	Sharp edges of a 3D solid or surface.
	Variable radius mode	Sharp edges of a 3D solid or surface.

Icon	Command	Applies to
		Sharp edges of a 3D solid or surface.
	Advanced Mode	Sharp edges of a 3D solid or surface.
		Faces of a solid or a surface. Whole 3D solids and surfaces. Free edges of a surface (a surface edge is called "free" if it has only one adjacent face) Edges between coplanar faces.
	Surface mode	3D curves and edges of 3D solids and surfaces.
	Surface mode	3D curves and edges of 3D solids and surfaces.
		Faces of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		Edges of a 3D solid or a surface.
		3D solids, surfaces or regions.

9.4 Direct modeling operations

9.4.1 Controlling the coordinate system

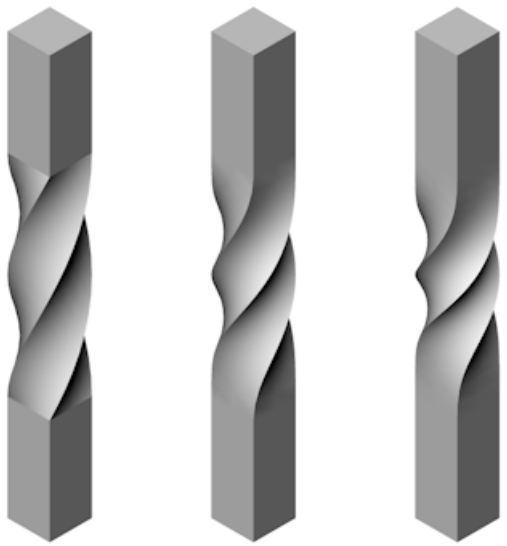
Since entities are created in the XY-plane of the current coordinate system (UCS), it is necessary to align the UCS with the 3D solid face. You have two options here:

- The **Align UCS** tool ( ) launches the UCS command with the **Face** option selected.
- Click the **DUCS** field in the **Status Bar** to toggle the dynamic UCS feature ON and OFF. When ON, the UCS aligns automatically to the solid face under the cursor when executing a command and the UCS-icon changes accordingly. Press the Tab key to select an obscured face. Press the Shift key to temporarily lock the UCS, which allows you to work outside the hovered entity or face. Press the Shift key again to unlock it.

9.4.2 Selecting geometry

- Hit the Tab key to select obscured geometry.

- The SELECTIONMODES system variable controls which subentities highlight: edges, faces, and/or closed boundaries. Hold down the Ctrl key to overrule SELECTIONMODES, and select the subentity under the cursor.
- Window selection: to select the faces or edges of 3D solids and surfaces, press the CTRL key to cycle the options. The icon next to the cursor indicates what will be selected.
- The DMSELECT command allows you to select edges and faces based on their geometrical and topological properties:
  - Edges of a particular length.
  - Edges that form a concave/convex network.
  - Edge loops on the border of a set of connected faces.
  - Faces of a particular area.
  - Smoothly connected faces (G1 - tangent faces, or G2 - curvature continuity).



G0: No continuity (left), G1: Tangent Faces (middle), and G2: Curvature continuity (right)

  - Fillets of a specific radius.
  - Cylindrical protrusions of a specific radius.
  - Protrusion or depression features; pockets, extrusions, or ribs.

The value of a particular length, area, or radius can be taken from a selected edge/face.

9.4.3 Moving faces to control dimensions

A dynamic dimension displays with the DMPUSHPULL command.

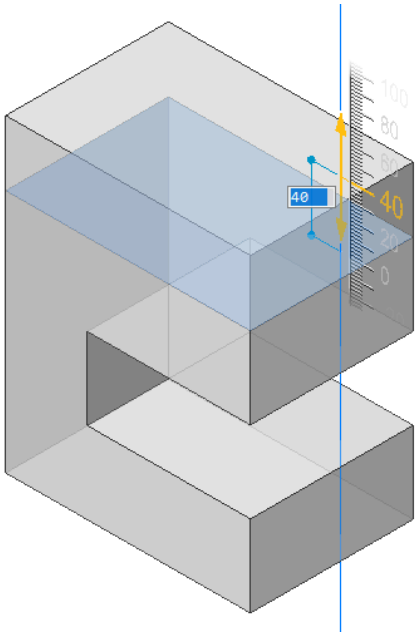
Make sure the **Quad** and **Dynamic Dimensions** are switched ON.

1 Hover the cursor over the face you want to move.

1 The face highlights.

2 Choose **Push/Pull** ( ) in the **Model** tab of the Quad.

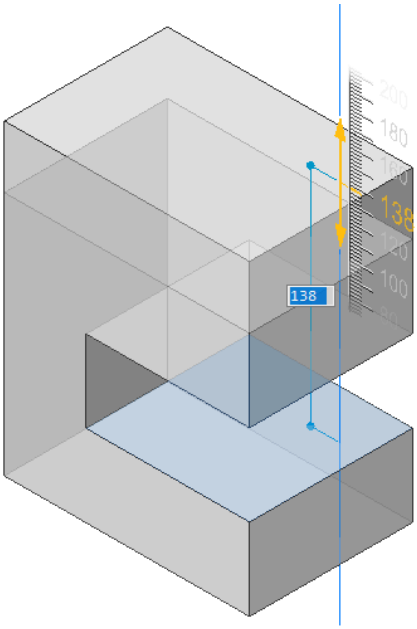
The selected face moves with the cursor and a dynamic dimension field displays the relative distance along with a ruler.



When the cursor is on the ruler, the face moves with fixed intervals. Zoom in or out to increase or decrease the interval.

When the cursor is outside the ruler, the face moves freely.

3 (Option) repeatedly hit the Tab key to see the distance with respect to an internal face, parallel to the selected face (left) or the bottom face (right).



Press and hold the Shift key and hit the Tab key to reverse the display order of reference faces.

4 Do one of the following.
   - Type a value in the dynamic dimension field, then hit Enter.
   - Using the ruler: click when the desired value displays.

9.4.4 Making holes

Method 1:

1 Create the subtractor solid,

2 Use the SUBTRACT command.

3 Depending on the value of the DELETETOOL system variable, the subtractor solid is either deleted or retained.

Method 2:

1 Create the outline of the opening on the 3D solid face.

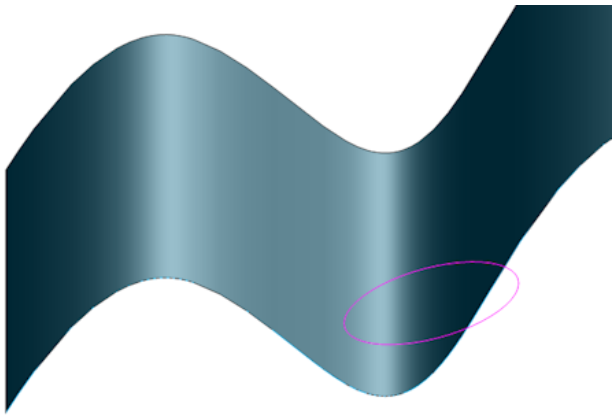
2 Place the cursor over the outline, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

3 Move the cursor inside the solid to make the opening.

4 Do one of the following:
   - Specify a point.
   - Type a value.

To make a hole in a surface

1 Create a 2D closed entity, e.g. an ellipse.

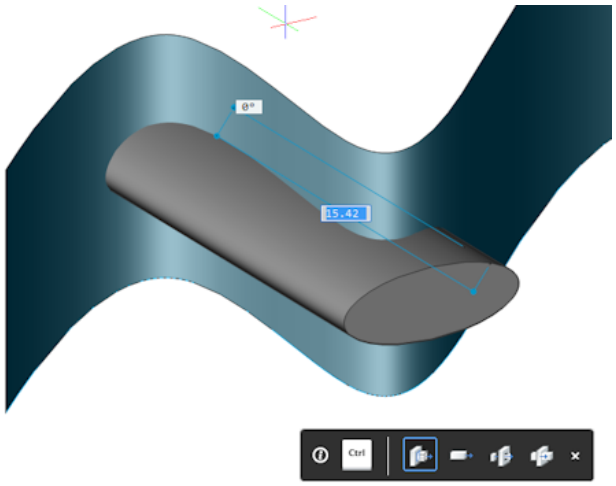


2 Place the cursor over the 2D closed entity, then choose **Solid Extrude** ( ) (DMEXTRUDE command) in the **Model** tab of Quad.

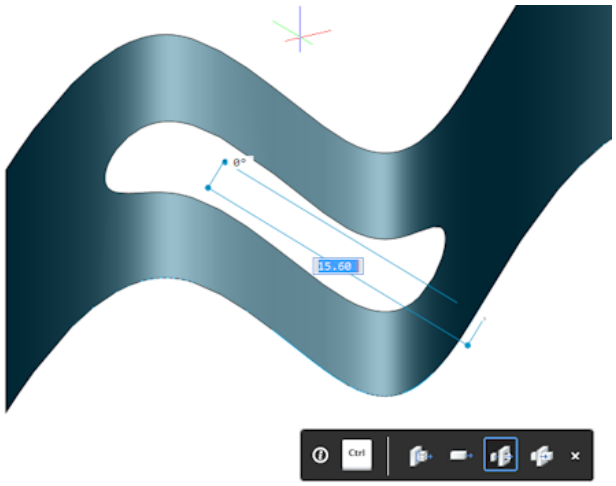
3 Extrude the 2D entity in the direction of the surface.

4 The default extrusion direction is perpendicular to the plane of the 2D entity.

The **Hotkey Assistant** (HKA) appears at the bottom of the screen.



5 Hit the CTRL key twice to select the **Subtract** option of the DMEXTRUDE command.



6 Click to create the opening.

9.4.5 Deleting features

The **Feature** option of the DMSELECT command allows to easily select all faces of a hole, depression or protrusion.

In the **Mechanical** workspace, the **Protrusion** and **Depression** tools are available in the **Select** panel of the **Solid** ribbon tab and the **Select** tab of the Quad.

The tools sit in the **Sheet Metal** toolbox. See **Available Tools** in the **Customize** dialog box.

To delete a depression (hole or pocket) or a protrusion feature (extrusion or rib)

1 Click the **Protrusion** ( ) or **Depression** ( ) tool button in the **Select** panel of the **Solid** ribbon tab or from the **Select** command group of the Quad.

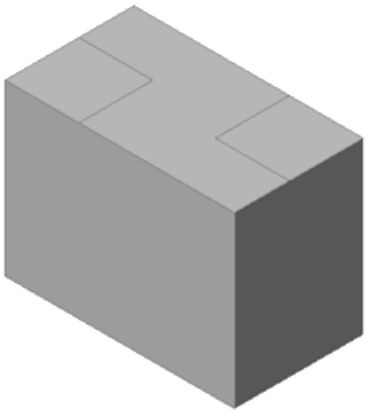
2 Select one or more faces of the feature.

3 All adjacent faces of the feature are detected.

4 Click the **Erase** ( ) tool button.

9.4.6 Removing edges between coplanar faces

When two or more faces are made coplanar, e.g. with the DMPUSHPULL command, it might be necessary to remove the shared edges using the DMSIMPLIFY command.

	
Two depressed faces	Two depressed faces moved to the top face, to create three coplanar faces.

1 +Launch the DMSIMPLIFY command.

1 **You are prompted**: Select entities to simplify or [Entire model/SEttings/selection options (?)] <Entire model>:

2 Select the solid, then press Enter.

3 The edges are removed.

9.4.7 Copying a Feature

The DMCOPYFACES command allows you to copy one or more features from a 3D solid (depressions - holes, grooves, and pockets - or protrusions - extrusions and ribs), to a new location on the same or a different 3D solid.

Make sure dynamic UCS is enabled.

1 Select all faces of the entire feature(s).

2 Click the **Copy Faces** ( ) tool:
- on the ribbon: **Solid | Edit | Copy Faces**.
- on the **Direct Modeling** toolbar.
- in the menu bar: **Model | Direct Modeling | Copy Faces**
- in the **Model** tab on the Quad

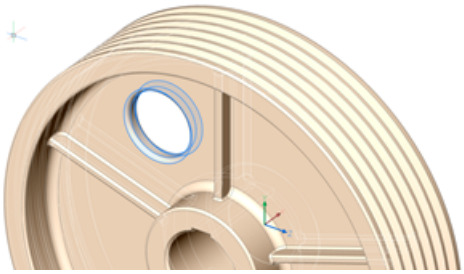
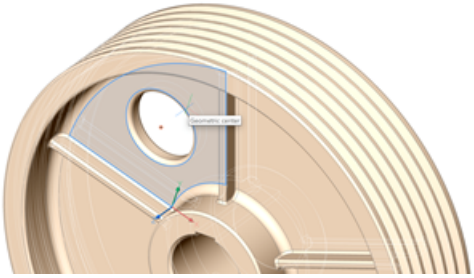
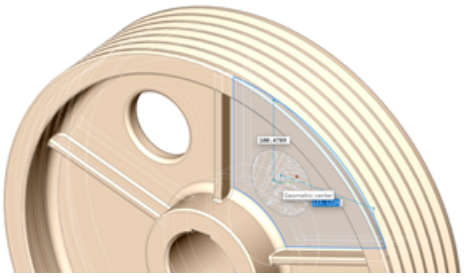
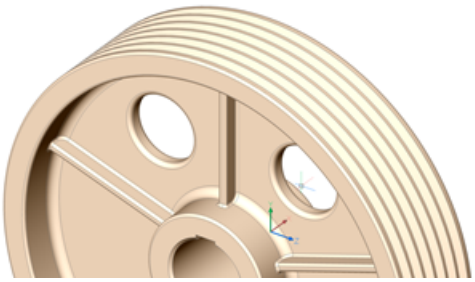
3 Specify the base point.

4 **You are prompted:** Select an insertion point or [Rotate/mUltiple/MOde]:

5 (Optional):
   - Use the **Rotate** option to rotate the faces around the Z axis of the UCS.
   - Use the **Multiple** option to insert the multiple copies of a feature.

6 Specify the insertion point.

7 The feature is copied from the base point to the insertion point with dynamic UCS.

	
Select the faces of the entire feature	Specify base point
	
Specify the insertion point	The feature is copied

9.4.8 Deformable modeling

Use the tools on the **Deform** toolbar, to deform a set of adjacent faces on a 3D solid or a surface, to create complex aesthetic shapes.

The faces are always deformed if:

- The external boundary (= edges, which are adjacent to only one of the selected faces) remains unchanged.
- The internal boundaries (= edges, which are adjacent to two of the selected faces) are deformed together with the faces.
- The initial continuity between deformed faces (G1 – tangent faces, or G2 – curvature continuity) is kept when deformed.

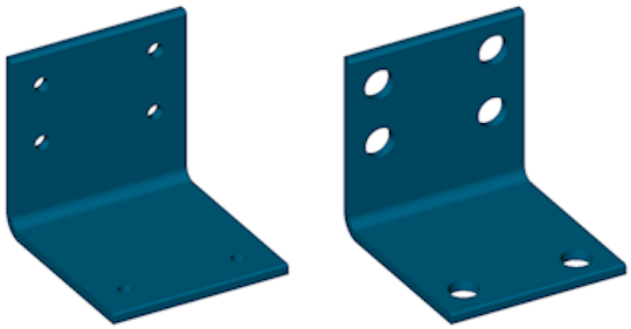
For a twist deformation, it is possible to specify the continuity between the deformed and the fixed portion of the entities.

Deformation can be defined in several ways:

- DMTWIST command: Twist the whole entity or a portion.
- DMDEFORMPOINT command: Move a point on a face in any direction in 3D space.
- DMDEFROMMOVE command: Move or rotate one or more edges.
- DMDEFORMCURVE command: Replace an edge with a given 3D curve.

9.5 Design intent recognition

Direct modeling operations allow you to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized.



Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations.

To define the design intent recognition, click the buttons on the **Design Intent** toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active.

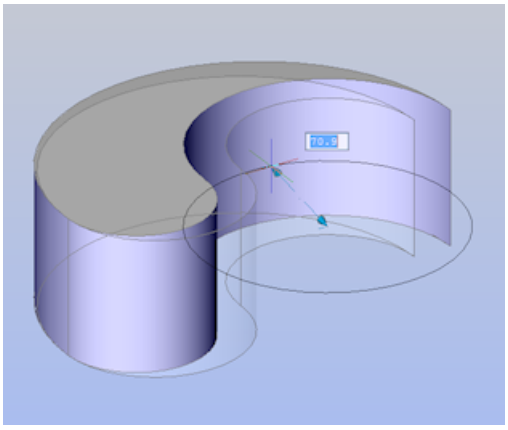
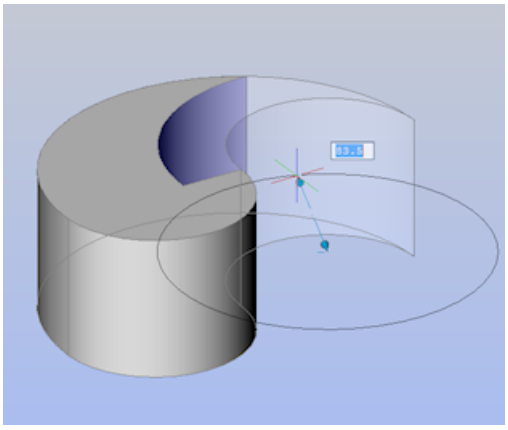
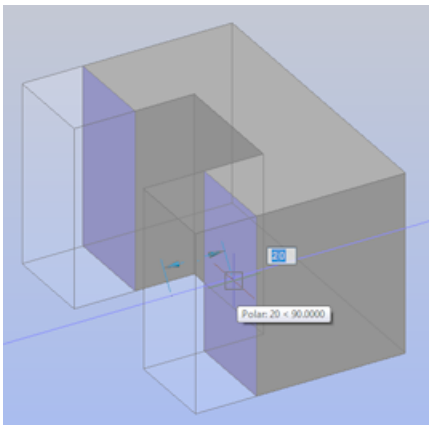
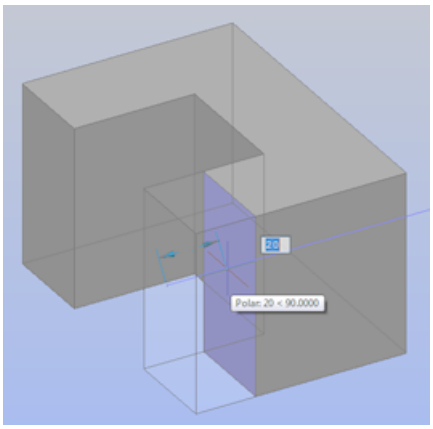
- tangent surfaces (planes, cylinders and cones) ( )
- coincident planes ( )
- parallel planes ( )
- perpendicular planes ( )
- cylinders perpendicular to planes ( )
- coaxial surfaces (cylinders and cones) ( )
- surfaces of equal radius (cylinders and spheres) ( )
- vertices between 4 or more faces ( )
- edges between coincident faces ( )
- switch off/on all ( )

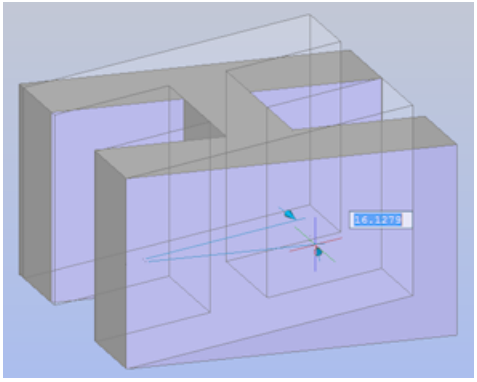
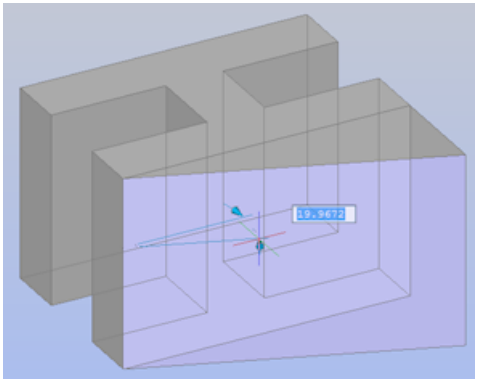
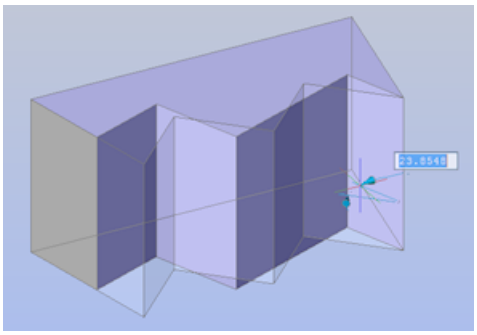
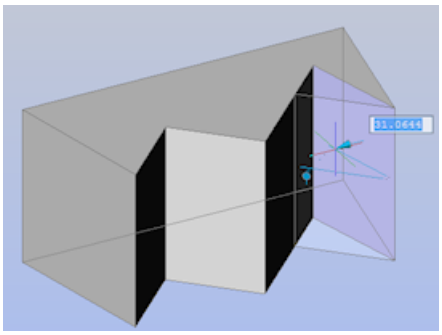
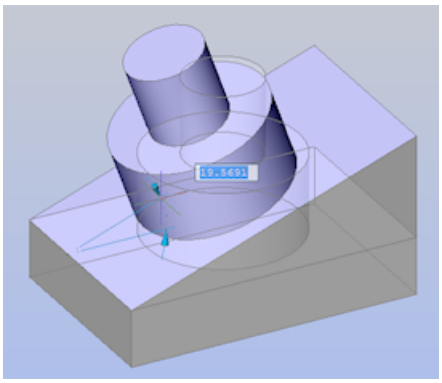
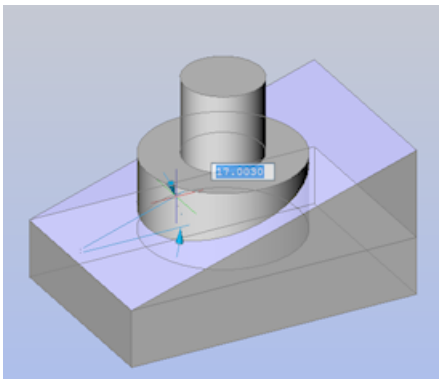
The design intent options can also be set by editing the DMRECOGNIZE system variable in the **Settings** dialog box.

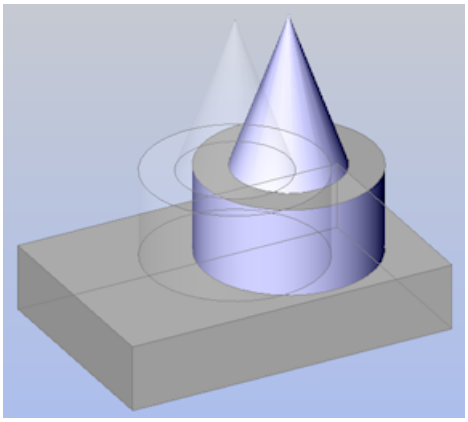
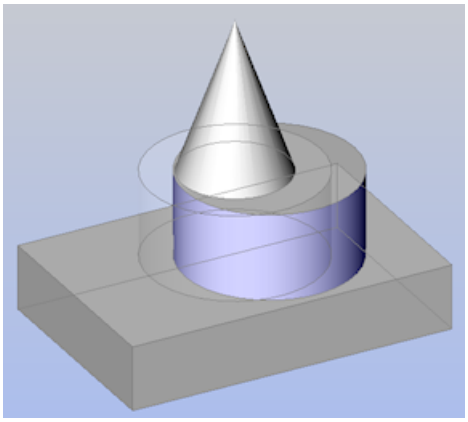
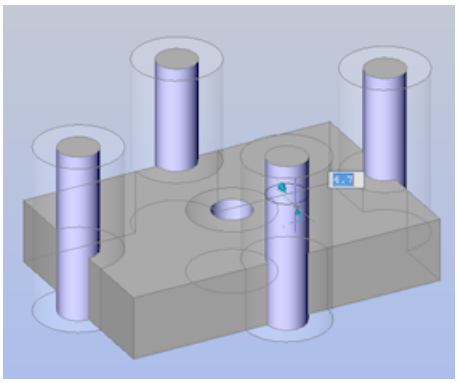
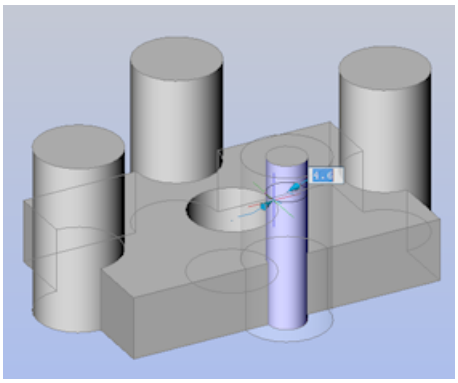
Click the **Options** tool button ( ) on the **3D Constraints** toolbar to set the **Automatic 3D geometry constraints recognition**. The state of tool buttons on the **Design Intent** toolbar (pressed or unpressed) is adjusted accordingly.

Nota: 3D Constraints which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

The following examples illustrate the difference in the behavior whether automatic 3D geometry constraints are recognized or not.

	
tangent surfaces recognized ( )	tangent surfaces not recognized ( )
	
coincident plane recognized ( )	coincident planes not recognized ( )

	
parallel planes recognized ( )	parallel planes not recognized ( )
	
perpendicular planes recognized ( )	perpendicular planes not recognized ( )
	
perpendicular cylinders recognized ( )	perpendicular cylinders not recognized ( )

	
coaxial surfaces recognized ( )	coaxial surfaces not recognized ( )
	
equal radius recognized ( )	equal radius not recognized ( )

9.6 Working with sections

The SECTION command creates section planes of 3D solids; the results are region entities.

The SECTIONPLANE command creates a section entity that creates sections of 3D solids, surfaces, and meshes.

The LIVESECTION command toggles the **Live Section** property of a section plane.

The SECTIONPLANESETTINGS command defines the properties of a section plane entity in the **Drawing Explorer - Section Planes** dialog box.

The SECTIONPLANETOBLOCK command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

Nota:

When using a BIM license level:

- The BIMSECTION command creates **BIM section** entities, which are Section entities with extended properties, such as type (plan, section, elevation) and initial scale of the 2D section result.

- The BIMIFY command can assign BIM properties to a Section entity.

9.6.1 Difference between Live Section and Clip Display

In both statuses all drawing content is sectioned and clipped-away parts are no longer displayed.

Live Section: entities in a drawing are temporarily modified, e.g. clipped solids are replaced by temporary solids with a different shape.

Clip Display: the display of entities is clipped. Entities are not replaced by temporary versions. Clip Display allows to continue using all modeling and editing operations on all drawing content. As opposed to the Live Section status, the Clip Display status can be set for multiple section entities simultaneously.

9.6.2 Align a section plane to a face

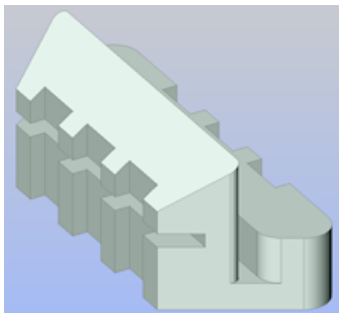
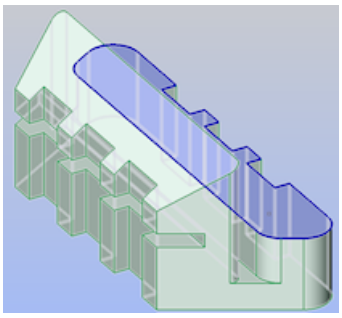
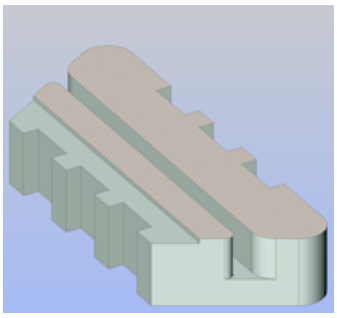
1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Select Face** option.

You are prompted: Select entity:

3 Select the face of a solid you want to align the section plane to.

		
Solid	Select face	Section plane aligned to face

4 The section plane is aligned to the selected face.

5 The **Clip Display** property of the section entity is switched **On** automatically.

6 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.3 To create a vertical section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

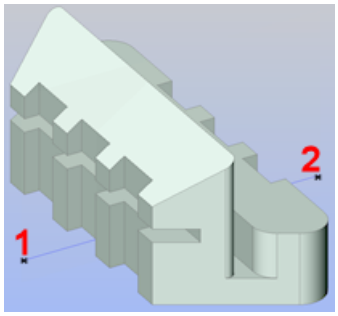
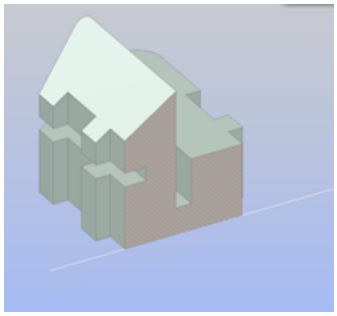
2 Specify two points.

3 The section plane is a vertical plane created through these points.

The part of the solid at the right hand side (with respect to the draw order of the points) of the section

plane is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

	
Specify two points	Vertical section plane

4 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.4 To create a horizontal section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

Nota: If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

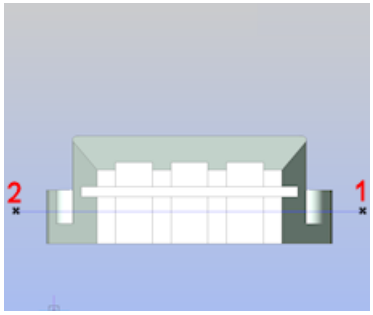
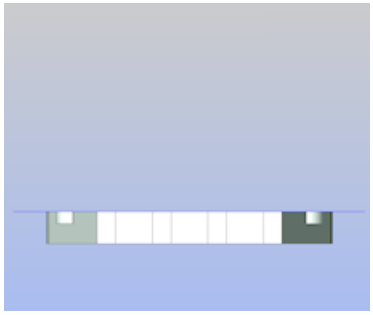
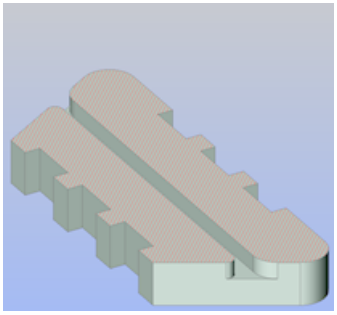
5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Specify two points.

7 The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.

8 The part of the solid at the right hand side (with respect to the draw order of the points) of the section plane is cut away.

9 The **Clip Display** property of the section entity is switched **On** automatically.

Specify two points	The part of the solid at the right hand side of the section plane is cut away	Horizontal section plane

10 (option) Edit the properties of the section plane.

9.6.5 To create a vertical jogged section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Draw** option.

You are prompted: Specify start point:

3 Specify the first point (1).

4 **You are prompted**: Specify next point:

5 Specify the second point (2):

6 **You are prompted**: Specify next point or ENTER to finish:

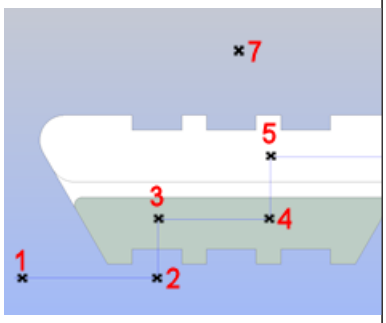
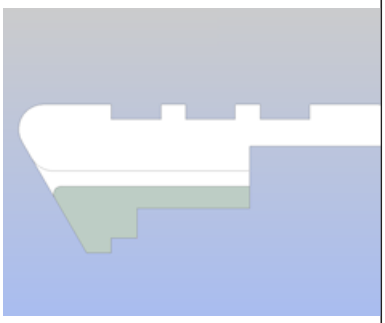
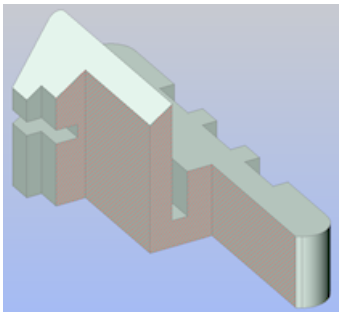
7 Specify the other points (3 - 6), then right click or press Enter to stop.

8 **You are prompted**: Specify the direction of the section entity:

9 Click a point (7) to specify the direction of the section entity.

10 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Vertical jogged section plane

11 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.6 To create a horizontal jogged section plane

1 Choose **Front** in the **Look From** widget.

2 Align the UCS to the front view:

3 Use the **View** option of the UCS command.

Nota: If UCSORTH0 = ON and NAVVCUBEORIENT = 0, the UCS aligns with the view automatically.

4 Launch SECTIONPLANE command.

5 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

6 Choose the **Draw** option.

You are prompted: Specify start point:

7 Specify the first point (1).

8 **You are prompted**: Specify next point:

9 Specify the second point (2).

10 **You are prompted**: Specify next point or ENTER to finish:

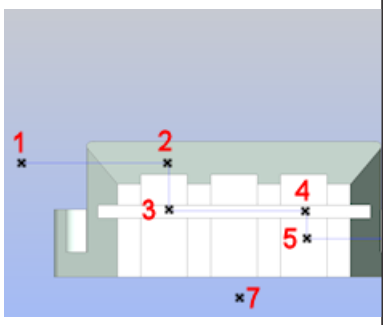
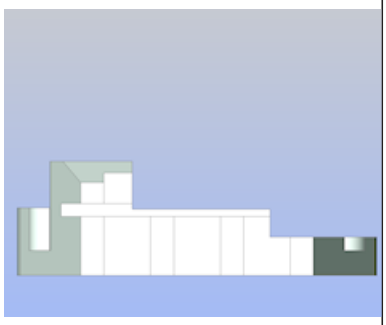
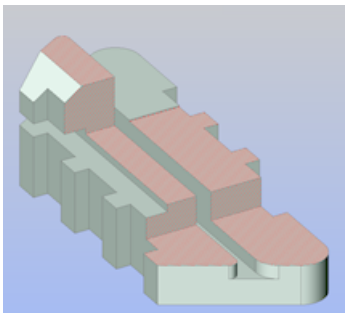
11 Specify the other points (3 - 6) , the right click or press Enter to stop.

12 **You are prompted**: Specify the direction of the section entity:

13 Click a point (7) to specify the direction of the section entity.

14 The part of the solid opposite to the direction of the section entity is cut away.

The **Clip Display** property of the section entity is switched **On** automatically.

		
Specify points	The part of the solid opposite to the direction of the section entity is cut away	Horizontal jogged section plane

15 (option) Edit the properties of the section plane in the **Properties** panel.

9.6.7 To create an orthographic section plane

Nota: The WCS must be the current coordinate system.

1 Launch SECTIONPLANE command.

1 **You are prompted**: Specify start point or [select Face/Draw/Orthographic]:

2 Choose the **Orthographic** option.

You are prompted: Align section to [Front/bAck/Top/Bottom/Left/Right] <Top>:

3 Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the Command line.

4 Orthographic section planes are created through the center of the solid, parallel to the XY-plane (Top and Bottom), YZ-plane (Left and Right) or XZ-plane (Front and Back) of the current coordinate system.

The **Clip Display** property of the section entity is switched **On** automatically.

5 (option) Edit the properties of the section plane in the **Properties** panel.

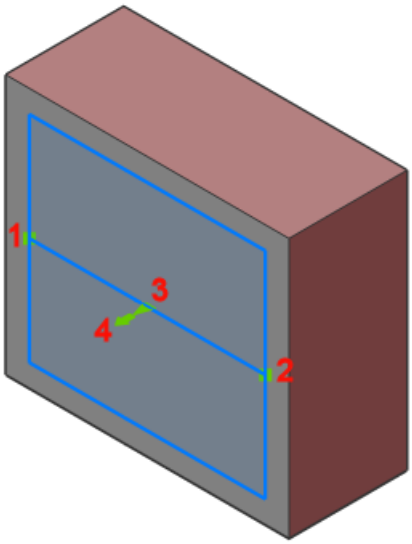
9.6.8 To modify the size and position of a section plane

1 Select the section plane entity in the drawing.

1 Depending on the current state of the section plane entity a number of grips display.

2 (option) Switch on the **Clip Display** property of the section plane in the **Properties** panel to see the result of the modifications dynamically.

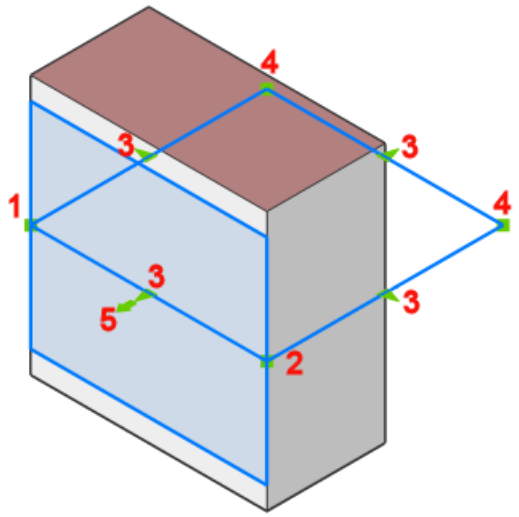
3 (option) Set the **State** property to **Plane**:



Four grips display:

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
- 2 ( ) = through point: redefines the through point of the section plane.
  The section plane is rotated around the origin point (1).
- 3 ( ) = midpoint: moves the section plane parallel in a direction perpendicular to the section plane.
- 4 ( ): click to flip the view direction of the section plane.

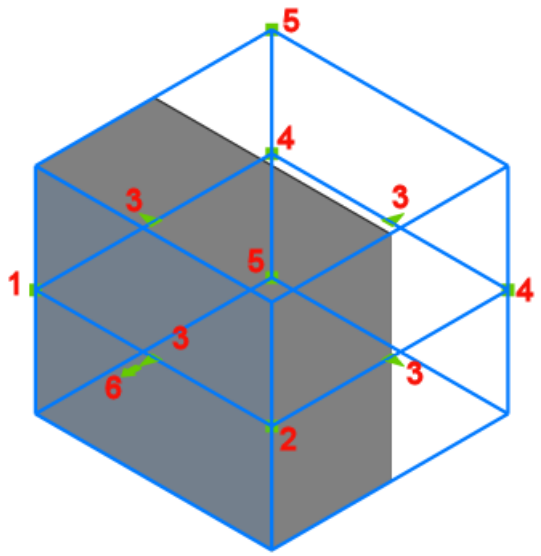
4 (option) Set the **State** property to **Boundary**:



Ten grips display.

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.
- 2 ( ) = through point: redefines the through point of the section plane.
  The section plane boundary is stretched and rotated around the origin point (1).
- 3 ( ): stretches the section plane boundary parallel.
- 4 ( ): modifies the shape of the section plane boundary.
- 5 ( )

5 (option) Set the **State** property to **Volume**:



Twelve grips display.

- 1 ( ) = origin point: moves the position of the section plane in the XY-plane.

- 2 ( ) = through point: redefines the through point of the section plane.
  The section plane volume is stretched and rotated around the origin point (1).
- 3 ( ): stretches the section plane volume parallel.
- 4 ( ): modifies the shape of the section plane volume.
- 5 ( ): modifies the height of the section plane volume.
  Polar Tracking must be on to move grip 5.
- 6 ( ): click to flip the view direction of the section plane.

9.6.9 To save a section

The SECTIONPLANETOBLOCK command allows to:

- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

9.7 Create a TIN Surface

9.7.1 Commands

TIN, TINASSIGNIMAGE, TINPROJECT

9.7.2 Overview

In BricsCAD, topographical surfaces are created using the TIN Surface tool (triangulated irregular network) or the TIN command.

A TIN surface uses two types of entry data: points and breaklines (polylines).

- Points contain X, Y coordinates and Z values. All points are used to make a connection with the two nearest neighbors to create triangles. The surface triangulation is based on the Delaunay algorithm, which ensures that no point is inside the circumcircle of any triangle.
- Breaklines represent linear infrastructure features like curbs, retaining walls, etc. These lines also define the edges of triangles. Breaklines can be created from linear entities such as line, polyline, arc or circle.
- Boundaries are the features to define bounded 3D surfaces. The surface boundaries can be created with closed polylines. Defining outer boundaries on a large 3D surface improves the performance of a TIN Surface plan.

Creating a TIN Surface is a process that needs to be followed to map a particular area in detail. On BIM models, these topographic surfaces are used to represent the site. The topographical surface can be generated based on drawing entities, data imported from a comma-delimited file format (CSV) or text file format (TXT), and points placed at different elevations, created from Civil 3D surface.

9.7.3 Setting the TIN Surface preferences

Civil

TIN Surface

Arc approximation mid-ordinate distance	0.01
Allow breakline crossings	☑
Elevation at breakline crossings	[0] Minimal

Civil General

Associativity	0x000F (15)
1	☑ TIN Surface associativity
2	☑ Grading associativity
4	☑ TIN Volume Surface associativity
8	☑ Corridor associativity

Arc approximation mid-ordinate distance

Nota: Mid-ordinate distance is the maximal distance between arc and chord (straight) segment which is used for arc approximation.

Defines how many points along the curve you want to add. When you have polylines with curves, this parameter allows you to tessellate the arcs in the polyline. The default value of this parameter is variable in BricsCAD predefined templates. For instance, the distance value is set to 0.01 in BIM-m, 1in BIM-cm, 10inBIM-mm, and 0.4in BIM imperial templates.

Allow breakline crossings

Defines if crossings breaklines are allowed. By default, it is set to **On**. If breakline crossings are allowed, intersections between breakline segments are calculated and added as points to the TIN Surface.

Elevation at breakline crossings

Defines how elevation at crossing breaklines should be determined. There are three options (e.g., minimal, maximal, and average) for defining breaklines elevation at the intersection.

Associativity

Enable/disable grading, surface and corridor associativities.

By default, the system variable is set to 15.

0: Neither TIN surface, grading nor corridor associativity is enabled.
1: Enables TIN surface associativity.
2: Enables grading associativity.
3: Both TIN surface and grading associativities are enabled.
4: Enables TIN volume surface associativity.
5: Both TIN surface and TIN volume surface associativities are enabled.
6: Both grading and TIN volume surface associativities are enabled.
7: TIN Surface, Grading and TIN volume surface associativities are enabled.
8: Enables corridor associativity.
9: Both TIN surface and corridor associativities are enabled.
10: Both grading and corridor associativities are enabled.
11: TIN surface, grading and corridor associativities are enabled.
12: Both TIN volume surface and corridor associativities are enabled.
13: TIN surface, TIN volume surface and corridor associativities are enabled.
14: Grading, TIN volume surface and corridor associativities are enabled.

15: All TIN surface, grading, TIN volume surface and corridor associativities are enabled.

9.7.4 Properties of a TIN Surface

General	
Handle	93
Color	Green
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Material	ByLayer
Data	
Name	Site
Description	School playground
TIN Visibility	
Boundary line	Off
Points	Off
Triangles	On
Contours	Off
TIN Contours	
Major contours interval	10 m
Minor contours interval	1 m
Major contours color	Red
Minor contours color	Green
TIN Statistics	
Number of points	4264
Number of triangles	8230
Minimum elevation	4.4 m
Maximum elevation	18.9192 m
2D area	10363.7386 m²
3D area	11990.9728 m²

TIN Visibility

Boundary line
Change boundary line visibility.

Points
Change points visibility.

Triangles
Represents how the elevation points are connected to each other. It follows the Delaunay triangulation method where breaklines are not included in densification.

Contours
Represents the joining of points of equal elevation within a line.

TIN Contours

Major contours interval
Sets the major contours interval.

Minor contours interval
Sets the minor contours interval.

Major contours color
Defines the color of the major contour.

Minor contours color
Defines the color of the minor contour.

TIN Statistics

The number of points
Displays the number of points that define the TIN surface.

The number of triangles
Displays the number of non-overlapping triangles in the TIN surface.

Minimum elevation
Displays the minimum elevation value found in the TIN surface.

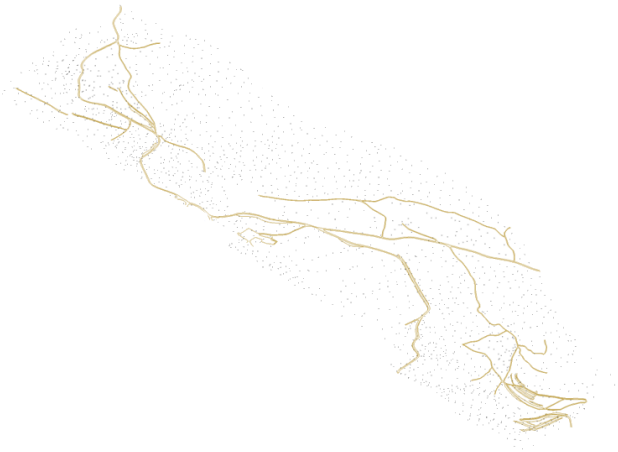
Maximum elevation
Displays the maximum elevation value found in the TIN surface.

2D area
Displays the 2D area calculated from the flat TIN surface.

3D area
Displays the 3D area that is calculated from the TIN surface.

9.7.5 Procedure: Creating a TIN Surface from a selection set of points and polylines

1 Open the drawing file that contains a set of points and polylines.

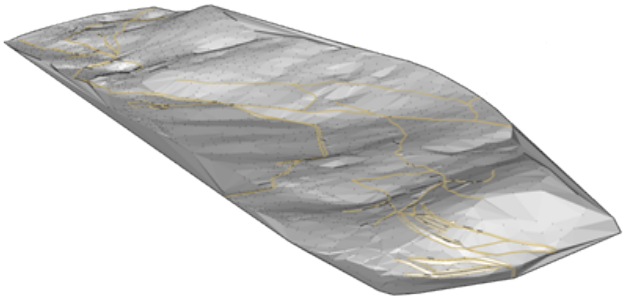


2 Launch the TIN command.

3 Select the entities in the drawing area to create a toposurface.

4 Press ENTER.

The toposurface is created from selected entities. Points are added to the surface as points, 3D polylines are added to the surface as breaklines.



9.7.6 Procedure: Creating a TIN Surface from an imported points file (.txt or .csv)

1 Launch the TIN command.

2 Select **Import from file** in the Command line.

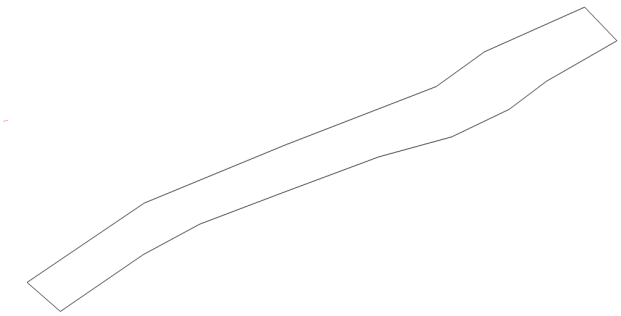
3 Choose the point data file you want to open or choose multiple files to import all at once.

4 Click Open to import the toposurface.

Nota: If the toposurface is not shown in the current viewport, use zoom extents to display it to the extents of the entities.

9.7.7 Procedure: Creating a TIN Surface of a selected area using Clip polygon

1 Open the drawing file that contains closed polyline.



2 Launch the TIN command.

3 Select the **cLip polygon** option in the Command line.

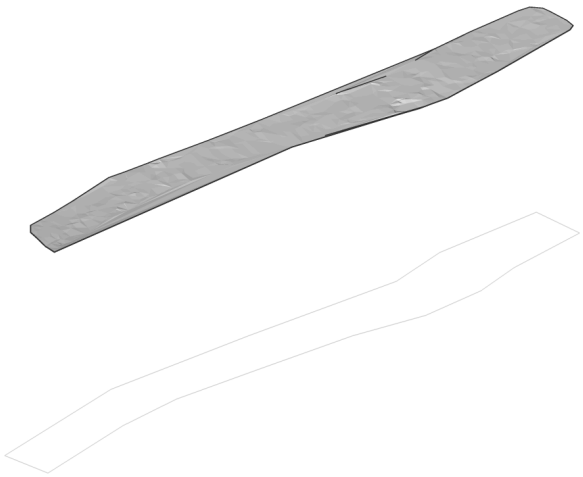
4 Select the closed polyline to use.

5 Specify if the crossed breaklines should be removed.

Nota: If the surface we are going to make contains breaklines, we define what should happen to the breaklines crossing the clipped polygon.

6 Select any of the options, **Import from file**, **Create from Faces**, **Create from civil 3D surface**.

Nota: Clip polygon works as a projection on the TIN surface. The TIN surface is created only in the area defined by the clip polygon.



9.7.8 Procedure: Creating a TIN Surface by placing points

1 Open a drawing.

2 Launch the TIN command.

3 Select the **place Points** option in the Command line.

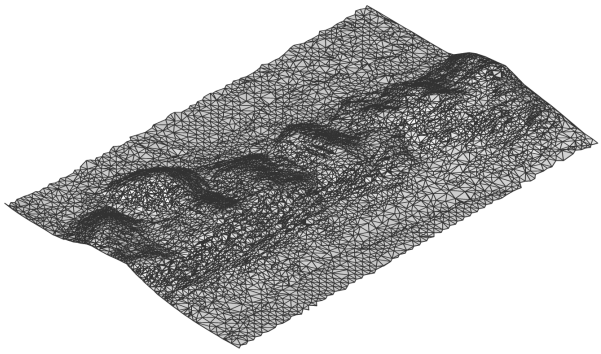
4 Pick a point where you want to start creating a TIN surface.

Specify the elevation value of each of those points, then press ENTER to accept.

Nota: At least three points are needed to create a TIN surface.

9.7.9 Procedure: Creating a TIN Surface from Faces

1 Open a drawing file that contains 3D faces.



2 Launch the TIN command.

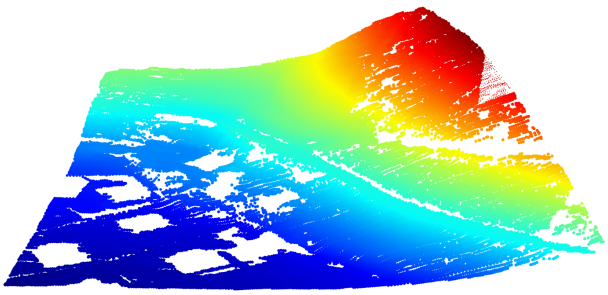
3 Select the **create from Faces** option in the Command line.

4 Select desired 3D faces.

5 Press Enter.

9.7.10 Procedure: Creating a TIN Surface from a Point Cloud

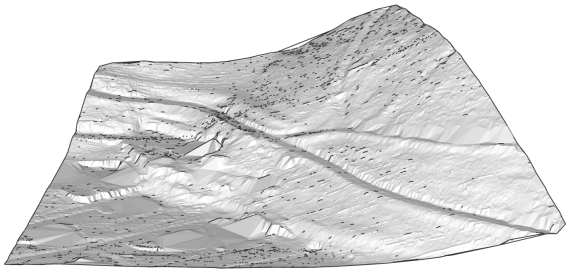
1 Open the drawing file that contains a point cloud.



2 Launch the TIN command.

3 Select the point cloud in the drawing area to create a toposurface.

4 Press ENTER. The toposurface is created from the point cloud.



9.7.11 Procedure: Creating a TIN Surface from a Civil 3D surface

1 Open a drawing with a Civil 3D surface.

2 Launch the TIN command.

3 Select **Create from civil3d surface**.

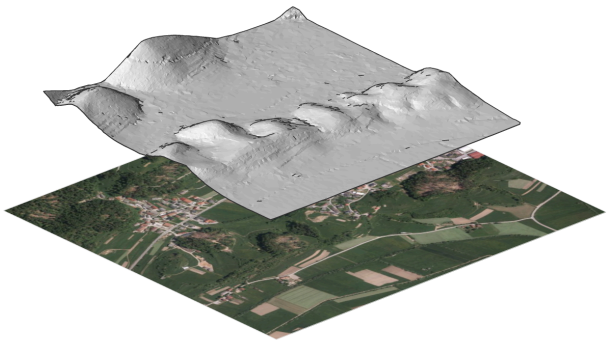
4 Select the Civil 3D surface. A TIN surface is created.

9.7.12 About TIN Assign Image

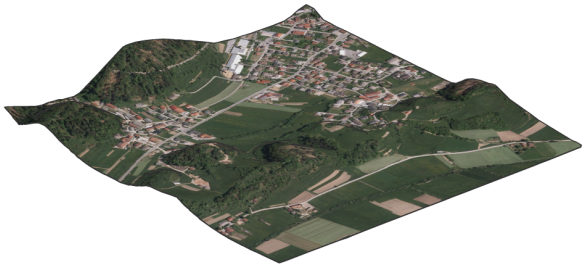
1 Open the drawing file that contains a TIN surface and an image.

2 Launch the TINASSIGNIMAGE command.

3 Select the TIN surface.



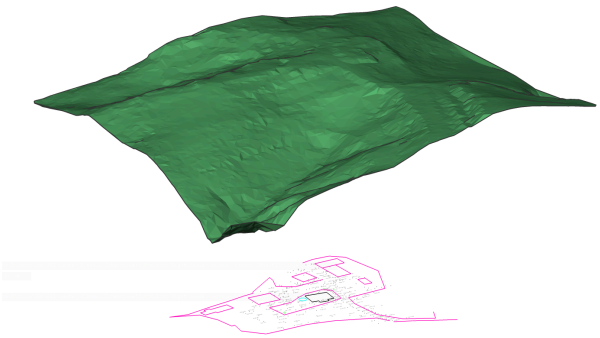
4 Select the raster image.



Nota: If the image is not shown on the TIN surface, change the Visual Style to Modeling or Realistic and set TIN surface triangles to ON.

9.7.13 About TIN Project

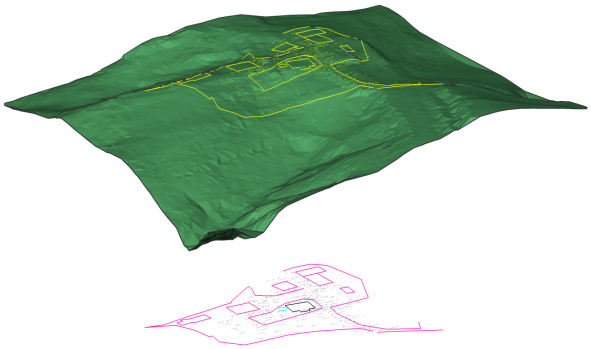
1 Open the drawing file that contains the TIN surface and the entities (points, blocks, text, lines, polylines, circles…).



2 Launch the TINPROJECT command.

3 Select the entities to project.

4 Select if the original entities are to be erased.



9.8 TIN Surface Edit

9.9 TIN Surface Direct Modeling

9.9.1 Commands

TINEXTRACT, TINMODIFY, TINMERGE

9.9.2 Overview

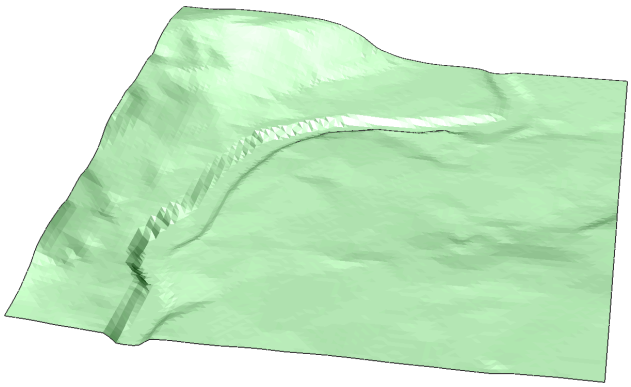
In BricsCAD, topographical surfaces created as a TIN Surface, can be adjusted using different tools.

9.9.3 About TINEXTRACT

The TINEXTRACT command is used to generate a solid, mesh, border, faces and points from a TIN surface. You can exchange these entities in other applications, to further analyze the surfaces. The solids or meshes can be created in different ways: as a vertical offset of the TIN Surface, as a model between two TIN Surfaces or as a model between a fixed elevation plane and the TIN Surface.

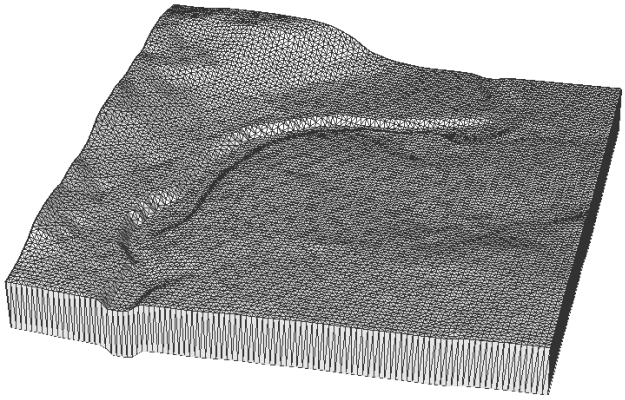
9.9.4 Procedure: Creating a solid from a TIN surface using option Elevation

1 Select a TIN Surface and type TINEXTRACT in the Command line.



2 Select the **Solid** option and press Enter.

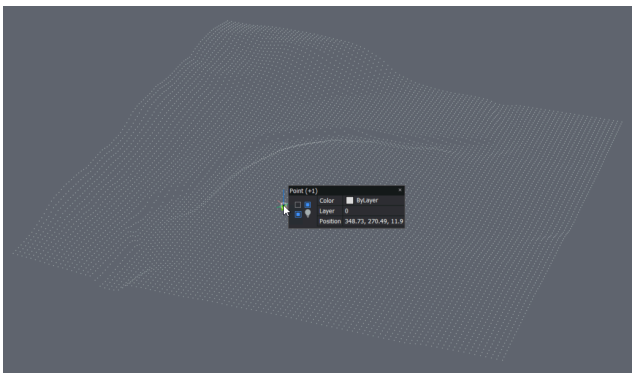
3 Choose the elevation height and press Enter.



9.9.5 Procedure: Creating Points from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

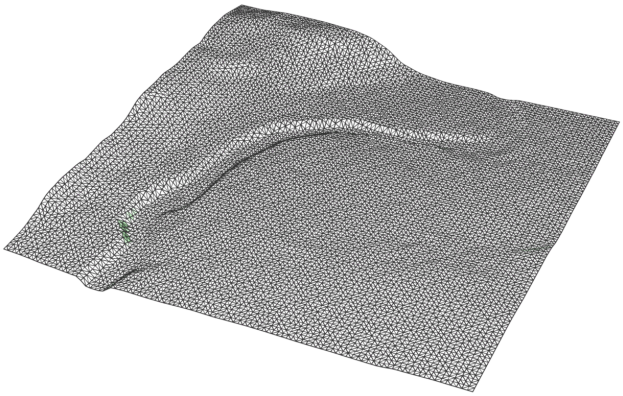
2 Select the **Points** option in the Command line.



9.9.6 Procedure: Creating Faces from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Faces** option.

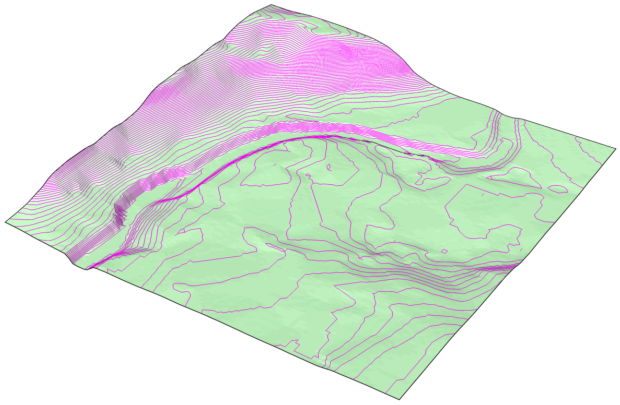


9.9.7 Procedure: Creating a Contours from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Contours** option in the Command line.

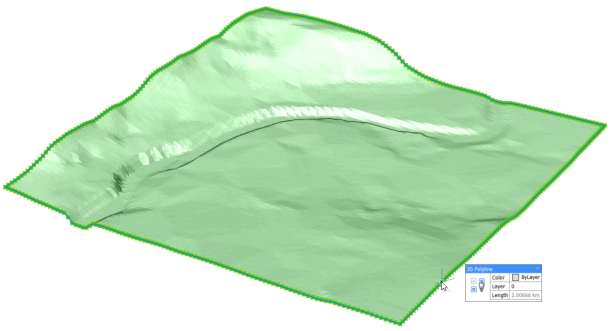
3 Type in **A** for All and press Enter.



9.9.8 Procedure: Creating a Border from a TIN surface

1 Select a TIN Surface and type TINEXTRACT in the Command line.

2 Select the **Border** option in the Command line.

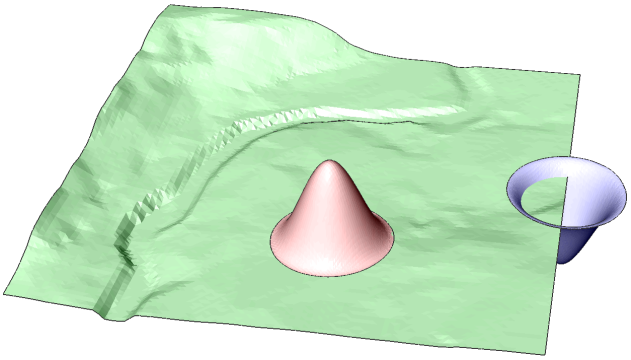


9.9.9 About TIN Merge

Different TIN Surfaces can be merged into one by using the TINMERGE command. A merge is executed between a base surface and one or more surfaces to merge with.

9.9.10 Merging TIN Surfaces

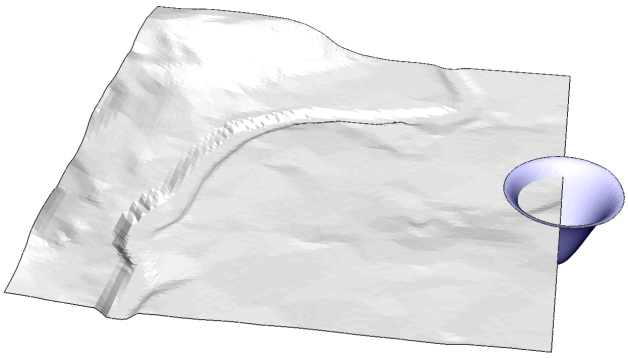
1 Type TINMERGE in the Command line.



2 Choose a base surface. In this example, we choose the green surface as the base surface.

3 Choose to keep or delete the original surfaces.

Nota: Selecting a different baseline, results in a different merge. It is important to remember that the surface to merge replaces the part of the base surface that it covers.



The first example shows the merge of the green surface as the base surface and the blue and red surface.

The second surface shows the merge of the blue surface as the base surface and the green surface.

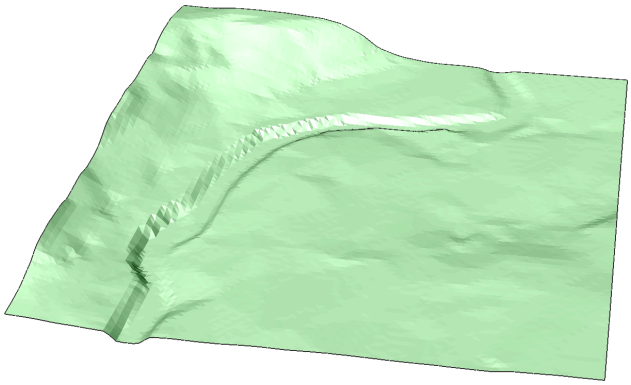
The third surface shows the merge of the red surface as the base surface and the green surface.

9.9.11 About TIN Modify

A TIN Surface can be modified by deforming or smoothening the surface. By deforming the surface, the TIN Surface is adjusted following the contours of an area defined by the user. By smoothening the surface, a selected area is leveled which is done by interpolation of this predefined area.

9.9.12 Procedure: Adding deformations on a TIN Surface using Radius

1 Select a TIN Surface and type TINMODIFY in the Command line.



2 Select the **Deform** option.

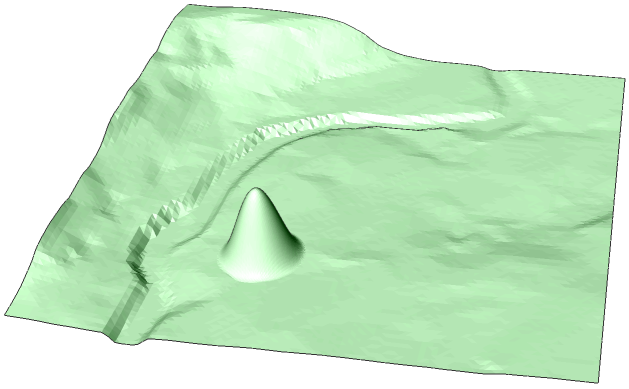
3 Choose a deformation radius.

4 Click on the area to apply the deformation.

5 Choose the elevation height.

6 Choose whether you want to add the adjustment as a new element or as a modification of the existing

TIN Surface.



Nota: By adding the modification as a new element, the structure of the original TIN Surface isn't modified but added as a new TIN surface.

7 Click on other areas to add new deformation or press ENTER to end the command.

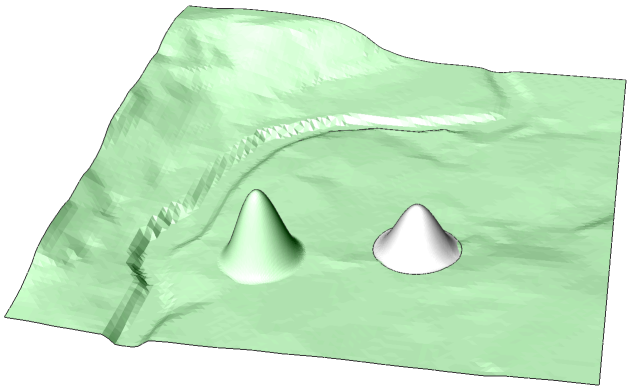
9.9.13 Procedure: Adding deformations on a TIN Surface using specify Contour

1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select the **specify Contour** option.

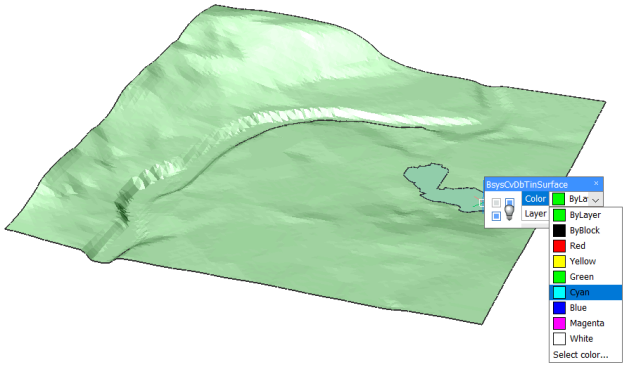
4 Pick a contour on the TIN Surface.



5 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for the adjustment.

6 Click on other contours to add new deformation or press ENTER to end the command.

7 You have now created a new element on your TIN Surface. You can change its properties as you like. For example: change the color of the element.



9.9.14 Procedure: Adding deformations on a TIN Surface using select Entity

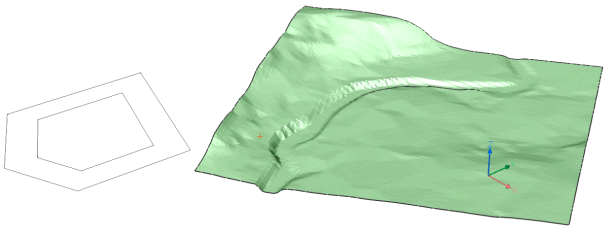
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

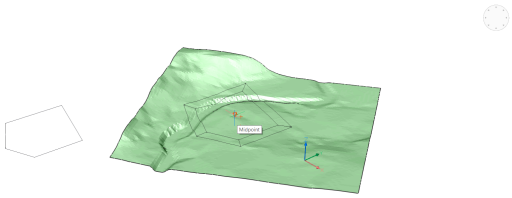
3 Type E and hit ENTER to choose **Select Entity**.

4 Select a closed entity (e.g. Polyline).

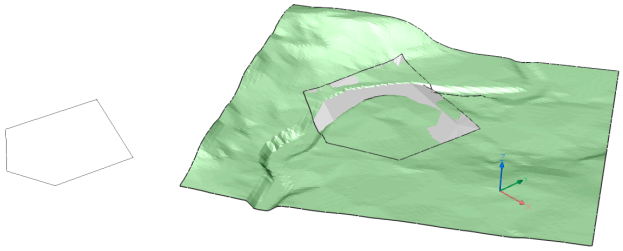
5 Choose an offset distance.



6 Click on the area to apply the deformation.



7 Choose the elevation height.



8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a New element for your adjustment.

9 Click on other areas to add new deformation or press ENTER to end the command. You have now added a deformation using **select Entity**.

9.9.15 Procedure: Adding deformations on a TIN Surface using draw Polygon

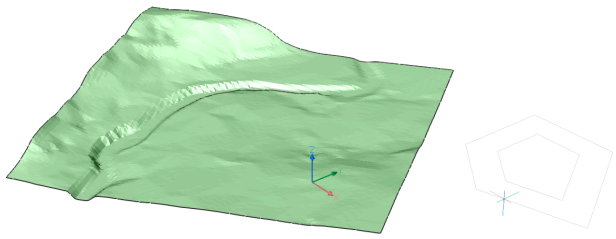
1 Select a TIN Surface and type TINMODIFY in the Command line.

2 Select the **Deform** option.

3 Select **draw Polygon**.

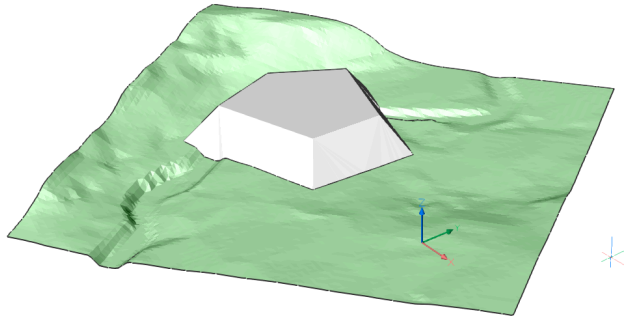
4 Draw a polygon, hit ENTER to close the polygon.

5 Choose an offset distance.



6 Select the area where to apply the deformation.

7 Choose the elevation height.

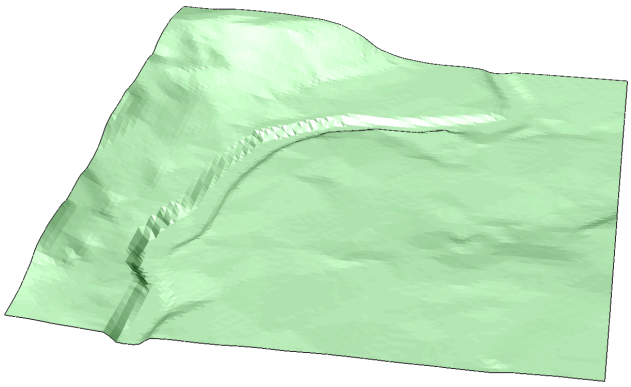


8 Choose whether you want to add the adjustment as a new element or as a modification of the existing TIN Surface. Type **N** to create a new element for your adjustment.

9 Click on other areas to add new deformation or press Enter to end the command. You have now added a deformation using draw polygon.

9.9.16 Procedure: Smoothening a TIN Surface

1 Select a TIN Surface and type TINMODIFY in the Command line.

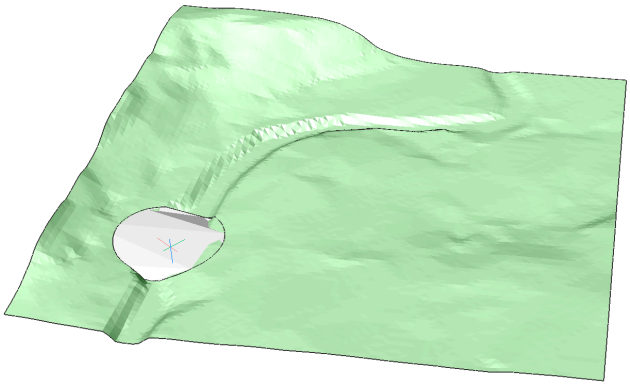


2 Select the **Smoothen** option.

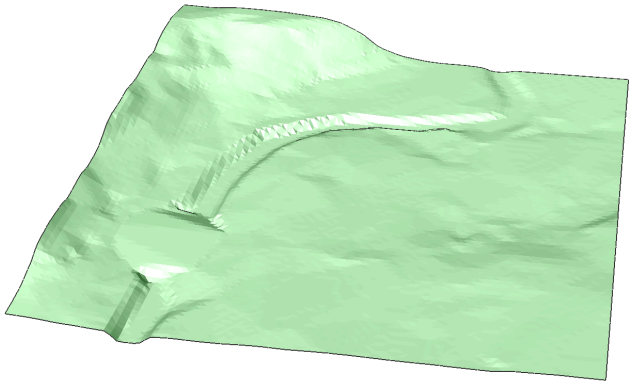
3 Choose a radius.

Nota: The procedure for **select Entity** or **draw Polygon** is analogue to **Adding deformations using select Entity** and **Adding deformations using draw Polygon** starting from step 3.

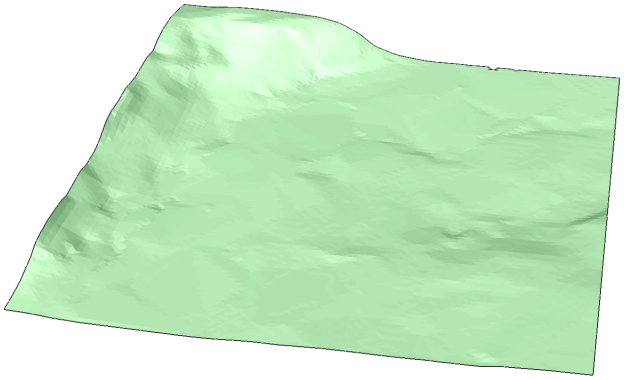
4 A preview of the intervention is shown.



5 Click to execute the command.



6 Click on other areas to smoothen or press ENTER to end the command.



9.10 Analyzing TIN Surfaces

9.10.1 Commands

TINVOLUME, TINWATERDROP

9.10.2 Overview

In BricsCAD, topographical surfaces created as a TIN Surface, can be analyzed using different tools.

9.10.3 About TIN Volume

A TIN volume surface provides an exact difference between the base and comparison surfaces. Therefore, the Z-value of any point in the volume surface is precisely the difference between the Z of the comparison surface at that point and the base surface at that point. A TIN Volume Surface is drawn as a custom entity with additional fill/cut volume properties.

You can use TIN Volume to calculate or analyze earthworks:

- Calculate fill/cut volumes inside bounding area.
- Draw zero contours which divide fill and cut areas.
- Visualize the mass distribution.

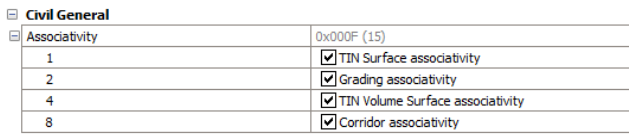
9.10.4 Procedure: Creating a TIN Volume

In the following procedures, a TIN Volume is created. Choose whether the volume is associative or not. If associativity is turned on: TIN Volume Surface is automatically updated if base or comparison surfaces are changed. To disable or enable the associativity, follow these steps:

1 Type **Settings** in the Command line and press Enter.

2 Go to **Civil** > **Civil General** > **Associativity**.

3 Check or uncheck **TIN Volume Surface Associativity**.

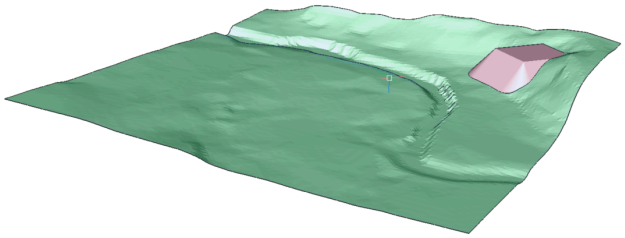


Civil General	
Associativity	0x000F (15)
1	☑ TIN Surface associativity
2	☑ Grading associativity
4	☑ TIN Volume Surface associativity
8	☑ Corridor associativity

9.10.5 Creating a TIN Volume using a comparison TIN Surface

1 Launch the TINVOLUME command.

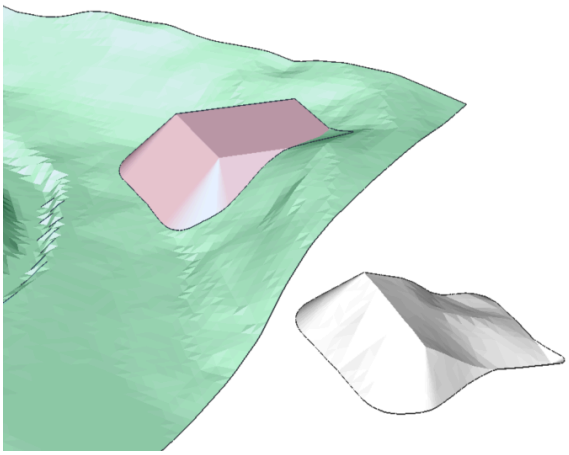
2 Choose a base surface. In this example, we choose the green TIN surface as the base surface.



3 Choose a comparison surface. In this example, we choose the red grading as the comparison surface.

4 Select a bounding area as the area to apply the operation orpress ENTER. The outer boundary of the surface will be used as a boundary.

5 The TIN Volume appears under the TIN Surface.



9.10.6 Creating a TIN Volume using Elevation

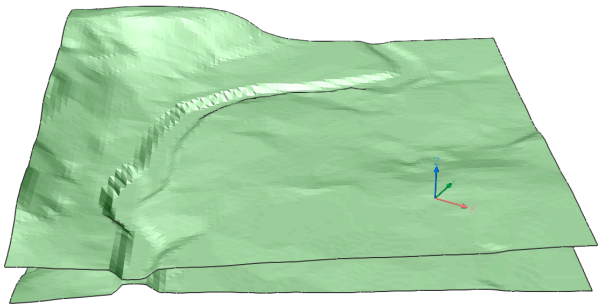
1 Launch the TINVOLUME command.

2 Choose a base surface.

3 Select **Elevation** in the Command line.

4 Choose the elevation height.

5 Select a bounding area as the area to apply the operation or press ENTER. The outer boundary of the surface will be used as a boundary.



9.10.7 About TIN Water Drop

The TINWATERDROP command creates water drop paths in real-time by moving the cursor over a TIN Surface. Water path is created as a 3D polyline in the current position if you click the mouse button.

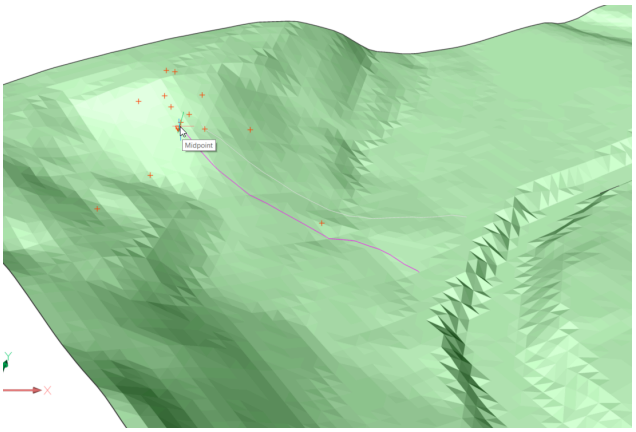
9.10.8 Procedure: Creating a water drop path

1 Launch the TINWATERDROP command.

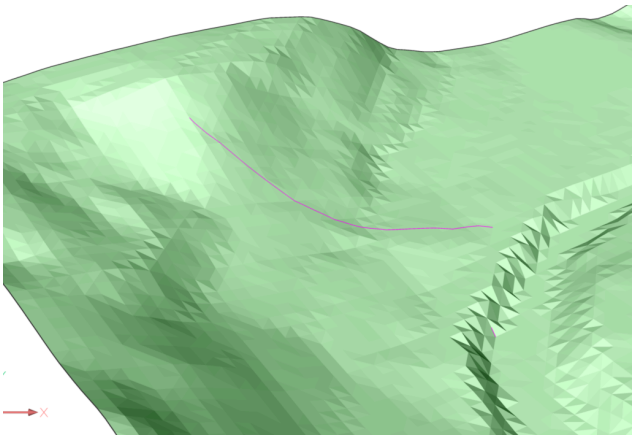
2 Select the TIN surface.

3 Move the cursor over the surface, a line representing the drop path is shown in real-time.

Nota: It would be better if the water drop path is colored so it could be seen clearer. To do so, you should change the current layer to another color, e.g. red.



4 Left click with the mouse to create a 3D polyline of the water drop path.



9.11 Working with Gradings

9.11.1 Commands

GRADING, GRADINGBALANCE

9.11.2 Overview

In BricsCAD, graded surfaces are created using the Create Grading tool.

Grading is a set of processes to maintain a base level for construction works or TIN surface design by defining a slope or adjusting the offset of a grading. For instance, creating parking areas, recreation reserves or road and railway designs.

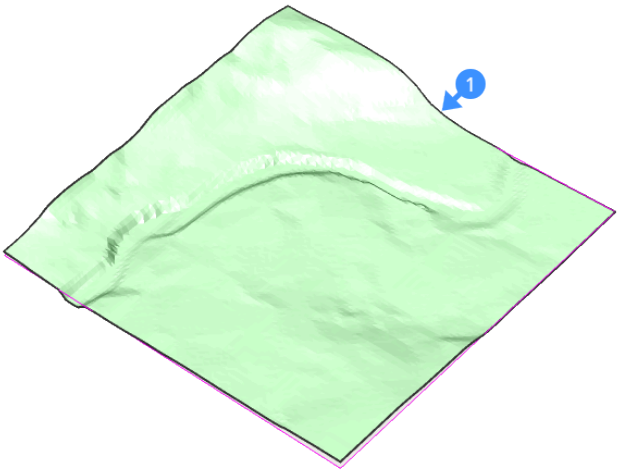
Two projection methods **Slope to surface** and **Offset to slope** are supported:

- **Slope to surface** needs an input entity (e.g., polyline, line, arc, circle), a surface and slope to grade the surface.
- **Offset to slope** needs an input entity, offset and slope.

Graded surfaces are not editable with the TINEDIT command.

9.11.3 Procedure: Grading a surface to create a pond

1 Open a drawing that has a topographical surface (1).

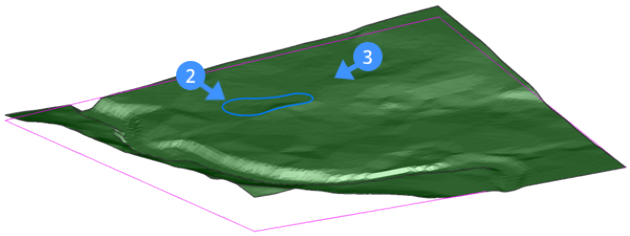


2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your pond.

Nota: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

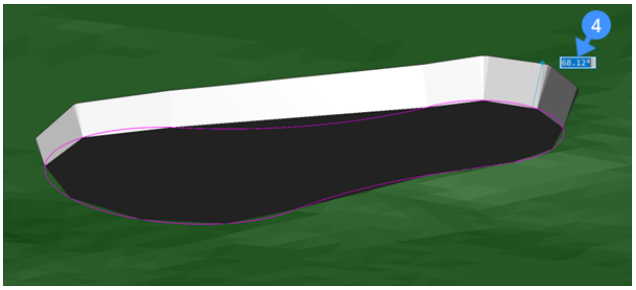
4 Select the target surface.



5 Define the slope of the grading.

Nota: Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Nota: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (4).



6 Press ENTER to accept the slope of the grading.

9.11.4 Displaying the properties of a graded surface

The properties are displayed in the Properties Panel after selecting the grading.

Data	
Grading type	Slope to surface
Target surface	BB
Name	Grading
Description	
Grading settings	
Cut slope	45
Fill slope	45
Side	Right
Infill	On
Boundary	Off
Grading statistics	
Fill volume	99.46
Cut volume	0.00
Net volume	99.46
TIN Visibility	
Boundary line	Off
Points	Off
Triangles	On
Contours	Off
TIN Contours	
Major contours interval	10
Minor contours interval	1
Major contours color	Red
Minor contours color	Green
TIN Statistics	
Number of points	389
Number of triangles	391
Minimum elevation	348.61
Maximum elevation	350
2D area	182.79
3D area	195.51

Grading type

Displays which projection method is used to grade the surface.

Target surface

Displays the handle number of the target surface

Cut slope

Displays the cut slope.

Fill Slope

Displays the cut slope.

Side

Controls on which side of the input entity the grading is drawn. Left or Right.

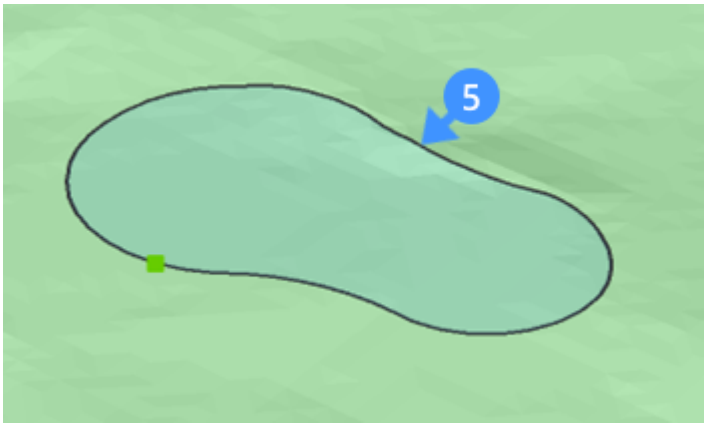
Infill

The internal are of entities is filled if this option is turned on. Valid for closed entities only (circle, closed polyline).

Boundary

Controls the boundary of the grading. When the grading boundary is on, that boundary is added as a **hide** boundary to the target surface.

The following images show when the grading boundary of the pond is off (5) and on (6). Based on the grading boundary, the net volume for the body of the pond is calculated and can be found in the properties panel (7).



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

7

Fill volume
Displays the fill volume for a body/shape limited with the target surface and graded surface.

Cut volume
Displays the cut volume for a body/shape limited with the target surface and graded surface.

Net volume
Displays the net volume for a body/shape

9.11.5 Procedure: Grading a surface to create a platform

1 Open a drawing that has a topographical surface (8).

2 Launch the GRADING command.

3 Select the entity that defines the footprint area of your platform (9).

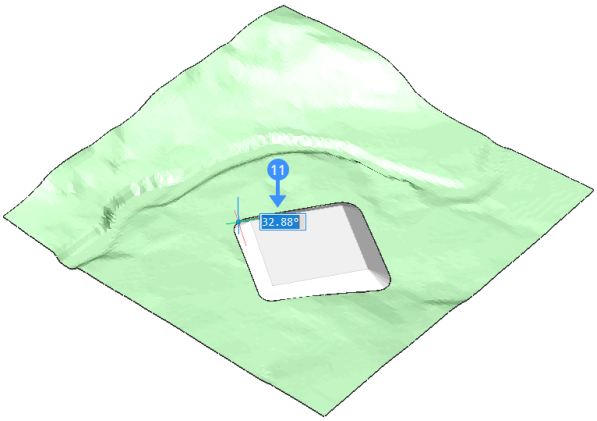
Nota: Before selecting the target surface you can change to **slOpe/offset** projection method which requires an input entity, offset and slope.

4 Select the target surface (10) to create a graded area.

5 Define the slope of the grading.

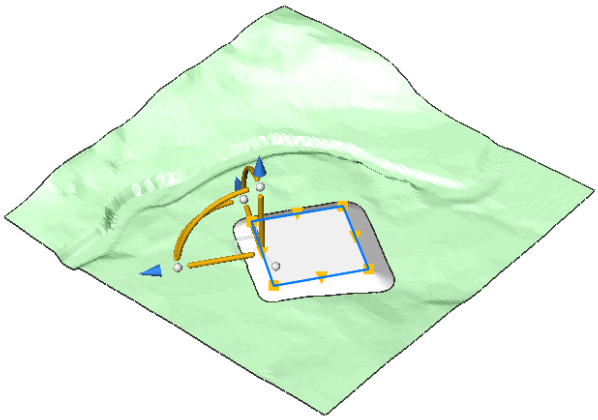
Nota: Grading is attached to the input entity, and it is connected with the target surface. Moving the mouse in/out dynamically increases/decreases the slope of the grading.

Nota: If DYNMODE variable is set to 3, you can also input the slope angle directly from keyboard (11).

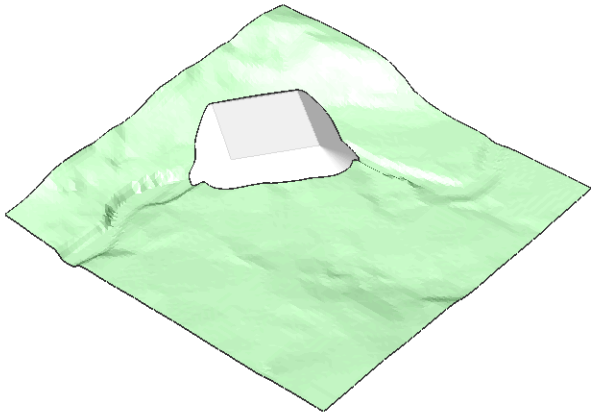


6 Press ENTER to accept the slope of the grading.

Nota: You can easily move the platform that you created. Do this by using the manipulator on the entity from which you created the platform. You can move both horizontally and vertically.



The grading will automatically be adapted to its surroundings.

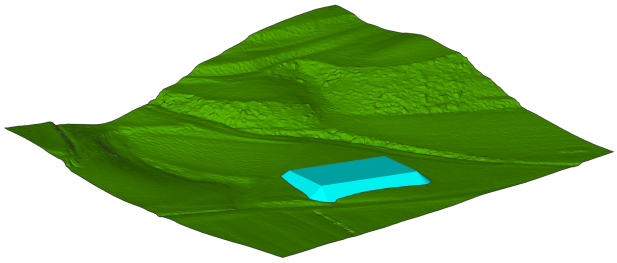


9.11.6 About Balance Grading Volumes

The GRADINGBALANCE command allows you to balance cut and fill volumes within the specified tolerance. This is achieved by raising/lowering the elevation of the grading input entity and the result is grading that has a net volume around zero, depending on the set tolerance.

9.11.7 Procedure: Adjusting the elevation of a grading to balance cut and fill volumes

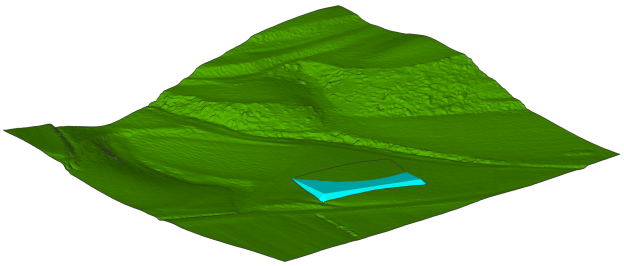
1 Open a drawing that contains a TIN surface and a grading representing a building pad.



2 Launch the GRADINGBALANCE command.

3 Select the grading.

4 Enter the tolerance in percent (1-100). 1 means that the difference between the cut and fill volume will be less than one percent and 100 means that the difference between the cut and fill volume will be less than 100 percent.



In the Properties panel you see that the cut and fill volumes are now balanced.

Grading statistics	
Fill volume	670.77
Cut volume	673.19
Net volume	-2.42

Nota: Balance Grading Volumes command works only on gradings that has Boundary set to **OFF**.

Grading settings	
Cut slope	50
Fill slope	35
Side	Left
Infill	On
Boundary	Off

9.12 Create and edit alignments

9.12.1 Commands

ALIGNMENT, ALIGNMENTEDIT, ALIGNMENTVIEW

9.12.2 Overview

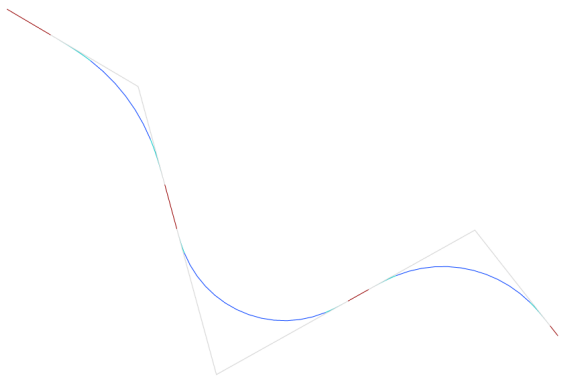
In BricsCAD, alignments are created using the Alignment tool. Alignment is the route of the linear objects, defined as a combination of lines and curves that are viewed as one object.

9.12.3 About ALIGNMENT

Alignment is a reference system used to position linear objects, like roads, railways, pipe networks, streets, retaining walls and bridges, in space. An alignment is represented with horizontal, vertical 2D curves, resulting in 3D alignment.

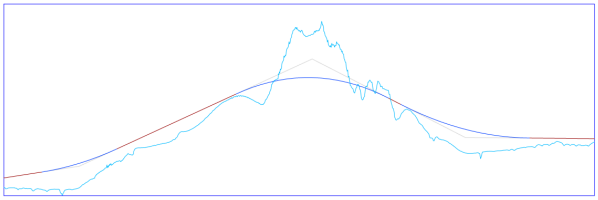
Horizontal alignment

Horizontal alignment is a 2D curve that is needed to precisely position linear objects in a horizontal direction. It becomes defined by the Points of Intersection (PI). Horizontal alignment is the horizontal trace on your TIN Surface which is used to generate the height profile and vertical alignment. A single horizontal alignment can have multiple vertical alignments. The horizontal alignment will appear under the surface.



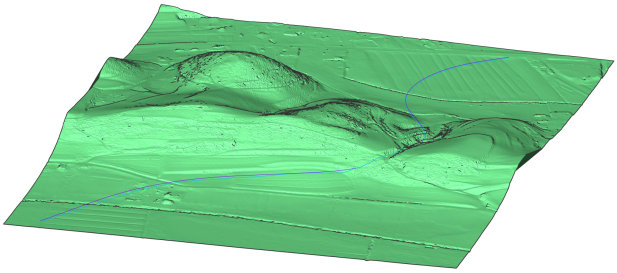
Vertical alignment

A vertical alignment is needed to precisely define a linear object in the vertical sense. The horizontal path defined by the horizontal alignment corresponds with a vertical profile of the TIN Surface. By defining Points of Vertical Intersection (PVI) on these vertical profiles of the target surfaces, the new 2D curve defines the vertical alignment.



3D alignment

A 3D Alignment is a 3D polygonal curve defined by the horizontal and vertical alignment.



9.12.4 Procedure: Creating an Alignment

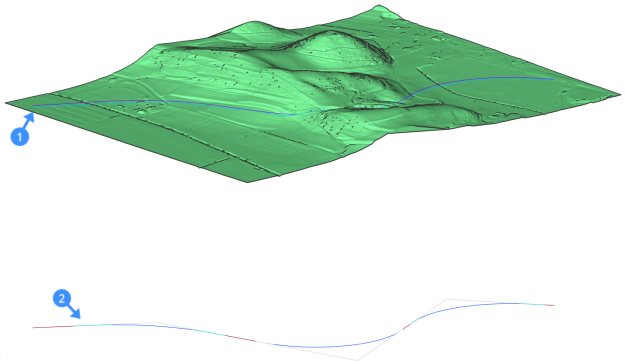
1 Open the drawing file that contains a TIN Surface.

2 Launch the ALIGNMENT command in the Command line.

3 Select **Horizontal** alignment in the Command line.

4 Select the TIN surface.

5 Pick the horizontal alignment PI points on your surface. Press ENTER when done. Your 3D alignments (1) and horizontal alignment (2) are now generated.



Nota: Three alignments are shown. The horizontal alignment on plane 0,0, light blue 3D alignment shows the projection of horizontal alignment on TIN Surface and the purple 3D alignment shows the design alignment that is automatically generated.

9.12.5 Procedure: Creating a Horizontal Alignment without 3D alignments

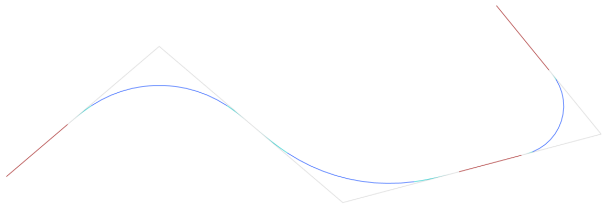
1 Open an empty drawing and launch the ALIGNMENT command in the Command line.

2 Select **Horizontal** alignment in the Command line.

3 Select the **Pick Pi point** option in the Command line.

4 Pick the horizontal alignment PI points. Press ENTER when done.

Your horizontal alignment is now generated.

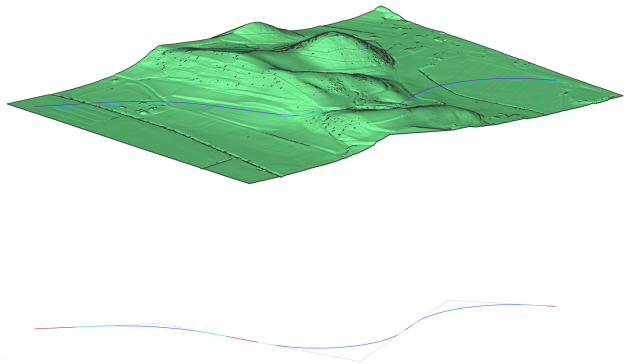


9.12.6 About Vertical Alignment View

The horizontal alignment creates a vertical section of the target surface. This 2D curve can be displayed in a separate viewport inside your model space using the command ALIGNMENTVIEW.

9.12.7 Procedure: Creating a Vertical Alignment View

1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and 3D Alignments.

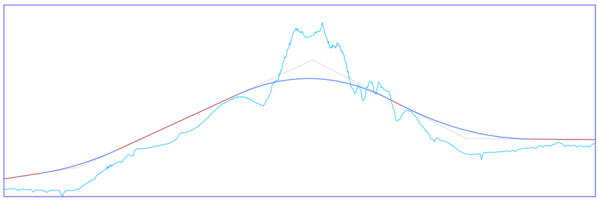


2 Launch the ALIGNMENTVIEW command in the Command line.

3 Select a horizontal or 3d alignment in the drawing.

4 Pick a point as origin for the vertical alignment view.

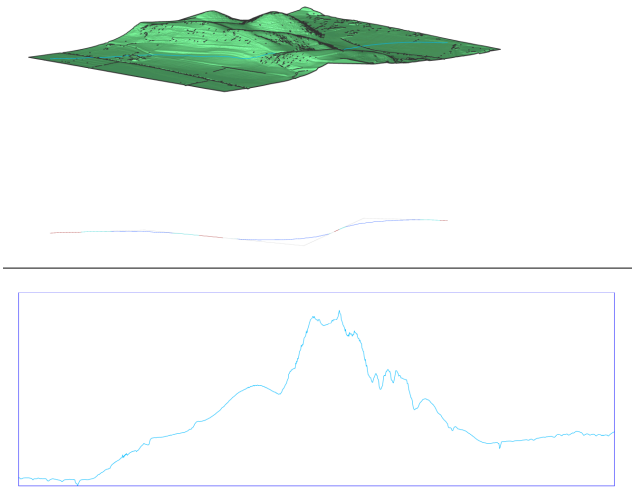
Your vertical alignment view is now generated.



Nota: Two vertical alignments are shown. Light blue represents terrain line from the TIN Surface, and another is design vertical alignment. The design vertical alignment has straight gradients and vertical curves and is created automatically based on the accuracy factor. You can change the accuracy factor in the Properties Panel from 1 to 100. When you choose 1 the accuracy will be the lowest.

9.12.8 Procedure: Creating a Vertical Alignment

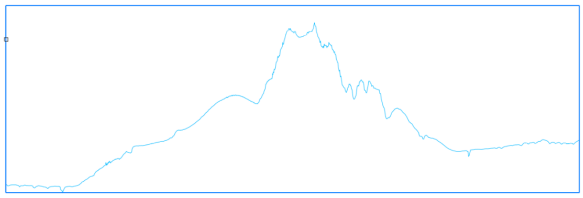
1 Open the drawing file that contains a TIN Surface, Horizontal Alignment and Vertical Alignment View.



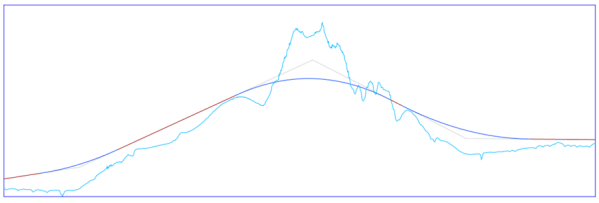
2 Launch the ALIGNMENT command in the Command line.

3 Select the **Vertical** option in the Command line.

4 Select vertical alignment view in the drawing.

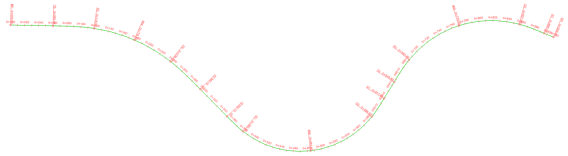


5 Pick the vertical alignment PVI points. Press ENTER when done. If you want to go one step back, use Undo. Automatic 3d alignment option generates design vertical alignment automatically. Your design vertical alignment is now generated.



9.12.9 Procedure: Creating an Alignment from Civil 3D drawing

1 Open the drawing file from Civil 3D that contains a Civil 3D Alignment.

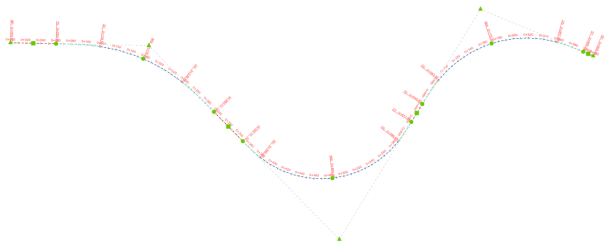


2 Launch the ALIGNMENT command in the Command line.

3 Select the **create from Civil 3d alignment**.

4 Select the civil alignment in the drawing and press ENTER.

Civil 3D alignment is converted to BricsCAD Alignment.



9.12.10 About ALIGNMENTEDIT

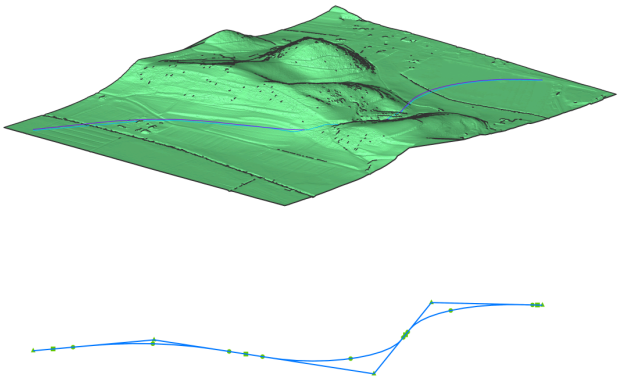
After creating an alignment, you can continue a horizontal or vertical alignment, add or delete PV/PVI points or change the TIN Surface using the command ALIGNMENTEDIT. The 3D alignment will be updated automatically.

9.12.11 Procedure: Continuing a horizontal alignment with ALIGNMENTEDIT

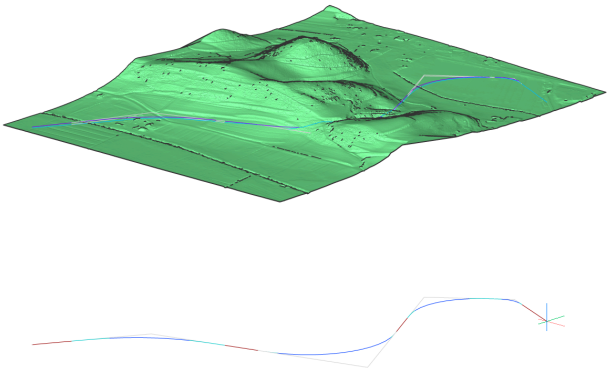
1 Draw a horizontal alignment following the steps from the Procedure: Creating an alignment.

2 Type ALIGNMENTEDIT in the Command line.

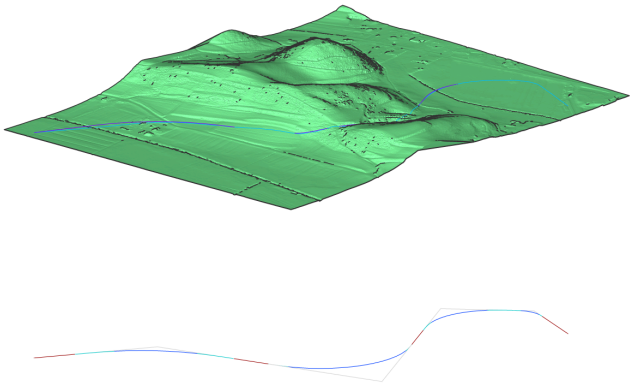
3 Select the horizontal alignment you want to edit.



4 Select points on your surface to continue your existing alignment.

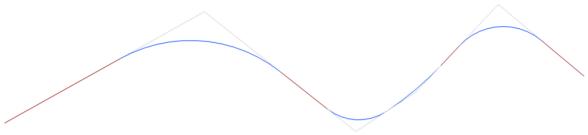


5 Press enter when finished. You have now continued your horizontal alignment. The 3D alignment will be updated automatically.



9.12.12 Procedure: Adding PI to an existing horizontal alignment

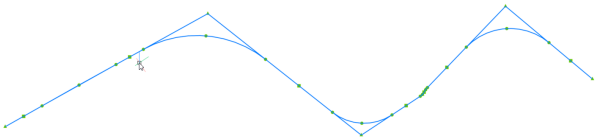
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit. You are prompted: Pick a point to continue alignment or [Add Pi/Remove Pi/change Tin surface].

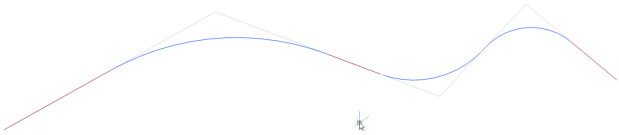
3 Select **Add Pi** in the Command line.

4 Pick point to add new PI.



9.12.13 Procedure: Removing PI to an existing horizontal alignment

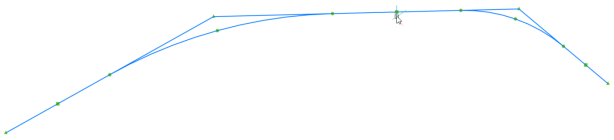
1 Type ALIGNMENTEDIT in the Command line.



2 Select the horizontal alignment you want to edit.

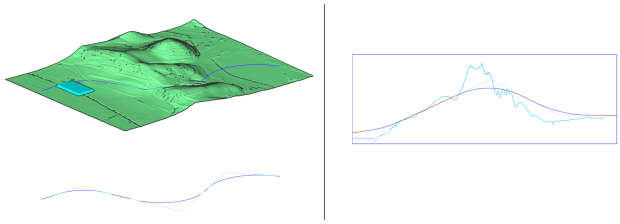
3 Select the **Remove Pi** option in the Command line.

4 Pick a PI point to remove.



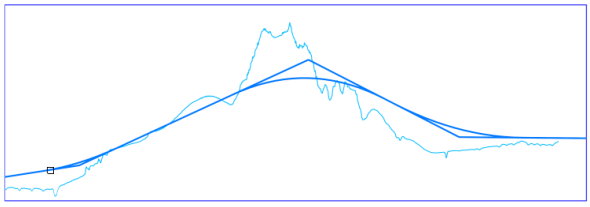
9.12.14 Procedure: Adding additional TIN Surface to Vertical alignment view

1 Open the drawing file that contains a TIN Surface, Alignments, Vertical Alignment View and grading.

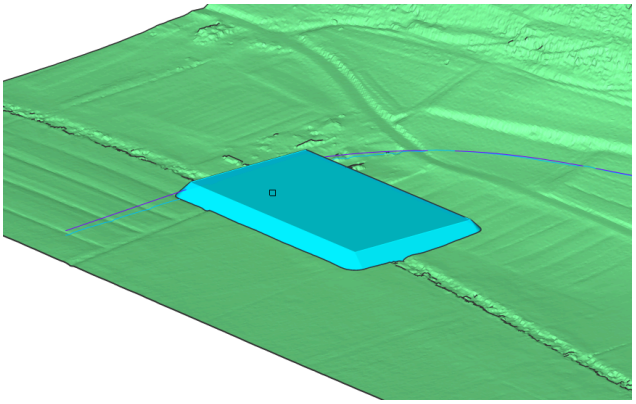


2 Launch the ALIGNMENTEDIT command in the Command line.

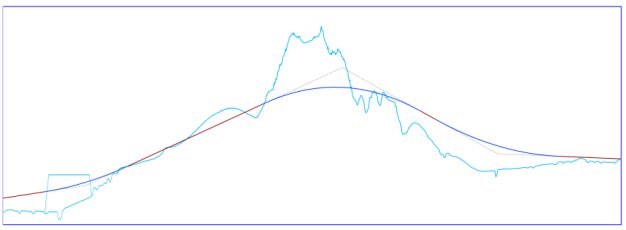
3 Select the **change Tin surface** option in the Command line.



4 Select the TIN surface to add.



In the vertical alignment view, you can see an additional light blue line representing the grading object.



9.13 Modeling a Corridor

9.13.1 Commands

CORRIDOR, CORRIDORTEMPLATE, CORRIDORTEMPLATEELEMENT, CORRIDOREDIT, CORRIDOREXTRACT.

9.13.2 Overview

In BricsCAD, Corridors that follow a certain 3D alignment are created using the Corridor tool.

9.13.3 About Corridor

A Corridor is a BricsCAD Civil drawing object used for modeling three-dimensional linear objects such as roads, railway, retaining walls, and bridges. It principally consists of a 3D alignment of the linear object and its typical cross-section – which is called Corridor Template in BricsCAD.

The CORRIDOR command enables you to create a Corridor, based on a Corridor Template, that follows a 3D alignment on a certain topographical surface.

9.13.4 About Corridor Template

In BricsCAD, a Corridor Template entity represents the framework to append a collection of template element objects. The command CORRIDORTEMPLATE enables you to create a corridor template entity and append one or more template elements to it.

9.13.5 About Corridor Template Element

In BricsCAD, a Corridor Template Element entity defines the geometry of a component used in a corridor section. The CORRIDORTEMPLATEELEMENT command enables you to create a corridor template element from closed polylines.

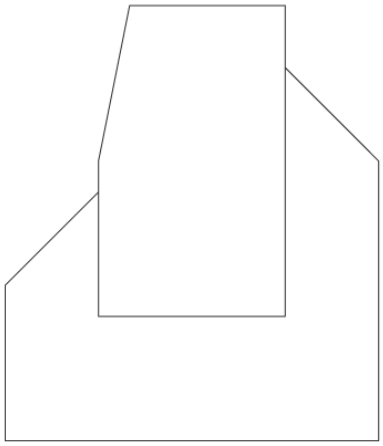
9.13.6 About Corridor Extract

The CORRIDOREXTRACT command is used to generate meshes, solids, polylines or outer boundaries from an existing Corridor.

9.13.7 Procedure: Creating a template element of kerb

1 Open an empty drawing.

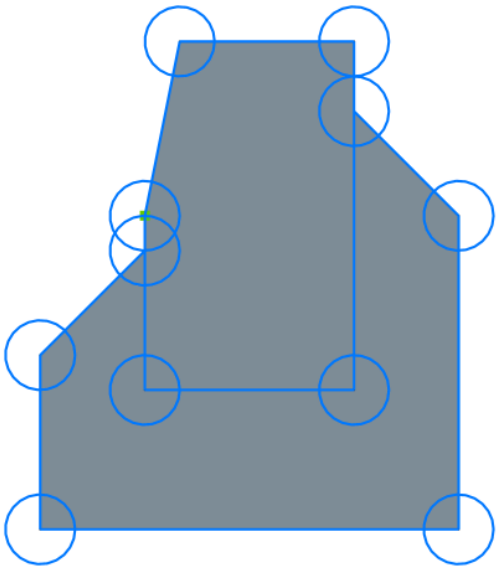
2 Draw closed polylines representing road kerb.



3 Launch the CORRIDORTEMPLATEELEMENT command in the Command line.

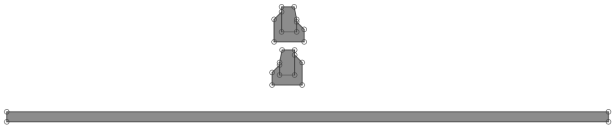
4 Select polylines representing kerb.

5 Pick one of the polyline vertices, the base point position can also be changed afterward.



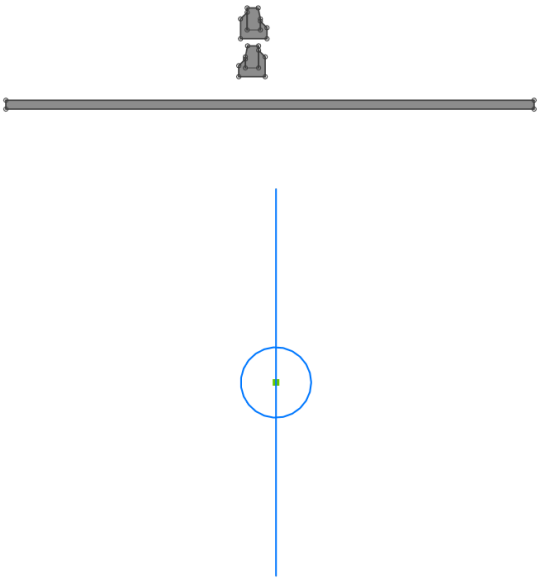
9.13.8 Procedure: Creating a template of road cross-section with asphalt layer and left and right kerb

1 Open the drawing file that contains a collection of template elements.



2 Launch the CORRIDORTEMPLATE command in the Command line.

3 Pick a point in the drawing where you want to create a corridor template.

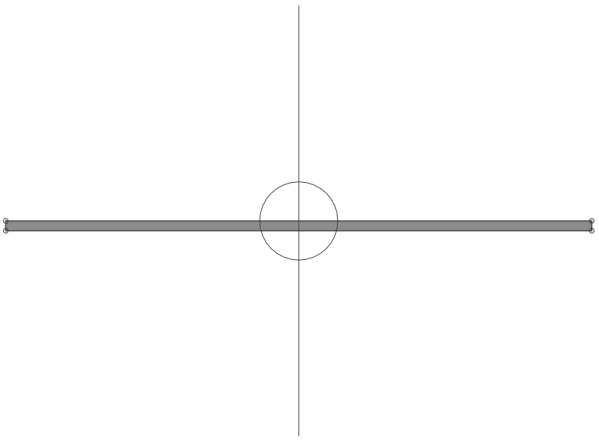


4 Launch the CORRIDORTEMPLATE command in the Command line.

5 Select the **Add template element** option in the Command line.

6 Select the template element representing the asphalt layer.

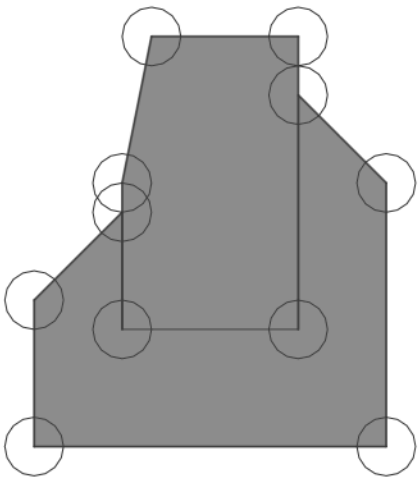
7 Select corridor template or template element point: (select the template)



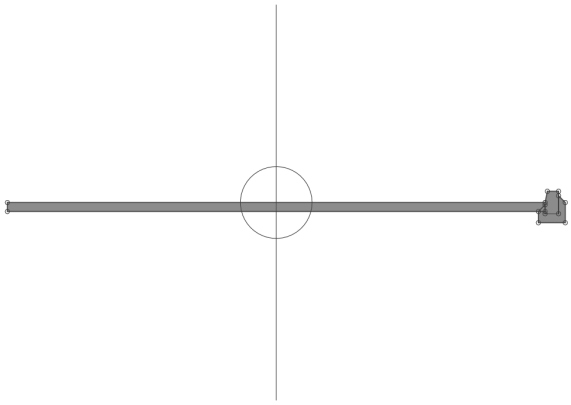
8 Launch the CORRIDORTEMPLATE command in the Command line.

9 Select the **Add template element** option in the Command line.

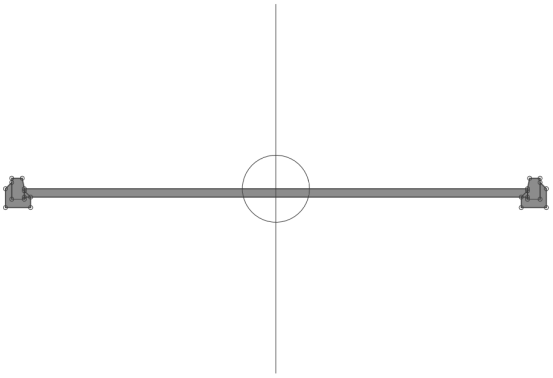
10 Select right top template element point of asphalt layer.



11 Select template element representing right kerb.

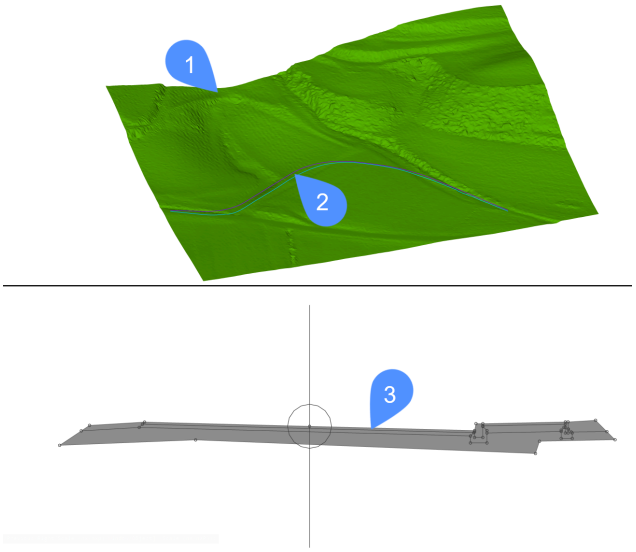


12 To add a kerb to the left side, repeat the steps described in points 9, 10, 11. By selecting the left kerb and left top point of asphalt layer this time.



9.13.9 Procedure: Creating a road corridor

1 Open the drawing file that contains the TIN Surface (1), 3D Alignment (2) and Corridor Template (3).



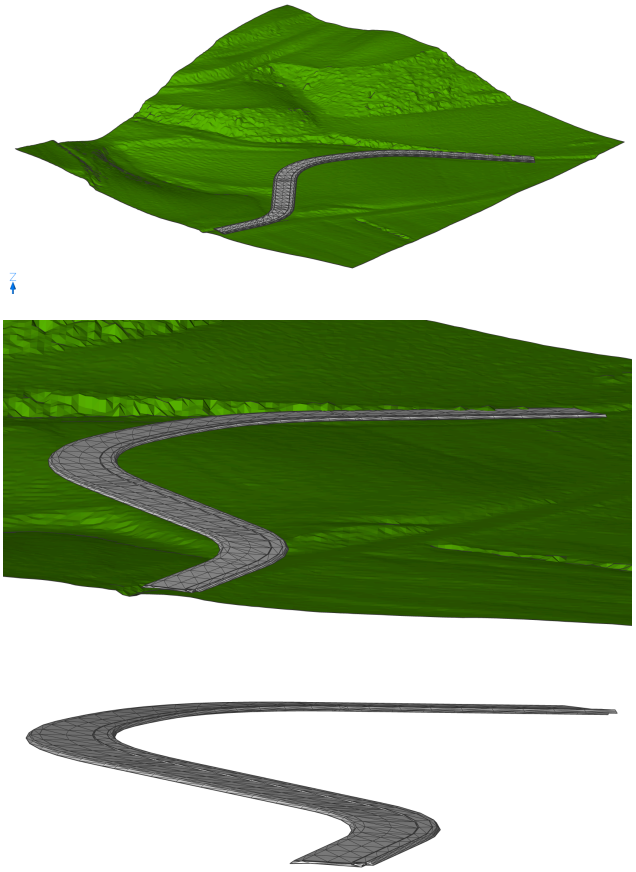
2 Launch the CORRIDOR command in the Command line.

3 Select 3D alignment (2) used as a corridor baseline.

4 Select corridor template (3).

5 Enter region start station. Press ENTER to start from beginning or type any station where you want your corridor to start.

6 Enter region end station. Press ENTER to create corridor to the end of selected alignment or type any station where you want your corridor to end.



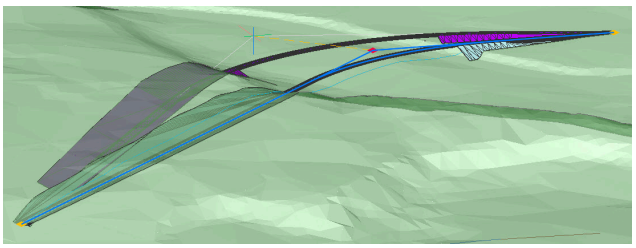
9.13.10 About Corridor Edit

The CORRIDOREDIT command adds or removes regions from an existing Corridor.

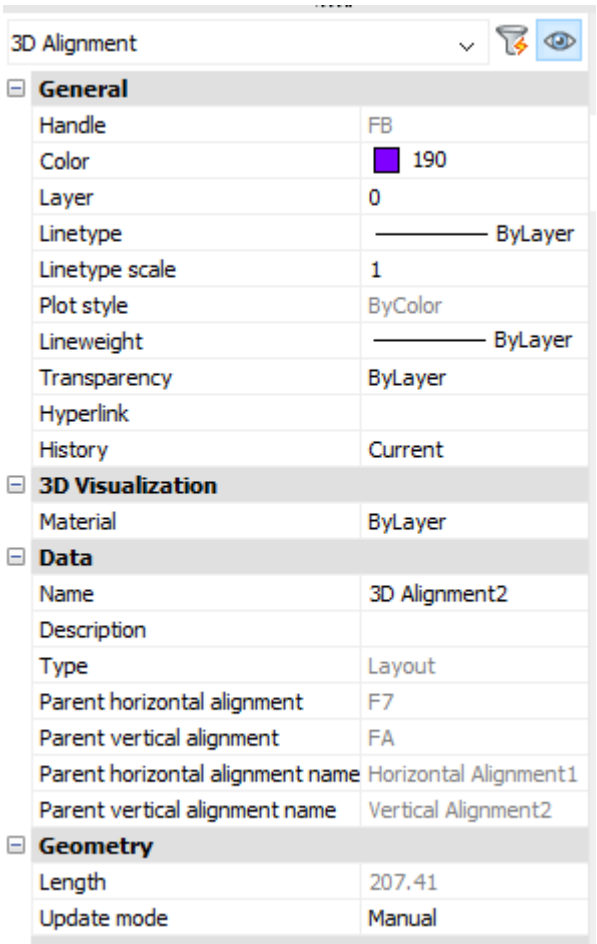
9.13.11 Procedure: Editing a corridor

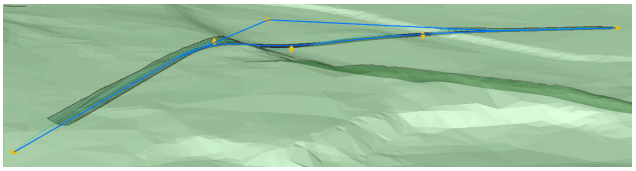
1 Open the drawing file that contains the corridor you would like to edit.

2 Select the 3D alignment that the corridor has as a reference and move the grip points to edit the corridor horizontally.

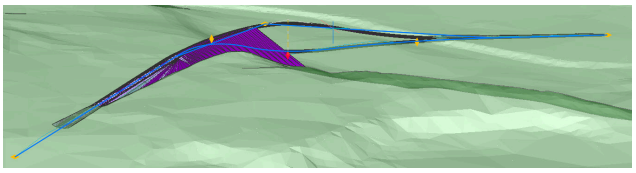


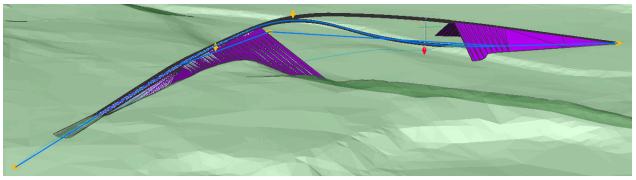
3 To edit the corridor vertically, select the 3D alignment that the corridor has as a reference and open the Properties Panel. Set the **Update mode** to Manual. Extra grip points will appear.



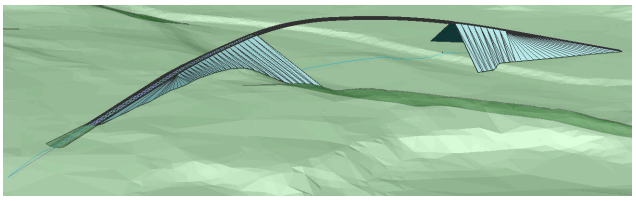


4 Move these grip points to edit the corridor vertically.





5 Press ENTER and you have finished your edited corridor.



9.13.12 Displaying the properties of a Corridor

1 Select the Corridor.

2 The properties of the Corridor are displayed in the Properties panel.

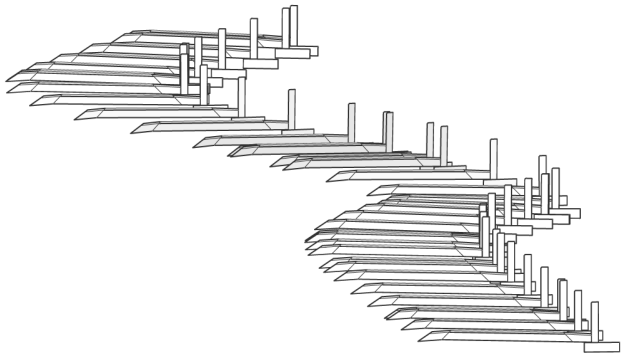
Corridor	
General	
Handle	147
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Data	
Name	Corridor
Description	
Baseline handle	FB
Regions	1
Template handle	144
Start region	100
End region	300
Horizontal tangent increment	5
Horizontal curve method	By increment
Horizontal curve increment	5
Horizontal curve mid ordinate	0.1
Horizontal spiral increment	5
Horizontal keypoints	On
Vertical curve increment	5
Vertical keypoints	On
Vertical local extremes	Off
Visual Style	
Applied template points	Off
Applied template links	Off
Applied template shapes	Off
Meshes	On
Stringlines	Off

Applied template points

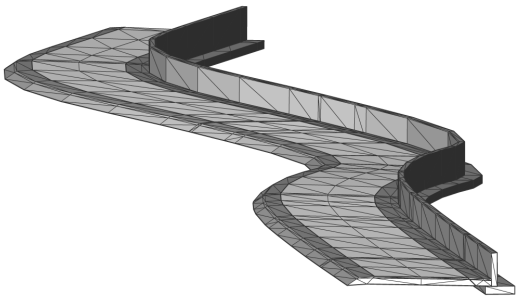
Represents the corridor with template element points.

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. Modellazione parametrica

10.1 Parametric modeling overview

BricsCAD provides a rich toolset to control 2D and 3D geometric models with parameters of different kinds. This includes:

- **Global parameters**: parameters of 2D/3D dimensional constraints, user-defined parameters, configuration parameters of design tables, etc.
- **Local parameters**: parameters of arrays and constrained entities inside arrays, parameters of inserts of parametric components, etc.

All parameters and their current values can be found in different sections in the **Mechanical Browser** panel. Parameter values can be linked together via expressions. When you change the value of a parameter, the model is updated automatically. This workflow allows you to add parametric behavior to any geometry and explore design variants easily.

Nota: In BricsCAD there are no parent-child dependencies between geometric elements: for example, if you change a sketch used to create an extruded 3d solid, the solid is not changed accordingly. However, you can create dependencies with parameters and expressions afterward.

10.1.1 Global and local parameters

Commands:

- -PARAMETERS command manages dimensional constraint parameters through the Command line.
- MECHANICALBROWSEROPEN command opens the **Mechanical Browser** panel.
- CLEANUNUSEDVARIABLES command purges parametric variables not used by constraint expressions and not linked to dimensions.

Parameters in BricsCAD can be either global or local.

Local parameters are attached to a particular entity, an associative array or a mechanical component.

Global parameters are not attached to a particular entity, they are subdivided into four groups:

- Parameters of 2D dimensional constraints
- Parameters of 3D dimensional constraints
- User-defined parameters
- Configuration parameters of design tables

10.1.2 Defining an expression for a parameter

You can define an expression for both parameter types, global and local. The simplest expression is a number or the name of a global parameter (local parameters cannot be referenced in expressions). A more complex expression can include standard operators and functions.

To define an expression for a parameter: in the **Mechanical Browser** panel, click on the parameter name and fill out the **Expression** field.

Nota: You can also define an expression for a parameter in the Command line using the -PARAMETERS command.

10.1.3 Defining dependent parameters

In expressions, you can use both: names of global parameters and numbers. This is known as a 'non-constant expression'. If a non-constant expression is set to a parameter, its icon changes from an open padlock ( ) to a closed padlock ( ) in the **Mechanical Browser** panel.

This means that the parameter becomes dependent on (an)other (defining) parameter(s) and its value will be automatically recalculated when the value(s) of the defining parameter(s) change(s).

You can create chains of parameters, where each element is dependent on the previous one. There is no limit to the length of such chain. However, be careful not to create a loop where the next element in a chain drives one of the previous elements. This might result in an infinite evaluation of expressions. BricsCAD automatically detects such problems and does not accept expressions that lead to a parametric loop.

10.1.4 Parameters of associative arrays

Associative arrays are created by the ARRAY, ARRAYRECT, ARRAYPOLAR and ARRATYPATH commands. The properties of associative arrays that can be controlled with an expression are the following.

- **Rectangular arrays**: Number of columns, Column spacing, Number of rows, Row spacing, Row elevation increment, Number of levels, Level spacing, and Axis angle.
- **Polar arrays**: Radius, Number of Items, Angle between items, Fill angle, Number of rows, Row spacing, Row elevation increment, Number of Levels, and Level spacing.
- **Path arrays**: Item spacing, Start offset, Rows, Row spacing, Row elevation increment, Levels and Level spacing.

To define an expression for any of these properties, select the corresponding array entity, either in the drawing area or in the **Mechanical Browser** panel and enter the expression in the corresponding properties field in the **Mechanical Browser** panel or in the **Properties** panel.

If an associative array contains entities constrained with 2D or 3D dimensional constraints, then parameters of these constraints can also be evaluated with expressions. These nested parameters are listed in the **Mechanical Browser** panel. If you select any of them, you can define an expression.

10.1.5 Parameters of mechanical components

The BMUPDATEMODE system variablecontrols whether external assembly components are always reloaded or only when modified.

If you insert a mechanical component or just a plain .dwg file into a drawing with BMINSERT command and the insert contains global exposed parameters, these parameters are converted to local parameters. They are associated with the corresponding component insert and are visible in the **Mechanical Browser** panel under the name of this insert. Click the parameter(s) to edit.

10.1.6 Parameter properties

All parameters such as; user-defined, 3D constraints, design tables, parameters of parametric components and arrays can be managed via **Mechanical Browser** panel.

- **Name**: Identifies the parameter. The parameter name can be used in expressions of other parameters in the same **component**. Name cannot begin with a number, it should be alphanumeric without spaces.
- **Expression**: Type a value or an expression.
- **Value**: Displays the current value of the parameter.
- **Geometry-driven**: If yes, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.
- **Description**: Optional description of the parameter.
- **Exposed**: Defines whether the parameter is visible and can be modified when the component is inserted in an assembly. Select the field, then click the down-arrow button and choose an option. A new parameter gets the **Auto** option by default.
    - **Auto** = the parameter is exposed only if it does not depend on other parameters.
    - **ON** = the parameter is always exposed.
    - **OFF** = the parameter is never exposed.
- **Units**: Sets the dimension of the parameter: **Linear**, **2-dimensional** or **3-dimensional**. Select the field, then click the down-arrow button and choose an option.

10.1.7 3D constraints properties

BricsCAD provides a set of properties for 3D constraints which allow you to control mutual placement of the constrained geometrical entities (or constraint arguments) upon 3D constraints recalculation.

The properties may be assigned both to a constraint itself (**Directions** property) and/or to the constraints arguments, for each argument separately (**Placement** and **Use as** properties).

- **Type**: Indicates the constraint type: Path, Distance, Radius, Angle or Cone Angle.
- **Enabled**: Controls whether the constraint is On or Off.
- **Directions**: Specifies whether the directions of the constraint arguments are **Same** or **Opposite**. Changing from one to another flips constrained entities with respect to each other.

  Direction is used for lines, planes, circles, cylinders, cones and torii. It is not used for points and spheres.

  The direction of a plane is its normal vector. The direction of a circle is the normal vector of the circle's plane. The direction of a cylinder, cone, or torus is the direction of its axis. The direction of a planar face of a 3D solid is additionally adjusted to be external with respect to the 3D solid object.

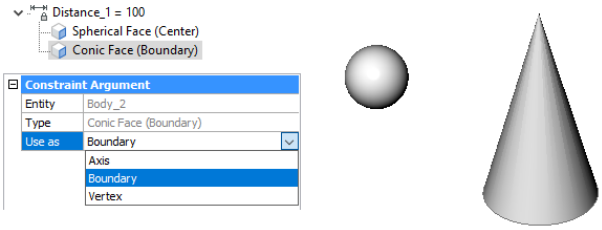
  Any option means that the directions may change upon direct modeling operations or upon a 3D constraint parameter change.

  The **Keep** option forces BricsCAD to keep the current directions. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Same** or **Opposite** (not both).

- **Placement**: Specifies how to place an argument with respect to another argument of the same constraint. For example, a point in a point-sphere distance may be placed either **Outside** or **Inside** of a sphere. Changing this property for a constraint argument moves the argument to the other side of another argument. The sides are defined for planes, spheres, cylinders, cones, and torii. And not defined for points, lines, and circles.

  Any means that both **Outside** and **Inside** placement is suitable for user's intentions.

  Keep option forces BricsCAD to keep the current placement of the argument. In other words, this option forces BricsCAD to analyze the mutual location of the arguments prior to 3D constraints solving and use either **Outside** or **Inside** (not both). The **Keep** option is used by default.
- **Use as**: This property is assigned to the arguments of the **Distance** and **Concentric** constraints and applies to circles, spheres, cylinders, cones and torii and defines which geometry is used in a 3D constraint to measure the distance or to coincide with the other entity.



  There are four options for the **Use as** property:

  - **Boundary**: the constraint is applied to the boundary of the argument;
  - **Axis**: applies to an argument that has an axis: circle, cylinder, cone, or torus;
  - **Center**: applies to an argument that has a central point: circle, sphere, or torus;
  - **Vertex**: applies to a cone vertex only.

10.1.8 Context menus

In the **Mechanical Browser** panel, the following context menus are available.

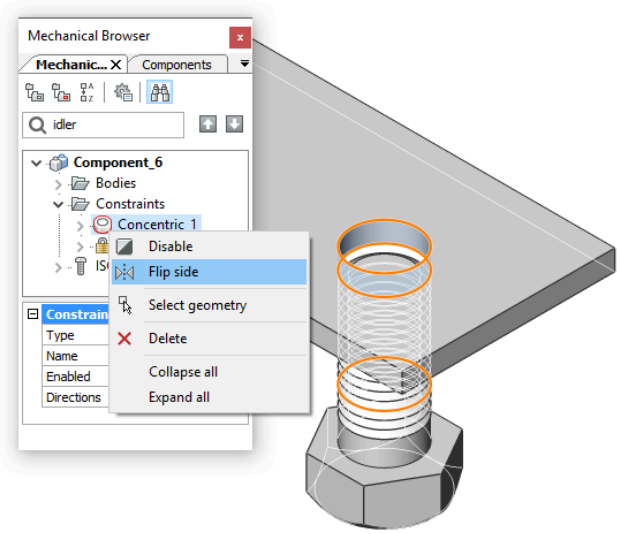
Main Node

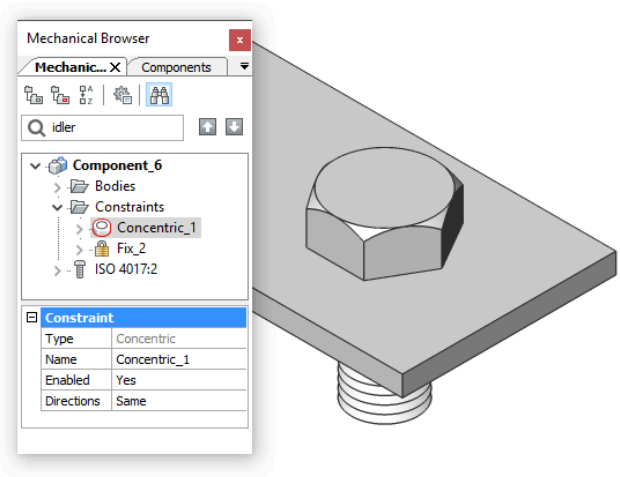
- **Add new parameter**: Creates a new parameter.
- **Collapse All**: Collapse all nodes.
- **Expand All**: Expand all nodes.

Sub-nodes

- **Delete**: deletes the selected parameter from file.
- **Create design table**: creates a design table to drive parametric block parameters.
- **Animate**: animates the parameter value within some range in the drawing.
- **Link to parameter**: links a subcomponent parameter to the main level parameter.
- **Enabled**: controls whether the constraint or parameter is evaluated or not.
- **Geometry driven**: if selected, disables the **Expression** property. The value of a parameter value is obtained from the model and can be changed with direct modeling operations.

- **Flip Side**: Allows you to change the relative position of an entity (vectors normal of the selected faces point in the same direction or in the opposite direction). This option can be applied to Parallel, Coincident and Concentric constraints only and on condition the constraint is applied to faces.





- **Select geometry**: Selects the geometry that is affected by the constraint, in the drawing.

10.1.9 The Parameters and Constraints panel

The **Parameters and Constraints** panel allows you to browse and manage global parameters of your model in tabular form. Every row in this table corresponds to one global parameter, while every column represents a particular property of a parameter - name, expression, value.

In the **Parameters and Constraints** panel you can:

- Create a new parameter
- Delete an existing parameter
- Click a parameter name field and change it
- Click a parameter expression field and change it

- Search a parameter by name
- Sort all parameters by name, expression, and value
- Hide all parameters except those relevant to the selected parameter
- Hide all parameters except those dependent on the selected parameter
- Animate a parameter

10.2 Operations and functions in expressions

10.2.1 Operators

Operation	Character	Description
Addition	+	Produces the sum of numeric values.
Subtraction	-	Subtracts the two numeric values.
Multiplication	*	Multiplies the numeric values.
Division	/	Divides two numeric values.
Exponentiation	^	Calculates the exponential value of the given numbers.
Remainder operator	%	Gives the remainder after the division of one numeric value to another. e.g. 5%2 = 1, because 5/2 leaves remainder of 1.

10.2.2 Expressions

Expressions are evaluated according to the standard mathematical rules of precedence:

1 Expressions within brackets; innermost sets first.

2 Standard operations order:
   - 1. exponent
   - 2. multiplication and division
   - 3. addition and subtraction

3 Operators of equal precedence from left to right.

The following functions can be used in expressions:

Function	Syntax
Cosine	cos(expression)

Function	Syntax
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2) 1
Smallest element in array	min(expression1;expression2) 1

Function	Syntax
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base	10 log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2) 1
Random decimal, 0-1	Random

(1) Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Nota:

- The constants **Pi=3.14...** and **e=2.72...** can be used in expressions. The constants names are not allowed to be used as a parameter or constraint name.
- The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

10.3 3D constraints

10.3.1 Variational Direct Modeling

Use geometric constraints (aka variational direct modeling) to model parametric solids and surfaces. Define constraints between edges and faces to modify solid and surface models.

As soon as a constraint is added, BricsCAD automatically modifies the model and these constraints are updated with subsequent modifications to the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraint creation order. It allows you to parametrize any feature of the 3D model without the worry of model creation history.

Dimensional constraints control the dimensions of the model. When such a constraint is introduced or the value of its parameter is changed, BricsCAD automatically updates the solid and surface geometry. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to its opposite faces.

BricsCAD provides the following 3D constraints:

- Geometrical constraints: **Fix** ( ), **Coincident** ( ), **Concentric** ( ), **Parallel** ( ), **Perpendicular** ( ), **Tangent** ( ), and **Rigid Set** ( ).
- Dimensional 3D constraints: **Distance** ( ), **Radius** ( ), **Angle** ( ), **Cone Angle** ( ), **Path Constraint** ( ).

3D Constraints are available in the **3D Constraints** toolbar, the **Parametric/3D Constraints** menu, the **Home / Parametrize** ribbon panel and the **Constraints** tab on the Quad.

All 3D constraints are listed in the **Mechanical Browser** panel. Select a constraint to check and edit its properties.

The following (sub)entities and items are supported for 3D constraint creation:

- Vertices, edges and faces of 3D solids, surfaces, meshes and regions and surfaces of planar, cylindrical, spherical, toroidal and conical geometry.
- Lines, rays, xlines, circles and arcs.
- Segments and vertices of 3D polylines.
- Arbitrary curves for coincident and path constraints with a point as another argument.
- Planes, axes and origin points of the WCS or a block coordinate system.

3D constraints have a number of properties that allow you to control how BricsCAD treats them in various situations.

10.3.2 Geometry-driven constraints

Dimensional 3D constraints can also be **geometry-driven**. They update the model to accurately reflect the current state of the model. You can not specify their expression. Instead, their value is obtained from the model and can be changed with direct modeling operations and changes to other constraints.

Other parameters, including constraints, can depend on the values of geometry-driven constraints. In some circumstances a constraint dependent on geometry-driven constraints can make the model unresolvable. This should be avoided.

You can turn regular constraints into geometry-driven constraints and vice versa, in the **Mechanical Browser** panel and the **Parameters and Constraints** panel or using the DMCONSTRAINT3D command. After a regular constraint is turned into a geometry-driven one, its expression will be overwritten with the first modification of the model that affects this constraint.

10.3.3 3D constraints and direct modeling

3D constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint that fixes the placement of faces or edges, this constraint will be preserved during Push/Pull, Move and Rotate operations. A 3D constraint can be temporarily disabled, or erased permanently in the **Mechanical Browser** panel.

Geometry-driven constraints update their values when a direct modeling operation is completed, which possibly leads to additional charges in the model.

If some faces or edges disappear during direct modeling or Boolean solid editing operations, 3D constraints applied to such entities are removed. However, if there is an exact correspondence between the initial faces and the faces obtained, 3D constraints are automatically applied to the new entities.

Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometrical relations between the surfaces of a solid and will preserve them during direct modeling operations. This feature is called 'design intent recognition'.

10.4 Suppression feature

The suppression feature temporarily suppresses a set of entities in your drawing. If the condition is equal to zero, all entities are displayed. Otherwise, entities are suppressed.

Suppressed entities are internally placed on a special frozen layer. This allows you to speed up orbit, zoom and pan operations, improves object selection performance and reduces regeneration time for complex drawings. Suppressed entities are also excluded from mass properties calculations (generated drawing views and bill of materials). Use a parametrically-driven expression for the condition parameter of a suppression feature, to create different design variants of your parametric components.

10.4.1 Creating a suppression feature

Execute the SUPPRESS command.

10.4.2 Managing suppression features

Select the suppression feature in the **Mechanical Browser** panel, then right-click to display a context menu.

The available options are:

- **Suppress**: Suppresses the entity set.
  - The **Suppressed** icon ( ) indicates the suppressed state.
  - The option now reads **Unsuppress**.
- **Unsuppress**: Unsuppresses the entity set.
  - The **Unsuppressed** icon ( ) indicates the unsuppressed state.
  - The option now reads **Suppress**.
- **Dissolve**: Deletes the suppression feature.
- **Edit Selection**: Allows you to add/remove entities from the selection.

  All members of the selection are highlighted.
  - Select entities to add.
  - Shift + Select (hold down the Shift key, then use a selection method on) entities to remove.
  - Right-click to stop editing.
- **Zoom to**: Zooms onto selected entities.

  To compose a selection set:
  - Click a suppression feature to select all its members.
  - Expand a suppression feature, then click to select an entity.

- Ctrl + Click to add/remove entities from the selection.

- **Select**: Selects all members of a suppression feature.

10.4.3 Setting a condition for a suppression feature

You can set an expressional condition for a suppression feature. Depending on the value of the current expression, the entities will be suppressed or unsuppressed automatically.

In the Mechanical Browser panel:

1 Select the suppression feature (use the "Search" field to search by name).

2 Enter an expression in **Condition** field in the properties grid.

In the Parameters and Constraints panel:

1 Select the parameter with the same name as the suppression feature (use the "Search" field to search by name).

2 Enter an expression in the **Expression** column.

10.5 Working with design tables

A design table allows you to evaluate several global parameters at once with the value of one configuration parameter. You can create design tables from scratch in BricsCAD or import them from a .csv file (a comma-separated values text file). You can create such files with a spreadsheet software package or a text editor.

10.5.1 Creating a design table from a .csv file

The **From file** option of the DESIGNTABLE command imports .csv files that contain sets of parameter values, or else creates them from scratch. Parameters are added to the **Mechanical Browser** panel and to parametric component properties.

The expected format of the .csv file is:

Configuration_Name,Parameter1,Parameter2,...,ParameterN

Configuration_1,Value1Parameter1,Value1Parameter2,...,Value1ParameterN

Configuration_2,Value2Parameter1,Value2Parameter2,...,Value2ParameterN

Configuration_3,Value3Parameter1,Value3Parameter2,...,Value3ParameterN

...

The first row lists the parameter names separated by a comma. The first name must be different from the name of any global parameter in your drawing. It is called the configuration name. No spaces are allowed in the configuration name. Other names in the first row can be identical to the names of a global parameter The next rows list possible values for these parameters as strings or numbers.

Example:

Size,Length,Width,Height

Small,300,200,100

Medium, 450,300,150

Large,600,400,200

When the .csv file is imported into BricsCAD using the DESIGNTABLE command, a special global parameter is created by the name of the first name in the first row of the .csv file ('Size' in the above example). This is called the 'configuration parameter'. It will drive the values of other parameters in the design table. These are called 'configuration-dependent parameters'.

Nota: BricsCAD does not maintain an association with the .csv file used to create a design table: if this file is modified, then the design table is not updated automatically. Use the -DESIGNTABLEEDIT command to replace a design table with a new or modified .csv file.

10.5.2 Direct lookup in a design table

To evaluate a configuration parameter:

- Select it in the **Mechanical Browser** panel and type the desired value in its **Expression** field.
- Select a value in the drop-down list of the **Value** field:

If you change the value of a configuration parameter, all configuration-dependent parameter values change automatically in the corresponding design table. The values of these parameters are taken from the row in the design table. The first element coincides with the current value of the configuration parameter.

10.5.3 Reverse lookup in a design table

You can evaluate parameters that are defined by a configuration parameter with a design table independently.

Select the dependent parameter and apply an expression to it.

- If the value of this expression matches one of the values in the corresponding design table, then the configuration parameter and all dependent parameters of the design table will automatically change to the values specified in the corresponding row in the table.
- If several configurations are possible for a particular value of a dependent parameter, BricsCAD selects the one closest to the current values of the other parameters.
- If you enter an expression and the result is not available in the design table, or if you enter expressions for two or more design table parameters and the values cannot be found in a single configuration (the values are not found in the same row in the design table), then the values of the configuration parameter are changed to 'Custom' and the values of the other dependent parameters are not changed. This case corresponds to a configuration that does not exist in the design table.

10.5.4 Adding a new configuration to a design table

You can add any 'Custom' configuration to a design table and assign it a name. To do so, right-click the configuration parameter in the **Mechanical Browser** and select **Save current configuration** in the context menu. The configuration will be added to the design tables as Configuration_N:

- **Configuration**: the name of the configuration parameter.
- **N**: an index number - one more than the current number of configurations in the design table.

If you want to save a configuration in the design table under a different name, launch the -DESIGNTABLEEDIT command.

10.5.5 Creating an empty design table

If you use the procedure described above, it is possible to start your design with an empty design table and then fill it with the desired values.

Use the **Empty** option of the DESIGNTABLE command.

You will be prompted to specify:

- The name of a new design table
- The name of the corresponding configuration parameter
- The names of global parameters that will be dependent on this design table.

10.5.6 Using a parametric component with design tables

You can use both configuration parameters and configuration-dependent parameters in assemblies. If you then insert such a component into your drawing, its parameters will include the exposed parameters of design tables. A drop-down list of available values will be available. Both direct and reverse lookup actions are supported for such parameters.

You must associate the design table with the parametric component drawing first.

1. Open the drawing.
2. On the **Mechanical Browser** panel, you will see the list of parameters and their values.
3. Launch the DESIGNTABLE command and press Enter to accept the **From file** option.

   The **Select design table file** dialog box appears.
4. Select the .csv file, then double-click or click the **Open** button.
5. Check the **Value** property for each of the main parameters. You can select the default values to use when you insert the component.
6. Check the **Exposed** property for each of the main parameters. They should all be set to **Auto**.
7. Save the component drawing.

10.5.7 Other actions with design tables

The -DESIGNTABLEEDIT command allows you to:

- Delete a design table
- Replace a design table
- Link a new parameter
- Unlink an existing parameter
- Add new configurations, remove an existing one
- Export a design table as a .csv file.

11. Nuvola di punti

11.1 Working with Point cloud

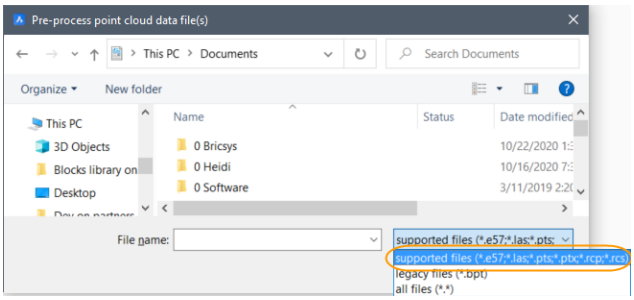
11.1.1 Commands

,,,,,,,, ,, ,,,,,,, ,, ,,

Point cloud functionality is significantly faster and more powerful in BricsCAD v21 and is even more powerful with a BIM license.

11.1.2 Pre-processing

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

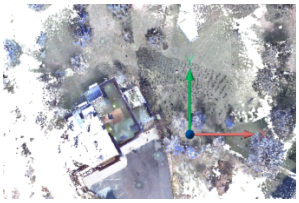


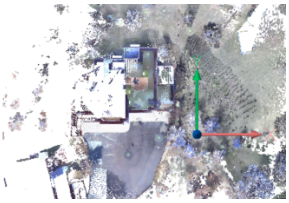
When you attach a point cloud that requires pre-processing, it converts 5 to 8 times faster in BricsCAD v21 than in v20.

The log file can be found in *C:\Users\USERNAME\AppData\Roaming\Bricsys\BricsCAD \V21x64\en_US\PointCloudCache\{Folder_for_processed_pointcloud}*, only if this path is set in POINTCLOUDCACHEFOLDER system variable. Inside this folder, the current state of processing can be observed.

11.1.3 Alignment

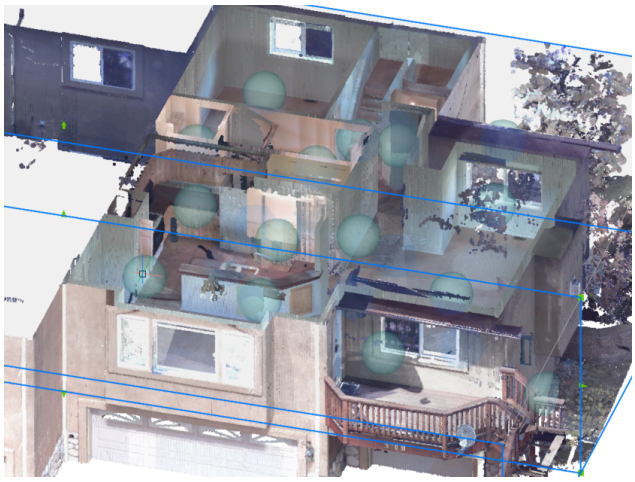
The new POINTCLOUDALIGN command automatically rotates a point cloud to optimally align it with the X and Y axis. To determine the best alignment, it can analyze the entire point cloud, or you can specify the most relevant area.





11.1.4 Bubble Viewer

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

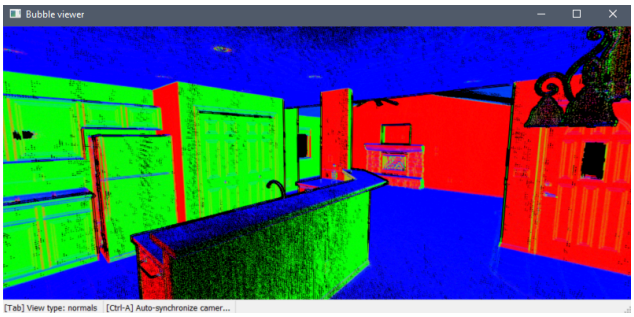


Double-click one of the bubbles in model space to open the Bubble viewer. You can press the middle mouse button and move the mouse to view the point cloud in any direction from that scan location. You can also zoom in and out using the mouse wheel.

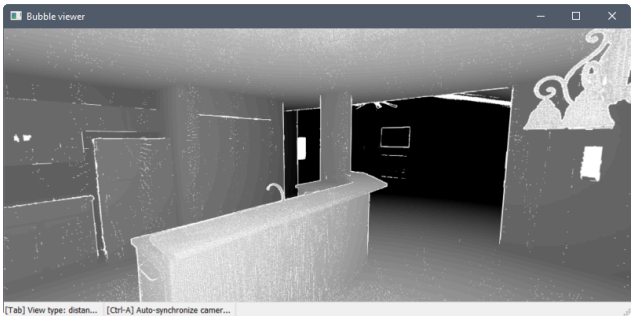
Press the Tab key to cycle between three different visual modes.

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

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



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

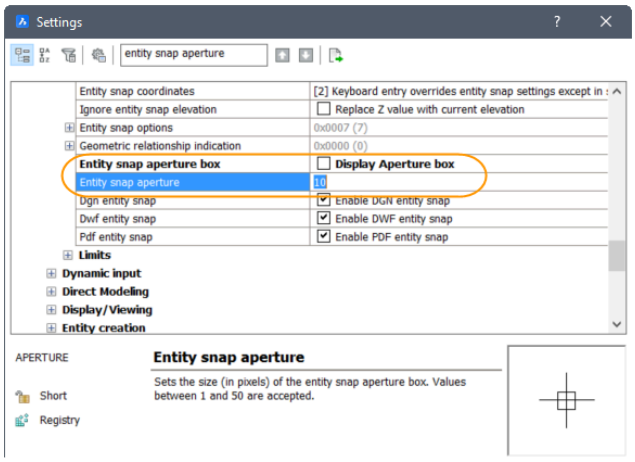


You can easily sync the drawing view to match the Bubble viewer by pressing Ctrl A.

11.1.5 Entity snaps

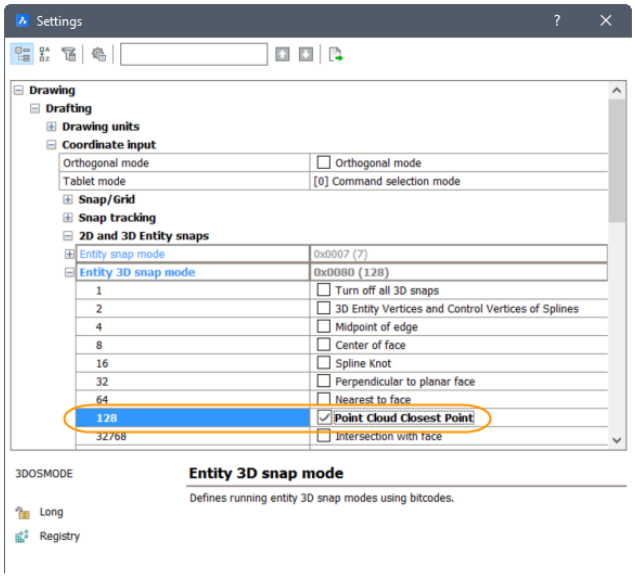
The new Point Cloud Nearest Point entity snap significantly improves your ability to select relevant point cloud points. It uses an imaginary cylinder from the current viewpoint toward the cursor.

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



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

Nota: Make sure that only the bit 128 of 3DSMODE is set.



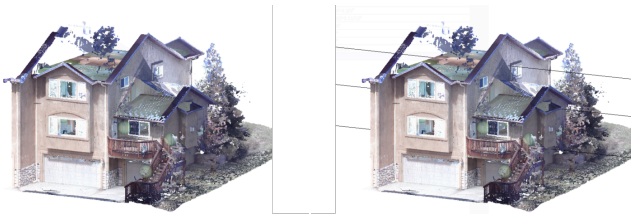
11.1.6 Export

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

11.1.7 Floor Detection

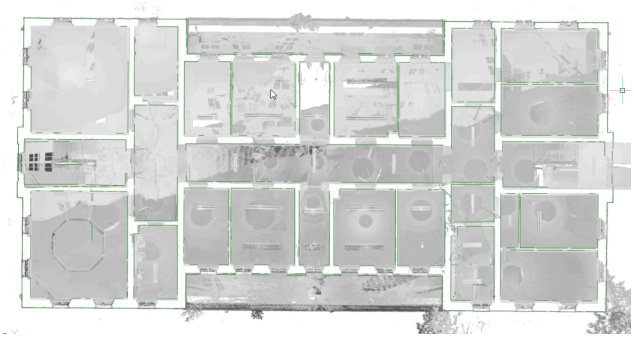
SectionPlanes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and SectionPlanes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

The new POINTCLOLUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

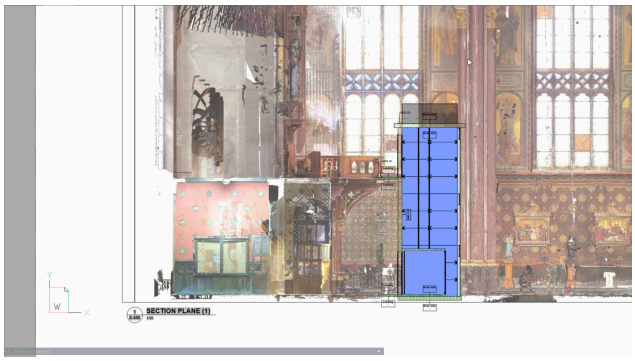


11.1.8 Point cloud projection

The newPOINTCLOUDPROJECTSECTIONcommand enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building. You can use these sections to generate 2d lines to create a 2d floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.



At the same time, a raster image will be generated. In some cases, it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2d geometry but in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern intervention are made in historical buildings.

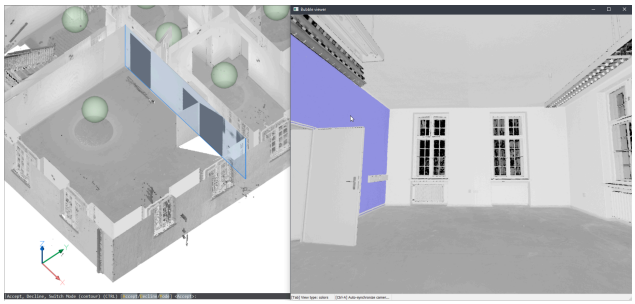


11.1.9 Planar Fit

The 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.

11.1.10 In bubble view

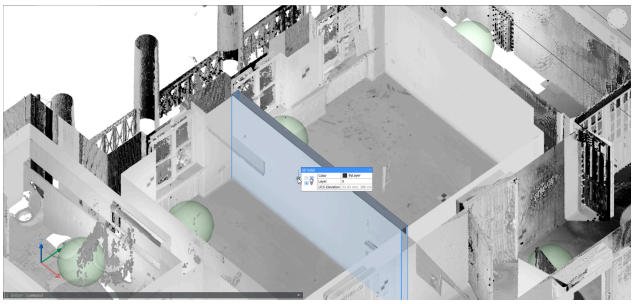
If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the 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. Blocchi, attributi, riferimenti esterni e componenti

12.1 Creazione blocchi

Il comando BLOCCO raggruppa le entità esistenti in un blocco utilizzando una finestra di dialogo.

Il comando -BLOCCO crea blocchi da entità esistenti e utilizza la barra dei comandi per le richieste necessarie.

I comandi MODIFBLOC e CHIUDIBLOC creano o modificano blocchi in una sessione di disegno temporanea.

Il comando MBLOCCO esporta blocchi ed altre parti di disegno in un file di disegno separato.

Il comando INCOLLABLOC incolla le entità dagli **Appunti** come blocco nel disegno corrente.

Il comando PIANOSEZABLOCCO salva il risultato del piano di sezione selezionato in un blocco 2D di sezione trasversale/elevazione o in un blocco di sezione 3D.

12.1.1 Creare un blocco

1 Opzionalmete selezionare le entità per il blocco.

2 Avviare il comando BLOCCO.

3 Viene visualizzata la finestra di dialogo **Definizione Blocco**:

4 Digitare un nome per il nuovo blocco nel campo **Nome**.

5 Effettuare una delle seguenti operazioni per specificare il **Punto Base**:
- Selezionare l'opzione **Specifica su schermo**.

  Dopo aver fatto clic sul pulsante **OK** verrà richiesto di specificare il punto di base.
- Cliccare sul pulsante **Seleziona Punto** ( ).

  La finestra di dialogo **Definizione Blocco** si chiude temporaneamente per consentire di scegliere un punto nel disegno.
- Digitare le coordinate del punto base nei campi X, Y- e Z.

6 Viene indicato il numero di entità preselezionate nel passaggio 1.

7 Per selezionare le entità di origine del nuovo blocco, effettuare una delle seguenti operazioni:
- Selezionare le entità prima di avviare il comando BLOCCO.
- Selezionare l'opzione **Specifica su schermo**.

  Verrà richiesto di selezionare le entità dopo aver cliccato sul pulsante **OK**.
- Cliccare sul pulsante **Seleziona entità** ( ).

  La finestra di dialogo **Definizione Blocco** si chiude temporaneamente per consentire di selezionare le entità nel disegno.

8 Impostare le proprietà del blocco:
- **Annotativo**: crea un blocco annotativo.
- **Abbina l'orientamento del blocco al layout**: se spuntata, l'orientamento di un blocco annotativo viene mantenuto in una finestra ruotata.

- **Scala uniformemente**: se spuntata, la scala X, Y e Z non può essere impostato in modo diverso.
- **Consenti esplosione**: se non spuntata, impedisce l'esplosione del blocco.
- **Unità del blocco**: imposta l'unità di inserimento del blocco (vedere anche: Inserimento Blocchi).

9 Impostare il comportamento delle entità selezionate dopo che il blocco è stato creato:
- **Mantieni**: le entità selezionate rimangono nel disegno.
- **Converti in blocco**: le entità selezionate verranno sostituite dal blocco appena creato.
- **Elimina**: le entità selezionate verranno eliminate.

10 Fare clic sul pulsante **OK** per creare il blocco.

Note:

- Se è stata impostata l'opzione di **Elimina** nel passaggio 6, il comando OOPS ripristina le entità del blocco cancellate.
- Se si digita un nome di un blocco esistente nel passaggio 3, verrà richiesto di sovrascrivere la definizione del blocco esistente. Le istanze del blocco esistenti verranno sostituite con la nuova definizione del blocco.

12.1.2 Salvare un blocco in un file separato

1 Opzionalmete selezionare le entità per il blocco.

2 Avviare il comando MBLOCCO.

3 Viene visualizzata la finestra di dialogo **Scrivi Blocco su File**.

4 Cliccare sul pulsante sfoglia accanto al campo **Nome e percorso**.

Viene visualizzata la finestra di dialogo **Salva Blocco**.

5 Selezionare una cartella e digitare un nome nel campo **Nome file**, quindi cliccare sul pulsante **Salva**.

La finestra di dialogo **Salva blocco** si chiude.

6 **Unità inserimento**: imposta il valore della variabile di sistema INSUNITS per il file del blocco.

Per impostazione predefinita, questo è il valore INSUNITS del disegno di origine.

7 Viene indicato il numero di entità preselezionate nel passaggio 1.

8 Per selezionare le entità di origine del nuovo blocco, effettuare una delle seguenti operazioni:
- Cliccare sul pulsante di opzione **Blocco**, quindi selezionare un blocco dall'elenco a discesa.
- Cliccare sul pulsante di opzione **Intero disegno** per selezionare tutte le entità nello spazio modello del disegno corrente.
- Cliccare sul pulsante di opzione **Entità**, quindi sul pulsante **Selezionare entità**.

  La finestra di dialogo **Scrivi Blocco su File** si chiude temporaneamente per consentire la selezione delle entità nel disegno.

  Cliccare il pulsante destro del mouse per interrompere la selezione delle entità.

9 Cliccare sul pulsante **Scegli punto** ( ) per definire il **punto base** del nuovo disegno

o

Digitare le coordinate del punto base nei campi X, Y- e Z.

10 Impostare il comportamento delle entità selezionate dopo che il blocco è stato creato:
   - **Mantieni**: le entità selezionate rimangono nel disegno.
   - **Converti in blocco**: le entità selezionate verranno sostituite dal blocco appena creato.
   - **Elimina**: le entità selezionate verranno eliminate.

11 Cliccare sul pulsante **OK** per creare il file.

Nota: Se è stata impostata l'opzione **Elimina** nel passaggio 6, il comando OOPS ripristina le entità del blocco cancellate.

12.2 Utilizzo di Rileva Blocchi

Lo strumento Rileva Blocchi consente di convertire automaticamente gruppi ripetitivi di entità in definizioni di blocco. Selezionare un insieme di entità 2D o solidi 3D e BricsCAD cercherà nel disegno gruppi identici di entità e li sostituisce automaticamente con riferimenti di blocco.

Per ulteriori informazioni su questo comando, consultare l'articolo BLOCKIFY del Riferimento Comandi.

12.2.1 Rileva Blocchi da un input di selezione

1 Avviare il comando BLOCKIFY.

1 **Viene richiesto**: Selezionare le entità di origine o [corrispondenza solidi Uguali/corrispondenza Blocchi esistenti/Impostazioni/opzioni di selezione (?)] <corrispondenza solidi Uguali>:

2 Selezionare le entità che si desidera convertire in un blocco e premere Invio.

3 Entità selezionate: xx

Viene richiesto: Selezionare le entità di origine o [Impostazioni/opzioni di selezione (?)]:

4 Selezionare altre entità o premere Invio o cliccare il pulsante destro del mouse per interrompere la selezione.

5 **Viene richiesto**: Selezionare il punto di inserimento del blocco o [usa Punto predefinito]:

6 Identificare il punto di inserimento del blocco.

7 **Viene richiesto**: Selezionare area di ricerca o [Intero disegno/opzioni di selezione (?)] <Intero disegno>:

8 Premere Invio o cliccare il pulsante destro del mouse per cercare l'intero disegno

9 **Viene richiesto**: Specificare il nome del blocco o <INVIO per usare il nome predefinito>:

10 Digitare un nome per il blocco, quindi premere Invio.

11 Report di BricsCAD:

Durata della ricerca di gruppi simili: x.xx sec

Numero di elementi blocco: xx

Nota: A seconda del valore della variabile di sistema BLOCKIFYMODE, alcune richieste vengono eliminate e i valori predefiniti vengono utilizzati per il punto di inserimento, lo spazio di ricerca e/o il nome del blocco.

12.2.2 Definizioni di blocco esistenti

Uno degli obiettivi del comando BLOCKIFY è quello di aggiungere e migliorare automaticamente la struttura di un modello. Se il modello contiene già definizioni di blocco, vengono verificate le definizioni di blocco esistenti.

- **Utilizza un gruppo di entità di input selezionato:**: quando una delle definizioni dei blocchi esistenti corrisponde al gruppo di input di BLOCKIFY, non verrà creata nessuna definizione. Tutte le istanze dell'input verranno sostituite da riferimenti di blocco alla definizione di blocco esistente.
- **Abbina blocchi esistenti**: L'intero disegno, o gruppo di selezione, viene ricercato per la geometria corrispondente nelle definizioni di blocco esistenti. Pertanto, la geometria ripetuta corrispondente a una definizione di blocco viene sostituita da riferimenti di blocco alle definizioni esistenti. Questa opzione consente di ricreare istanze di blocco esplose accidentalmente.

12.2.3 Combina definizioni di blocco duplicate

Le definizioni di blocco identiche possono essere combinate in un unico blocco e, opzionalmente, eliminare il duplicato utilizzando il comando ELIMINADUPLICATI.

1 Avviare il comando ELIMINADUPLICATI.

1 **Viene richiesto**: Selezionare entità o [Combina definizioni di blocco duplicate/opzioni di selezione (?)]:

2 Scegliere l'opzione **Combina definizioni di blocchi duplicate**.

Viene visualizzata la finestra di dialogo **Elimina Entità Duplicate**.

3 (opzionale) Selezionare le proprietà da ignorare.

4 (opzionale) Selezionare l'opzione **Rimuovi blocchi duplicati**.

5 Cliccare sul pulsante **OK**.

Report di BricsCAD:

- Numero di definizioni di blocco duplicate trovate
- Nomi delle definizioni di blocco duplicate
- Definizioni dei blocco eliminate

12.3 Modificare i blocchi

È possibile accedere all'Editor Blocchi con il comando MODIFBLOC o mediante il doppio clic sul blocco che si desidera modificare. Quando si utilizza l'Editor Blocchi, BricsCAD visualizza le entità di blocco in una sessione di disegno temporanea. Include tutti gli strumenti necessari per creare e modificare una definizione di blocco. Per uscire dalla sessione di modifica dei blocchi, utilizzare il comando CHIUDIBLOC e scegliere se salvare o annullare le modifiche. Se si seleziona il comando SALVA durante la sessione di modifica dei blocchi, BricsCAD salva le modifiche alla definizione del blocco ed esce dall'editor blocchi.

I blocchi possono anche essere modificati utilizzando il comando MODRIF.

12.4 Inserimento blocchi

Il comando INSER permette di inserire blocchi attraverso una finestra di dialogo.

Il comando -INSER inserisce blocchi richiedendo i dati necessari nella barra dei comandi. BricsCAD cerca il nome del blocco nelle definizioni dei blocchi presenti nel disegno corrente. Se il blocco non viene trovato nel disegno corrente, viene ricercato nei percorsi definiti dalla variabile di sistema SRCHPATH. Se il blocco non si trova nemmeno qui, BricsCAD risponde 'Impossibile trovare il file <nomeblocco>'.

Il comando INSERALLINEATO inserisce un'entità blocco, mediante facile allineamento alle entità esistenti. Il comando INSERALLINEATO è simile al comando INSER, ma durante il posizionamento il blocco si allinea dinamicamente a linee, polilinee, archi o cerchi esistenti. Il blocco viene allineato con l'entità a cui si aggancia il cursore. Se nessuna entità viene agganciata, il blocco viene allineato con l'asse X del WCS.

Il comando INSERM inserisce un blocco come serie rettangolare; combina i comandi -INSERT e SERIE (abbreviazione di inserimento "multiplo").

È possibile scegliere di selezionare una definizione di blocco esistente o di inserire un intero disegno come blocco. Quando si inserisce un disegno, una nuova definizione di blocco viene creata nel disegno corrente. Se si cambia il file originale di disegno, questi cambiamenti non hanno alcun effetto sul disegno corrente, a meno che non si ridefinisca il blocco reinserendo il disegno modificato. Quando un blocco viene inserito in un disegno, viene trattato come una singola entità.

Note:

- Quando si inserisce un disegno come blocco, l'origine è definita dalla variabile di sistema INSBASE del disegno inserito. La variabile è impostata dal comando BASE.
- Se un blocco contiene degli attributi si viene guidati a compilare il testo per ogni attributo nella barra dei comandi.
- Se la variabile di sistema DRAGOPEN è impostata su zero (OFF), i disegni possono essere inseriti come blocchi trascinandoli dalla finestra dell'**Esplora Risorse di Windows** nel disegno corrente, se, invece, DRAGOPEN = 1 (ON), il disegno verrà aperto come nuovo file.
- Le variabili di sistema INSUNITS e INSUNITSSCALING controllano la scalatura automatica quando si inseriscono blocchi o si attaccano Xrif quando il valore di INSUNITS è diverso nel blocco o Xrif e nel disegno di destinazione.
- Se il valore di INSUNITS nel disegno di origine è 0 (non specificato), viene invece utilizzato il valore della variabile di sistema INSUNITSDEFSOURCE.
- Se il valore di INSUNITS nel disegno di destinazione è 0 (non specificato), viene utilizzato il valore della variabile di sistema INSUNITSDEFECTARGET.
- I valori di INSUNITSDEFSOURCE e INSUNITSDEFECTARGET vengono salvati nel Registro di sistema e quindi si applicano a tutti i disegni in cui il valore della variabile di sistema INSUNITS non è specificato (senza unità).

- Quando si digita il nome di un blocco alla richiesta "Blocca da inserire:", BricsCAD cercherà il blocco nelle definizioni dei blocchi nel disegno corrente. Se il blocco non viene trovato, viene cercato nei percorsi definiti dalla variabile di sistema SRCHPATH. Se il blocco non si trova nemmeno qui, BricsCAD risponde 'Impossibile trovare il file <nomeblocco>'.

Nella finestra di dialogo **Inserisci Blocco**:

- Il campo **Unità** sotto **Unità Blocco** è impostato dalla variabile di sistema INSUNITS nel disegno di origine o dalla proprietà **Unità Blocco** di una definizione di blocco (vedere Creazione di blocchi).
- Il campo **Fattore** esprime la relazione tra il valore della variabile INSUNITS nel disegno di origine e il disegno di destinazione. Ad es., se INSUNITS è Millimetri nella sorgente e nella destinazione, il valore del campo **Fattore** è 0.1.

12.4.1 Utilizzare il comando INSER

1 Avviare il comando INSER.

1 Viene visualizzata la finestra di dialogo **Inserisci Blocco**.

2 Eseguire una delle seguenti operazioni:
   - Selezionare una definizione di blocco esistente nell'elenco **Nome**.

     Il nome del blocco selezionato viene visualizzato nel campo **Nome**.
   - Cliccare sul pulsante **Sfoglia** e selezionare un file di disegno.

     Il campo Percorso mostra la posizione del disegno selezionato.

3 (opzionale) Selezionare l'opzione **Usare le curve guida per l'inserimento**.

4 Impostare la modalità **Punto di inserimento**, **Scala** e **Rotazione**.

   Le opzioni sono:
   - Specificare le impostazioni nella finestra di dialogo **Inserisci Blocco**.
   - **Specifica su schermo**: verrà richiesto durante il processo di inserimento.

5 (opzionale) Selezionare l'opzione **Esplodi** per esplodere il blocco dopo l'inserimento.

6 Cliccare sul pulsante **OK**.

   Verrà richiesto in base alle opzioni selezionate.

12.4.2 Per inserire un blocco utilizzando la finestra di dialogo Esplora Disegno

Inserimento di blocchi dal disegno corrente.

1 Avviare il comando EXPBLOCKS.

1 Verrà visualizzata la finestra di dialogo **Esplora Disegno – Blocchi**.

2 (opzionale) Impostare la modalità di visualizzazione: **Dettagli** o **Icone**.

3 (opzionale) Selezionare un blocco, quindi cliccare il pulsante destro del mouse e scegliere **Opzioni...** nel menu contestuale.

   Viene visualizzata la finestra di dialogo **Opzioni Esplora Disegno**.

   Impostare le opzioni di inserimento, quindi cliccare sul pulsante **OK**.

4 Doppio clic sull'icona del blocco o sul numero quando si è in modalità **Vista dettagli**.

La finestra di dialogo **Esplora Disegno** si chiude per consentire di inserire il blocco nel disegno.

5 Specificare il punto di inserimento.
   - A seconda delle opzioni di inserimento correnti, viene richiesto di specificare la scala e/o la rotazione.
   - Quando l'opzione **Allinea** è selezionata, è necessario specificare la posizione del blocco rispetto all'entità lineare selezionata.

     Viene richiesto: Inserisci punto di controllo per la specchiatura blocco:

Viene riaperta la finestra di dialogo **Esplora Disegno**.

Inserimento blocchi contenuti in un altro disegno.

1 Aprire la finestra di dialogo **Esplora Disegno**.

2 Nel riquadro **Disegni**, cliccare sulla scheda **Cartelle**.

3 (opzionale) Se non è ancora disponibile alcuna cartella nella struttura **Cartelle locali** o se si desidera aggiungere una cartella, cliccare sul pulsante **Aggiungi cartella locale...**

Viene visualizzata la finestra di dialogo **Scegli una cartella**.

Individuare la cartella che si desidera aggiungere, quindi cliccare sul pulsante **Seleziona Cartella**.

La cartella e le relative sottocartelle vengono aggiunte nell'albero **Cartelle locali**.

4 Selezionare un disegno nell'albero **Cartelle locali**.

5 Espandere il disegno, quindi selezionare **Blocchi**.

Tutti i blocchi nel disegno selezionato vengono visualizzati nel riquadro **Blocchi**.

6 Procedere dal punto 2 della procedura precedente.

12.4.3 Inserire un blocco allineato ad un'entità

1 Assicurarsi che siano attivi gli Snap ad Entità appropriati.

2 Avviare il comando INSERALLINEATO.

3 **Viene richiesto**: Blocco da inserire (~ per aprire la finestra di dialogo file) o [? per elencare i blocchi nel disegno] <corrente>:

4 Eseguire una delle seguenti operazioni:
   - Premere Invio per accettare il blocco attualmente selezionato.
   - Digitare il nome del blocco da inserire.
   - Premere il ? per vedere una lista di blocchi disponibili.

Viene richiesto: Punto di inserimento del blocco o [Scala/X scala/Y scala/Z scala/blocchi Multipli]:

5 (opzionale) Scegliere l'opzione **blocchi Multipli** o specificare una scala fissa.

6 Specificare un punto.

7 Il blocco visualizza il punto di snap ad entità selezionato, allineato con l'entità lineare.

Viene richiesto: Inserisci punto di controllo per la specchiatura blocco:

8 Spostare il cursore per selezionare la posizione desiderata, quindi cliccare per inserire il blocco.

Nota: Il blocco è allineato come segue:

Il blocco viene ruotato in modo che il suo asse locale Z sia parallelo con l'asse Z dell'UCS attivo, ed il suo asse locale X sia parallelo con la tangente all'entità nella posizione del punto di snap ad entità. Cliccando un punto su un'entità il blocco viene allineato con l'entità. Poi, spostando il puntatore del mouse intorno al punto di inserimento, è possibile scegliere come il blocco deve essere specchiato. Il punto di origine del blocco si trova sull'entità.

12.4.4 Inserire più istanze di un blocco in una serie rettangolare

1 Avviare il comando INSER.

1 **Viene richiesto**: Blocco da inserire (~ per aprire la finestra di dialogo file) o [? per elencare i blocchi nel disegno] <corrente>:

2 Eseguire una delle seguenti operazioni:
- Premere Invio per accettare il blocco attualmente selezionato.
- Digitare il nome del blocco da inserire.
- Premere il ? per vedere una lista di blocchi disponibili.

Viene richiesto: Punto di inserimento del blocco o [Scala/X scala/Y scala/Z scala/blocchi Multipli]:

3 (opzionale) Scegliere l'opzione **blocchi Multipli** o specificare una scala fissa.

4 Specificare un punto.

5 **Viene richiesto**: Angolo rotazione del blocco <0°0'0">:

6 Premere Invio per accettare l'angolo di rotazione predefinito o specificare un angolo.

7 **Viene richiesto**: Digitare il numero di righe <1>:

8 Specificare un numero.

9 **Viene richiesto**: Digitare il numero di colonne <1>:

10 Specificare un numero.

11 **Viene richiesto**: Distanza verticale tra righe, o cella unitaria:

12 Eseguire una delle seguenti operazioni:
- Specificare un valore.
- **Viene richiesto**: Distanza orizzontale tra le colonne:

  Specificare un valore.
- Cliccare su due punti per specificare un rettangolo, che imposta le distanze di spaziatura orizzontale e verticale.

Il blocco viene inserito come entità **Serie MBlocco**.

Note:

- Scegliendo **Blocchi Multipli** al punto 3, viene richiesto di creare più istanze della serie di blocchi.
- Non è possibile esplodere un'entità **Serie MBlocco**.
- Puoi modificare le proprietà di un'entità **Serie MBlocco** nel pannello **Proprietà**.

12.5 Esplodere Blocchi

Se è la proprietà **Esplodibile** di un blocco è attiva, è possibile esplodere un blocco inserito alla sue entità nidificate originali. L'esplosione di un blocco interessa solo la singola istanza del blocco. La definizione del blocco originale rimane nel disegno e si può ancora inserire come blocco nel disegno. Se di esplode un blocco che contiene degli attributi, gli attributi verranno persi, ma rimangono le definizioni originali degli attributi.

La variabile di sistema EXPLMODE controlla se è possibile esplodere blocchi scalati in modo non uniforme.

12.5.1 Esplodere un blocco

1 Avviare il comando ESPLODI.

1 **Viene richiesto**: Seleziona entità da esplodere [opzioni di selezione (?)]:

2 Selezionare uno o più blocchi.

3 Entità nel gruppo: x

Viene richiesto: Seleziona entità da esplodere [opzioni di selezione (?)]:

4 Selezionare altre entità o premere Invio per interrompere la selezione.

5 I blocchi selezionati vengono esplosi.

Nota: L'esplosione dissocia le entità costituenti il blocco nel loro livello di complessità più semplice: blocchi o polilinee contenuti in un blocco diventano rispettivamente blocchi e polilinee.

12.5.2 Impostare la proprietà Esplodibile di un blocco

La proprietà **Esplodibile** di un blocco viene impostata durante la creazione del blocco utilizzando il comando BLOCCO.

Per modificare la proprietà, procedi come segue:

1 Avviare il comando EXPBLOCKS.

1 Verrà visualizzata la finestra di dialogo **Esplora Disegno – Blocchi**.

2 Scegliere la modalità **Vista dettagli**.

3 Cliccare sulla casella di controllo **Esplodibile** per attivare o disattivare la proprietà **Esplodibile**.

12.6 Blocchi parametrici 2D

Se provieni da un ambiente di progettazione AutoCAD®, potresti avere familiarità con i Blocchi Dinamici. È anche possibile avere familiarità con i vincoli parametrici 2D. Sono due diverse funzionalità che in qualche modo si sovrappongono.

Invece di offrire le funzionalità dei blocchi dinamici, BricsCAD si concentra interamente sui parametri. Consente di creare definizioni di blocchi 2D e 3D utilizzando lo stesso gruppo di strumenti e flusso di lavoro. Se non hai mai lavorato con vincoli parametrici, potrebbero sembrare spaventosi. Ma sono piuttosto intuitivi e logici. E, incredibilmente potenti!

12.6.1 Procedura generale per creare blocchi parametrici 2D

Creare la geometria.

Creare vincoli geometrici.

Creare vincoli dimensionali e parametri.

Modificare i vincoli dimensionali e i parametri nel pannello **Parametri e Vincoli** o nel pannello **Navigatore Meccanico**.

Facoltativamente, creare stati di visibilità per mostrare/nascondere la geometria, utilizzando il comando VISIBILITYSTATES.

Salvare il disegno come blocco di libreria, utilizzando il comando CREATELIBRARYBLOCK.

Nota: Il comando PARAMETRIZE2D crea automaticamente vincoli geometrici e dimensionali.

12.6.2 Inserimento di un blocco parametrico

È possibile inserire i blocchi parametrici nel disegno utilizzando i comandi INSER o BMINSERT o dal pannello **Libreria**.

12.6.3 Impostazione dei valori dei parametri

Selezionare il blocco parametrico, quindi modificare i valori dei parametri esposti in **Parametri** nel pannello **Proprietà**.

12.7 Blocchi Dinamici

Se provieni da un ambiente di progettazione AutoCAD®, potresti avere familiarità con i Blocchi Dinamici. È anche possibile avere familiarità con i vincoli parametrici 2D. Sono due diverse funzionalità che in qualche modo si sovrappongono.

Se si inseriscono blocchi dinamici AutoCAD nel disegno di BricsCAD, è possibile, nella maggior parte dei casi, modificare tali istanze di blocco. Tuttavia, non è possibile creare o modificare definizioni di blocchi dinamici in BricsCAD. Invece di offrire le funzionalità dei blocchi dinamici, BricsCAD si concentra interamente sui parametri. Consente di creare definizioni di blocchi 2D e 3D utilizzando lo stesso gruppo di strumenti e flusso di lavoro. Se non hai mai lavorato con vincoli parametrici, potrebbero sembrare spaventosi. Ma sono piuttosto intuitivi e logici. E, incredibilmente potenti!

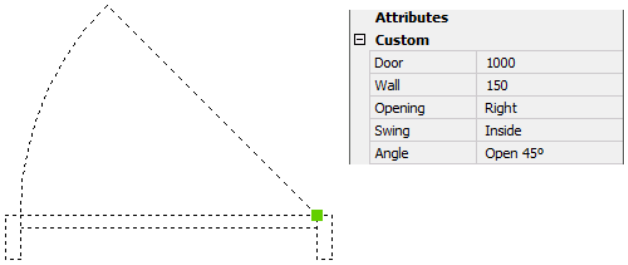
BricsCAD supporta parzialmente i blocchi dinamici, è possibile:

- inserire blocchi dinamici da librerie di blocchi dinamici esistenti
- modificare le proprietà personalizzate nel pannello **Proprietà**.
- modificare i blocchi dinamici mediante i grips

12.7.1 Modificare i blocchi dinamici

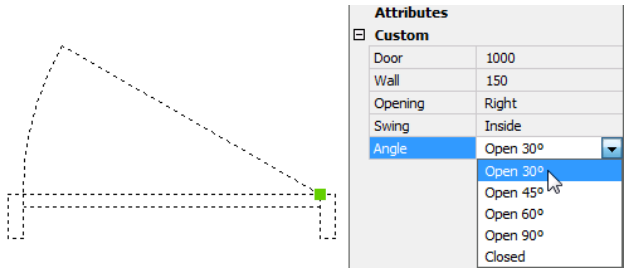
1 Selezionare il blocco dinamico nel disegno.

1 Le proprietà del blocco selezionato vengono visualizzate nel pannello **Proprietà**.



2 Sotto **Attributi** > **Personalizzato**: modificare le proprietà necessarie.

Il blocco dinamico viene aggiornato automaticamente.



12.7.2 Modificare i blocchi dinamici mediante i grips

1 Selezionare il blocco dinamico nel disegno.

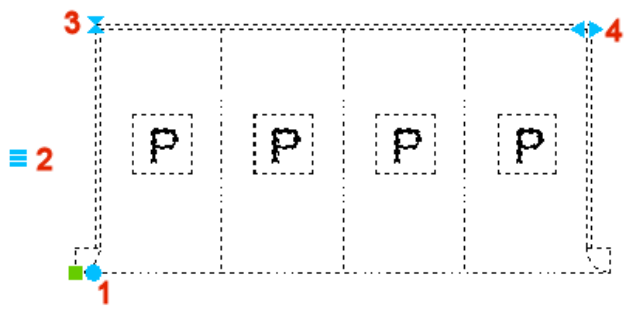
1 Vengono visualizzati i seguenti tipi di grip:

- Puntiforme
- XY
- Lineare
- Polare
- Rotazione
- Inverti
- Visibilità

2 Trascinare un grip per modificare il blocco.

3 Nell'esempio riportato di seguito:

- Ruota il blocco (1): trascinare il grip per ruotare il blocco in modo dinamico o digitare un valore nel campo di immissione dinamica.
- Mostra/nascondi entità (2): cliccare per selezionare un'opzione.
- Inverti (3): cliccare per specchiare il blocco
- Modifica la lunghezza (4): selezionare il grip per modificare dinamicamente la lunghezza o digitare un valore nel campo di immissione dinamica.



12.8 Definizione degli Attributi

Il comando DEFATT definisce gli attributi tramite una finestra di dialogo.

Il comando -DEFATT definisce gli attributi nella barra dei comandi.

12.8.1 Procedura generale per definire gli attributi:

- Specificare le caratteristiche dell'attributo, includendo il suo nome, l'istruzione e il valore predefinito.
- Specificare le coordinate di inserimento.
- Definite le proprietà del testo.
- (opzionale) Impostare le opzioni degli attributi.

12.8.2 Definire un attributo

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Attributi** > **Definisci attributi...** ( ) sulla barra degli strumenti **Strumenti**.
- Scegliere **Attributi > Definisci Attributi...** nel menu **Strumenti**.
- Avviare il comando DEFATT.

Viene visualizzata la finestra di dialogo **Definisci Attributo**.

2 Definire l'Attributo:

- Digitare un nome nel campo **Etichetta**.

  L'Etichetta identifica ogni occorrenza di un attributo nel disegno. Il nome digitato nel campo **Etichetta** può includere qualsiasi combinazione di caratteri tranne spazi. Le lettere minuscole vengono convertite automaticamente in maiuscole.
- Digitare una richiesta nel campo **Richiesta**.

  La richiesta dell'attributo viene visualizzata quando si inserisce un blocco contenente l'attributo. Se viene omessa una richiesta, come richiesta verrà utilizzata l'Etichetta.
- Eseguire una delle seguenti operazioni:
- Digitare un valore predefinito nel campo **Predefinito**.

  Per gli attributi variabili, il valore predefinito viene sostituito dal valore effettivo quando si inserisce un blocco contenente l'attributo.
- Cliccare sul pulsante **Inserisci Campo** ( ) per assegnare il valore di un campo all'l'attributo

(vedere Lavorare con i Campi).

- Se l'opzione **Linee Multiple** è selezionata, il campo **Predefinito** è disabilitato.
- Cliccare sul pulsante accanto al campo **Predefinito**.

  La finestra di dialogo **Definisci Attributo** si chiude temporaneamente per creare un'entità di TestoM.

- Eseguire una delle seguenti operazioni:
  - Specificare le coordinate X, Y e Z per il punto di inserimento dell'attributo.
  - Cliccare sul pulsante Seleziona punto ( ) per specificare il punto di inserimento nel disegno.

    Viene richiesto: Selezionare punto di inserimento.

    La finestra di dialogo **Definisci Attributo** si chiude temporaneamente.

- (opzionale) Impostare l'opzione attributo **Invisibile**.

  Gli attributi con l'opzione **Invisibile** impostato non vengono né visualizzati né stampati.

  Se la variabile ATTMODE (Modalità Visualizzazione Attributi) è impostata su **2**, vengono visualizzati tutti gli attributi, inclusi gli attributi **Nascosti**.

- (opzionale) Impostare l'opzione attributo **Costante**.

  Gli attributi con l'opzione **Constante** impostata non possono essere modificati.

- (opzionale) Impostare l'opzione attributo **Verifica**.

  Gli attributi con l'opzione **Verifica** impostata devono essere convalidati esplicitamente quando viene inserito un blocco contenente tali attributi.

- (opzionale) Impostare l'opzione attributo **Preimposta**.

  Quando si inserisce un blocco, non viene richiesto di definire gli attributi per i quali è impostato l'opzione **Preimposta**. È comunque possibile modificare gli attributi successivamente.

- (opzionale) Impostare l'opzione attributo **Blocca posizione** per bloccare la posizione dell'attributo all'interno del riferimento del blocco.

  Gli attributi sbloccati possono essere spostati rispetto al resto del blocco utilizzando la modifica mediante grip e gli attributi multilinea possono essere ridimensionati.

- Definire le proprietà **Testo**: **Stile del Testo**, **Giustificazione**, **Annotativa**, **Altezza Testo** e **Rotazione**.

3 Cliccare sul pulsante **OK** per creare l'attributo nel disegno.

  La finestra di dialogo **Definisci Attributo** si chiude.

12.8.3 Utilizzare le proprietà di un gruppo di fogli in un cartiglio

1 Creare il gruppo di fogli.

2 Definire il gruppo di fogli personalizzati e proprietà del foglio.

3 Aprire un foglio (layout) dal gruppo di fogli.

4 Creare le entità sorgente del cartiglio nel layout dello spazio carta:
  - Creare i bordi, linee e testi fissi.

- Crea attributi con campi che fanno riferimento a un gruppo di fogli, sottoinsieme e proprietà del foglio.

Vedere il comando GRUPPOFOGLI per saperne di più circa i gruppi di fogli predefiniti e personalizzati, proprietà di sottoinsiemi e del foglio.

Proprietà gruppo di fogli personalizzato:

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

Proprietà foglio e proprietà personalizzate foglio:

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 Creare un blocco dalle entità di origine del cartiglio.

6 Copia il blocco in un disegno in una delle tue cartelle di blocchi.

7 Eliminare le entità di origine del cartiglio.

Il blocco title può essere inserito negli altri fogli del gruppo di fogli e in qualsiasi altro gruppo di fogli in cui esistono le stesse proprietà personalizzate del gruppo di fogli e del foglio. Tutti i campi che fanno riferimento al gruppo di fogli, al sottoinsieme o alle proprietà del foglio predefinito o personalizzato vengono compilati automaticamente. Se nessun valore è disponibile, un campo visualizza dei trattini (----).

12.9 Lavorare con gli attributi

Un attributo è un'entità particolare che è possibile salvare come parte di una definizione di blocco. Gli attributi consistono in un testo basato su dati. E' possibile utilizzare attributi per indicare elementi come codici e prezzi. Gli attributi hanno valori sia fissi che variabili. Quando si inserisce un blocco che contiene degli attributi, il programma aggiunge i valori fissi al disegno assieme al blocco e si verrà guidati a fornire ogni possibile valore variabile.

Dopo aver inserito i blocchi che contengono attributi, è possibile estrarre le informazioni degli attributi in un file separato e quindi utilizzare tali informazioni in un foglio di calcolo o in un database per produrre un elenco di parti od una distinta dei materiali. È inoltre possibile utilizzare le informazioni degli attributi per tenere conto del numero di volte che un particolare blocco è inserito in un disegno.

Gli attributi possono essere visibili oppure nascosti. Gli attributi nascosti non vengono né visualizzati né stampati, ma le informazioni sono sempre conservate nel disegno e possono essere esportare in un file esterno.

Comando DEFATT - definisce gli attributi tramite una finestra di dialogo.

Comando -DEFATT - definisce gli attributi tramite la barra dei comandi.

Comando VISATT - consente di attivare/disattivare la visualizzazione del testo dell'attributo.

Comando EDITATT - modifica i valori e le proprietà degli attributi.

Comando -EDITATT - Estrae attributi dai disegni e li memorizza in file di dati tramite la barra dei comandi.

Comando ATTREDEF - ridefinisce un blocco e aggiorna gli attributi associati.

Comando SINCATT - sincronizza le definizioni degli attributi in tutti i riferimenti di blocco di una definizione di blocco specificata.

Il comando GESTATTBL (abbreviazione di "Gestione Attributo Blocco") modifica tutti gli aspetti degli attributi in una definizione di blocco, quindi facoltativamente applica le modifiche a tutti i blocchi con lo stesso nome nel disegno.

Comando MODIFATTBL - modifica i valori di attributo e la maggior parte delle proprietà.

12.9.1 Procedura generale per lavorare con gli attributi

1 Definire gli attributi.

2 Creare un blocco.

E' possibile allegare gli attributi ad un blocco. Includere gli attributi quando il programma chiede di selezionare le entità da includere nella definizione di blocco. Se un blocco contiene degli attributi, il programma richiede di immettere il loro valore ogni volta che si inserisce il blocco, in modo che sia possibile specificare valori differenti degli attributi ogni volta che si inserisce il blocco nel disegno.

1 Inserire il blocco in un disegno.

2 (opzionale) Modificare gli attributi in un blocco.

3 Estrarre le informazioni degli attributi

12.10 Editare definizioni di attributo

Le proprietà degli attributi possono essere modificate nel pannello **Proprietà**.

Il comando EDITATT modifica i valori e le proprietà degli attributi.

Il comando GESTATTBL (abbreviazione di" block attribute manager") modifica tutti gli aspetti degli attributi in una definizione di blocco, quindi facoltativamente applica le modifiche a tutti i blocchi con lo stesso nome nel disegno.

Nota: Gli attributi in una definizione di blocco possono essere modificati nel riquadro **Proprietà** quando si utilizzano i comandi MODIFBLOC per modificare il blocco.

12.11 Editare attributi blocco

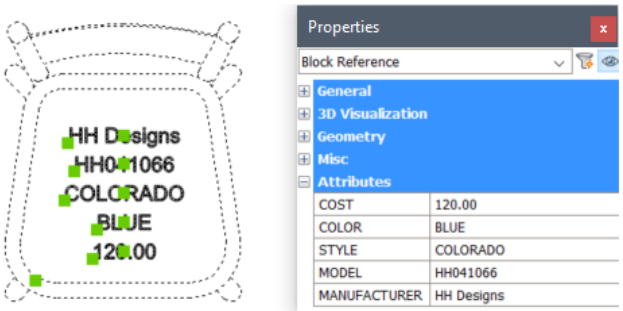
Gli attributi del blocco possono essere modificati nel pannello **Proprietà**.

1 Selezionare un blocco contenente attributi.

1 Il valore corrente degli attributi viene visualizzato nel pannello **Proprietà**.

2 Selezionare l'attributo da modificare.

3 Il campo delle impostazioni dell'attributo selezionato è attivo.



4 Digitare un nuovo valore, quindi premere Invio o selezionare un altro attributo.

12.12 Inserimento di blocchi con attributi

Quando si posiziona un blocco che contiene attributi, verrà richiesto di definire gli attributi nella barra dei comandi o tramite una finestra di dialogo, a seconda del valore della variabile di sistema ATTDIA:

- ON: gli attributi vengono visualizzati in una finestra di dialogo.
- OFF: viene richiesto nella barra dei comandi di definire gli attributi.

12.12.1 Inserire un blocco con attributi

1 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Inserisci Blocco** ( ) sulla barra degli strumenti **Inserisci** o sul pannello **Inserisci/Blocchi** della barra multifunzione.
- Scegliere **Inserisci Blocco...** nel menu **Inserisci**.
- Avviare il comando INSER.

Viene visualizzata la finestra di dialogo **Inserisci Blocco**.

2 Selezionare il blocco, quindi cliccare sul pulsante **OK**.

Viene richiesto: Punto di inserimento del blocco o [Scala/X scala/Y scala/Z scala/Ruota/blocchi Multipli]:

3 Specificare un punto.

4 Tutte le entità blocco vengono posizionate nel disegno, ad eccezione degli attributi.

5 A seconda del valore della variabile di sistema ATTDIA:
   - **Viene richiesto**: <Richiesta Attributo 1> <Valore Predefinto 1>:
   - Viene visualizzata la finestra di dialogo **Editor Attributo**.

6 Eseguire una delle seguenti operazioni:
   - Nella barra dei comandi, per ogni attributo:
   - Premere Invio oppure cliccare il pulsante destro del mouse per accettare il testo predefinito.
   - Digitare un nuovo testo nella barra dei comandi, quindi premere Invio e definire l'attributo successivo.
   - Nella finestra di dialogo **Editor Attributo**:
   - Selezionare un attributo nell'elenco.
   - Il valore corrente viene visualizzato nel campo **Valore**.
   - Digitare un nuovo valore nel campo **Valore**.
   - Premere **OK** quando tutti gli attributi sono stati definiti correttamente.

12.13 Estrarre le informazioni degli attributi

Il comando ESTRATT copia i dati dagli attributi in un file di testo tramite una finestra di dialogo.

Il comando -ESTRATT copia i dati dagli attributi in un file di testo tramite la barra dei comandi.

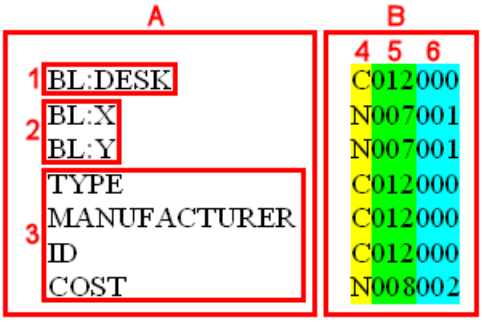
Il file di testo può essere successivamente importato in un database o in un foglio di calcolo.

E' possibile salvare il file in uno dei seguenti formati:

- **Formato Delimitato da Virgole** (CDF): Contiene una linea per ogni istanza di un blocco, con campi attributo individuali separati da virgole. I campi di testo vengono inclusi con singoli punti interrogativi. E' necessario specificare un file modello quando si estrae verso un file CDF.
- **Formato Delimitato da Spazi** (SDF): Contiene una linea per ogni istanza di un blocco. Ogni campo attributo possiede una lunghezza definita, non ci sono caratteri separatori o delimitatori di riga. E' necessario specificare un file modello quando si estrae verso un file SDF.
- **Formato di Interscambio Disegni** (DXF): Crea un sottoinsieme di un file DXF standard (un file *. Dxx) contenente tutte le informazioni di ogni blocco, tra cui i punti di inserimento, angoli di rotazione, e valori di attributo. Nessun file di modello è necessario.

Prima di estrarre gli attributi verso file CDF o SDF, è necessario creare un file modello. Il file modello è un file di testo ASCII che specifica i campi degli attributi che devono essere scritti nel file di esportazione. Ogni linea del file modello, specifica un attributo. BricsCAD riconosce 15 campi differenti, che contengono elementi come il nome del blocco, le coordinate X, Y e Z del suo punto di inserimento, il layer sul quale viene inserito, ecc. E' possibile includere qualsiasi di questi campi. Il file modello deve includere almeno un nome di attributo.

Ogni linea nel file modello deve iniziare con il nome del campo. Il nome del blocco ed i valori del punto di inserimento devono iniziare con **BL:**. Il successivo carattere deve essere o una C (ad indicare un campo di testo) od una N (ad indicare un campo numerico). Questo carattere è quindi seguito da tre caratteri che indicano la larghezza del campo (in caratteri). I tre caratteri finali indicano il numero di posizioni decimali (per i campi numerici). Nel caso dei campi di testo, gli ultimi tre caratteri devono essere zero (000). Un tipico file modello è simile a quello illustrato qui di seguito:



A. Nome campo

B. Formato campo

1. Nome del blocco (deve essere preceduto da **BL:**)
2. Campi delle coordinate (devono essere preceduti da **BL:**)
3. Etichette di Attributo
4. Indica il tipo di dati del campo: **C** per i campi di testo, **N** per i campi numerici
5. Lunghezza Campo (numero di caratteri)
6. Numero di cifre decimali per i campi numerici, **000** per i campi di testo

12.13.1 Estrarre le informazioni degli attributi

1 Eseguire una delle seguenti operazioni:
   - Cliccare lo strumento **Estrai Attributi** nella barra degli strumenti **Strumenti**.
   - Scegliere **Estrai Attributi** nel menu **Strumenti**.
   - Avviare il comando ESTRATT.

   Viene visualizzata la finestra di dialogo **Estrai Attributi**.

2 Cliccare sul pulsante **Seleziona**.

   La finestra di dialogo **Estrai Attributi** viene chiusa temporaneamente.

3 Selezionare i blocchi, quindi premere Invio o cliccare il pulsante destro del mouse.

4 Viene riaperta la finestra di dialogo **Estrai Attributi**.

5 Selezionare il formato del file di testo: **DXF** (formato di interscambio disegni), **CDF** (formato delimitato da virgole) **o SDF** (formato delimitato da spazi).

6 Selezionare il **File Modello**.

7 Specificare il nome del **File di Output**.

8 Cliccare sul pulsante **Estrai**.

Viene richiesto: <numero> record estratti.

12.14 Gestire gli Xrifs

Il comando EXPXREFS apre la sezione **Riferimenti Esterni** della finestra di dialogo **Esplora Disegno**.

Il comando XRIF apre il pannello **Riferimenti**.

La finestra di dialogo **Esplora Disegno - Riferimenti Esterni** consente di:

- Attaccare un disegno come riferimento esterno ( ).
- Staccare un riferimento esterno ( ).
- Ricaricare un riferimento esterno ( ).
- Scaricare un riferimento esterno ( ).
- Collegare un riferimento esterno ( ).
- Inserire un riferimento esterno ( ).
- Modificare il tipo di collegamento: **Attaccato** o **Sovrapposto**.
- Modificare il **Percorso di Salvataggio** di un Xrif.
- Modificare il **Percorso Trovato** di una sovrapposizione.

12.14.1 Staccare i riferimenti esterni

1 Selezionare gli Xrif che si desidera staccare.

1 Tenere premuto il tasto Ctrl per selezionare più Xrif o per deselezionare un Xrif selezionato.

2 Cliccare sul pulsante dello strumento **Stacca Xrif** ( ) sulla barra degli strumenti **Dettagli**.

Gli XRif selezionati vengono staccati. Non viene richiesto di confermare il distacco.

12.14.2 Scaricare i riferimenti esterni

Eseguire una delle seguenti operazioni:

- Cliccare sulla colonna **Carica** di un Xrif caricato.
- Selezionare gli Xrif, quindi fare clic sul pulsante **Scarica Xrif** ( ) della barra degli strumenti **Dettagli** o cliccare sulla colonna **Carica** di uno degli Xrif selezionati.

12.14.3 Ricaricare i riferimenti esterni

Eseguire una delle seguenti operazioni:

- Cliccare sulla colonna **Carica** di un Xrif scaricato.
- Selezionare gli Xrif, quindi fare clic sul pulsante **Ricarica Xrif** ( ) della barra degli strumenti **Dettagli** o cliccare sulla colonna **Carica** di uno degli Xrif selezionati.

12.15 Attaccare un riferimento esterno

Il comando XATTACCA attacca un disegno come riferimento esterno (Xrif). L'Xrif attaccato appare come un blocco nel disegno corrente. Come i blocchi, gli Xrif collegati possono essere annidati. Il disegno attaccato rappresenta la versione più recente salvata.

12.15.1 Attaccare un riferimento esterno

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Attaca Xrif...** ( ) sulla barra degli strumenti **Inserisci** o sul pannello della barra multifunzione **Inserisci/Riferimenti**.
- Scegliere **Attacca Xrif** nel menu **Inserisci**.
- Cliccare sul pulsante dello strumento **Attacca Xref** ( ) nella barra degli strumenti **Dettagli** nella finestra di dialogo **Esplora Disegno/Xrifs**.
- Cliccare sul pulsante dello strumento **Attacca Dwg...** ( ) sul pannello **Riferimenti**.
- Avviare il comando XATTACCA.

Viene visualizzata la finestra di dialogo **Seleziona il file di riferimento**.

2 Selezionare il disegno da attaccare, quindi premere il pulsante **Apri**.

Viene visualizzata la finestra di dialogo **Attacca Riferimento esterno**.

3 Il percorso completo del disegno selezionato viene visualizzato nel campo **Percorso**.

(opzionale) Selezionare il **Tipo di percorso**: **Completo**, **Relativo** o **Nessun percorso**.

4 Scegliere il **Tipo di Riferimento Esterno**:

- **Attaccato**: se il disegno corrente è attaccato come Xrif in un altro disegno, l'attaccamento diventa un Xrif nidificato.
- **Sovrapposto**: una sovrapposizione viene ignorata quando il disegno a cui è attaccato viene successivamente attaccato come Xrif in un altro disegno.

5 Scegliere se specificare il **Punto di inserimento** sullo schermo o nei campi **X**, **Y**, **Z.**

6 Scegliere se specificare la **Scala** sullo schermo o nei campi **X**, **Y**, **Z**.

Selezionare l'opzione **Scala Uniforme** per specificare la scala nel campo **X**.

7 Scegliere se specificare l'angolo di **Rotazione** a schermo o nel campo **Angolo**.

8 Cliccare sul pulsante **OK** per collegare l'Xrif.

A seconda delle opzioni di cui ai punti 5, 6 e 7, viene richiesto di specificare il punto di inserimento, la scala e/o l'angolo di rotazione.

Note:

- La variabile di sistema XDWGFADECTL (controllo della dissolvenza del database Xrif) imposta il livello di dissolvenza per i riferimenti esterni.
- Il punto di origine di un riferimento esterno è controllato dalla variabile di sistema INSBASE (inserimento punto base). Utilizzare il comando BASE per definire la variabile di sistema INSBASE nel disegno del riferimento esterno.

- Il campo **Unità** in **Unità Blocco** è impostato dalla variabile di sistema INSUNIT (Unità di inserimento) nel riferimento esterno. Il campo **Fattore** esprime la relazione tra il valore della variabile di sistema INSUNIT nell'Xrif e il disegno padre.

12.16 Lavorare con i riferimenti esterni

E' possibile collegare interi disegni al disegno corrente come riferimenti esterni. A differenza dell'inserimento di un disegno come blocco, nel quale tutte le entità dal disegno separato vengono aggiunte nel disegno corrente, i riferimenti esterni inseriscono un collegamento al file esterno. Le entità nel riferimento esterno vengono visualizzate nel disegno corrente, ma le entità stesse non vengono aggiunte al disegno.Le entità nel riferimento esterno appaiono nel disegno corrente, ma le entità stesse non vengono aggiunte al disegno. Quindi, attaccare un riferimento esterno non aumenta in modo significativo le dimensioni del file di disegno corrente.

I riferimenti esterni forniscono capacità aggiuntive non disponibili quando si inserisce un disegno come blocco. Quando si inserisce un disegno come blocco, le entità vengono salvate nel disegno. Qualsiasi modifica fatta sul disegno originale non viene riportata nel disegno nel quale è stato inserito. Quando si attacca un riferimento esterno, invece, ogni cambiamento che viene eseguito sul disegno originale viene riportato nel disegno che lo richiama. Questi cambiamenti appaiono automaticamente ogni volta che si apre il disegno contenente il riferimento esterno. Se si è a conoscenza del fatto che il disegno originale è stato modificato, è possibile ricaricare il riferimento esterno in qualsiasi momento durante il lavoro.

I riferimenti esterni sono utili per assemblare i disegni master a partire dai vari disegni componenti. Utilizzare i riferimenti esterni per coordinare il proprio lavoro con quello degli altri in un gruppo di lavoro. I riferimenti esterni aiutano a ridurre le dimensioni del file di disegno ed assicurare che si stia sempre lavorando con la versione più recente di un disegno.

Il comando XRIF attacca i file DWG al disegno corrente tramite l'**Esplora Disegno**.

Il comando -XRIF attacca i file DWG al disegno corrente attraverso la barra dei comandi.

Il comando XATTACCA attacca i disegni con riferimenti esterni attraverso una finestra di dialogo.

Il comando XRITAGLIA ritaglia i disegni dei riferimenti esterni e regola i piani di taglio anteriore e posteriore.

La variabile di sistema XDWGFADECTL controlla la dissolvenza dei riferimenti esterni. Sono accettati valori compresi tra 0 (nessuna dissolvenza) e 90. Il fattore di dissolvenza influisce solo sulla visualizzazione, non sulla stampa.

La variabile di sistema XNOTIFYTIME definisce la frequenza di verifica delle risorse dei riferimento esterni modificati: Xrif, immagini e sottoposti PDF se XREFNOTIFY, IMAGENOTIFY e/o PDFNOTIFY sono **On**.

La variabile di sistema XREFNOTIFY controlla se vengono emessi gli avvisi per gli Xrif mancanti o modificati.

12.17 Editare blocchi e riferimenti esterni

Il comando MODRIF (abbreviazione di "modifica riferimenti") modifica i riferimenti a blocchi e i riferimenti esterni (Xrif) tramite una finestra di dialogo. Ciò che fa MODRIF viene denominato **modifica del riferimento**

sul posto. Il comando funziona su una porzione del riferimento o sull'intero disegno o su tutte le entità nel blocco.

Il comando -MODRIF modifica i riferimenti di blocco ed i riferimenti esterni attraverso la barra dei comandi.

Il comando GRUPPORIF aggiunge e rimuove entità dal riferimento in corso di modifica.

Il comando CHIUDIRIF chiude l'editor riferimenti.

Il comando MODIFBLOC modifica le entità che compongono i blocchi in una sessione di disegno temporanea denominata **Editor Blocchi**, che utilizza il sistema di coordinate locale del blocco.

Il comando -MODIFBLOC richiede un nome di blocco nella barra dei comandi, quindi apre l'ambiente **Editor Blocchi**.

La variabile di sistema XFADECTL imposta il livello di dissolvenza per le entità che non sono abilitate nella sessione di modifica corrente. Sono accettati valori compresi tra 0 (nessuna dissolvenza) e 90.

12.17.1 Modificare gli Xrifs

1 Eseguire una delle seguenti operazioni:
   - Doppio clic sull'Xrif.
   - Cliccare sullo strumento **ModRif** ( ) sulla barra degli strumenti **Mod Rif**.
   - Avviare il comando MODRIF.

   Viene richiesto: Seleziona riferimento:

   Cliccare sull'Xrif o sul blocco.

   Viene visualizzata la finestra di dialogo **Edita Riferimento**.

   La finestra di dialogo consente di identificare visivamente il riferimento da modificare e controllare la modalità di selezione del riferimento.
   - **Percorso**: posizione del riferimento selezionato, se il riferimento è un blocco, non viene visualizzato alcun percorso.
   - **Nome Riferimento**: Il nome del riferimento attualmente selezionato e tutti i riferimenti che sono nidificati all'interno del riferimento selezionato.
   - **Anteprima**: visualizza l'anteprima del riferimento. Si prega di notare che l'immagine dell'anteprima non viene aggiornata quando le modifiche vengono nuovamente salvate al riferimento.

2 (opzionale) Se nell'albero **Nome Riferimento** sono visualizzati più riferimenti nidificati, selezionare un riferimento da modificare.

   Cliccare sul pulsante espandi/collassa per mostrare/nascondere i riferimenti nidificati.

   E' possibile modificare solo un riferimento alla volta.

3 Scegliere una delle seguenti opzioni su **Seleziona entità nidificate...**:
   - **Automaticamente**: tutte le entità e blocchi annidati nel riferimento selezionato vengono inclusi nella sessione di modifica del riferimento.
   - **Richiedi**: dopo aver chiuso la finestra di dialogo **Edita Riferimento**, viene chiesto di selezionare le entità ed i blocchi annidati nel riferimento che si desidera modificare. Solo questi oggetti

saranno temporaneamente estratti e messi a disposizione per la modifica nel contesto del disegno corrente.

Questo insieme temporaneo di entità può essere modificato e poi salvato per aggiornare il riferimento che è stato editato. Per distinguere il 'disegno corrente' dall'"insieme di modifica", BricsCAD visualizza tutte le entità del disegno corrente come sbiadite. Il livello di sbiadimento è controllato attraverso la variabile XFADECTL, il cui valore predefinito è impostato a **50**.

4 Controllare le **Impostazioni**:
   - **Abilita nomi univoci per layer e simboli**:
   - Se selezionato, gli oggetti con nome, contenuti negli Xrif verranno rinominati, aggiungendo un prefisso **$#$**.
   - Se non selezionato, i nomi dei layer e degli altri oggetti con nome rimarranno uguali al disegno di riferimento.
   - **Visualizza le definizioni attributo per la modifica**: Controlla se le definizioni attributo dei blocchi vengono estratte e visualizzate durante la modifica dei riferimenti.
   - **Blocca entità non presenti nel gruppo di lavoro**: Le entità del riferimento che si sta modificando che non sono incluse nel gruppo di lavoro temporaneo non possono essere modificate, come anche le entità nel disegno di origine o altri Xrif. Questo comportamento è simile agli oggetti contenuti nei layer bloccati.

5 Cliccare sul pulsante **OK** per iniziare a modificare il gruppo di lavoro.

   La finestra di dialogo **Edita Riferimento** si chiude.

6 (opzionale) Cliccare sul pulsante dello strumento **Aggiungi a ModRif** ( ) per aggiungere entità al gruppo di lavoro.

   Le entità selezionate verranno rimosse dal disegno che le ospita e aggiunte al riferimento da modificare quando il gruppo di lavoro viene salvato.

7 (opzionale) Cliccare sul pulsante dello strumento **Rimuovi da ModRif** ( ) per rimuovere le entità dal gruppo di lavoro.

   Le entità selezionate verranno rimosse dal riferimento in fase di modifica quando il gruppo di lavoro viene salvato. Queste entità vengono aggiunte al disegno che ospita il blocco.

8 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante **ChiudiRif e Salva** ( ) per salvare le modifiche nell'Xrif o nella definizione del blocco di origine.
   - Cliccare sul pulsante **ChiudiRif e Scarta** ( ) per interrompere la sessione di modifica senza salvare. Il disegno di origine o la definizione del blocco non verranno modificati.

12.18 Lavorare con i gruppi

Il comando GRUPPO crea e modifica il nome di gruppi di entità, in una finestra di dialogo.

Il comando -GRUPPO crea e modifica i nomi di gruppi di entità, dalla barra dei comandi.

Lavorare con i gruppi è una tecnica per manipolare una serie di molteplici oggetti come se fossero un singolo oggetto. Un gruppo può anche essere considerato come una 'Selezione con Nome'. E' possibile selezionare un oggetto singolo e l'intero gruppo verrà selezionato come se fosse un blocco, oppure è possibile modificare oggetti individualmente senza influenzare il resto del gruppo e senza esplodere il gruppo stesso. E' possibile utilizzare l'annidamento gruppi se necessario ed una singola entità può essere un membro di diversi gruppi.

Nota: Assicurarsi che il codice bit 1 della variabile di sistema PICKSTYLE sia impostato (valore = 1 o 3) per abilitare la selezione dei gruppi.

12.18.1 Creare un gruppo

1 Eseguire una delle seguenti operazioni:

- Clicccare sul pulsante dello strumento **Gruppo** ( ) sulla barra degli strumenti **Strumenti** o su **Inizio/Gruppi** nel l pannello della barra multifunzione.
- Scegliere **Gruppo** nel menu **Strumenti**.
- Avviare il comando GRUPPO.

Viene visualizzata la finestra di dialogo **Raggruppamento entità**:

2 Eseguire una delle seguenti operazioni:

- Digitare un nome nel campo **Nome**.
- Selezionare l'opzione **Senza nome** per creare un gruppo senza nome.

3 (opzionale) Inserire una descrizione nel campo **Descrizione**.

4 (opzionale) Attivare/ disattivare l'opzione **Selezionabile**.

5 Cliccare sul pulsante **Seleziona entità e crea gruppo**.

La finestra di dialogo **Raggruppamento Entità** si chiude temporaneamente per consentire la selezione delle entità.

6 Selezionare le entità che si desiderano includere nel nuovo gruppo.

7 Cliccare il pulsante destro del mouse per concludere la selezione delle entità.

8 La finestra di dialogo **Raggruppamento Entità** si riapre.

Il gruppo appena creato viene aggiunto.

9 Cliccare su **OK** per chiudere la finestra di dialogo **Raggruppamento Entità**.

12.18.2 Modificare un gruppo

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Gruppo** ( ) sulla barra degli strumenti **Strumenti** o su **Inizio/Gruppi** nel pannello della barra multifunzione.
- Scegliere **Gruppo** nel menu **Strumenti**.
- Avviare il comando GRUPPO.

Viene visualizzata la finestra di dialogo **Raggruppamento Entità**.

2 Selezionare il gruppo che si desidera modificare.

3 (opzionale) Doppio clic nel campo **Nome** per modificare il nome del gruppo.

4 (opzionale) Doppio clic nel campo **Descrizione** per modificare la descrizione del gruppo.

5 (opzionale) Cliccare nella colonna **Selezionabile** per commutare l'opzione **Selezionabile** da on a off e viceversa.

6 (opzionale) Cliccare sul pulsante **Aggiungi entità al Gruppo** per aggiungere nuove entità al gruppo.

La finestra di dialogo **Raggruppamento Entità** si chiude temporaneamente per consentire di selezionare le nuove entità di gruppo.

7 (opzionale) Cliccare sul pulsante **Rimuovi entità dal Gruppo** per rimuovere le entità dal gruppo.

La finestra di dialogo **Raggruppamento Entità** si chiude temporaneamente per consentire la selezione delle entità.

8 Cliccare su **OK** per chiudere la finestra di dialogo **Raggruppamento Entità**.

12.18.3 Esplodere un gruppo

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Gruppo** ( ) sulla barra degli strumenti **Strumenti** o su **Inizio/Gruppi** nel pannello della barra multifunzione.
- Scegliere **Gruppo** nel menu **Strumenti**.
- Avviare il comando GRUPPO.

Viene visualizzata la finestra di dialogo **Raggruppamento Entità**.

2 Selezionare il gruppo o i gruppi che si desidera cancellare.

3 Cliccare sul pulsante **Esplodi gruppi selezionati**.

I gruppi selezionati vengono rimossi.

Nota: Quando si esplode un gruppo, il gruppo viene eliminato, ma le entità rimangono nel disegno.

12.18.4 Cambiare l'ordine delle entità

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Gruppo** ( ) sulla barra degli strumenti **Strumenti** o su **Inizio/Gruppi** nel pannello della barra multifunzione.
- Scegliere **Gruppo** nel menu **Strumenti**.
- Avviare il comando GRUPPO.

Viene visualizzata la finestra di dialogo **Raggruppamento Entità**.

2 Cliccare sul pulsante **Riordina entità**.

Viene visualizzata la finestra di dialogo **Ordina Gruppo**:

3 (opzionale) Cliccare sul pulsante **Evidenzia** per visualizzare la posizione corrente di ciascuna entità.

Viene visualizzata la finestra di dialogo **Raggruppamento oggetti**.

4 (opzionale) Selezionare **Inverti Ordine** per invertire l'ordine di tutte le entità nel gruppo.

5 (opzionale) Per cambiare l'ordine di una singola entità eseguire una delle seguenti operazioni:
   - Nel campo **Rimuovi da posizione**, digitare la posizione corrente dell'entità che si vuole spostare.
   - Nel campo **Inserire nuovo numero posizione per l'entità**, digitare la nuova posizione.

6 (opzionale) Digitare il numero di oggetti che si desidera riordinare nel campo N**umero di entità**.

7 Cliccare sul pulsante **Riordina**.

Un messaggio conferma che il gruppo è stato riordinato.

12.19 Lavorare con i sottoposti

I file PDF possono essere attaccati come sfondo all'interno di un file di disegno. In modo analogo alle immagini raster ed ai riferimenti esterni (Xrif), un sottoposto PDF non fa parte del disegno, ma è ad esso collegato. Il percorso del sottoposto viene salvato nel disegno principale. È possibile modificare il percorso per assicurarsi che il file PDF venga trovato. Per impostazione predefinita i sottoposti vengono ricercati nella cartella del disegno principale. Se il sottoposto PDF non viene trovato, viene cercato nella cartella del percorso salvato. Se il sottoposto PDF non si trova neanche qui, BricsCAD riporta il messaggio 'Riferimento mancante o non valido'.

Il comando REGOLAPDF permette di regolare le impostazioni di sfumatura, contrasto e monocromatico di un sottoposto PDF nella barra dei comandi.

I comandi ALLEGAPDFe -ALLEGAPDF inseriscono un file PDF come sottoposto nel disegno corrente.

Il comando LAYERPDF consente di controllare la visualizzazione dei layer in un sottoposto PDF.

Il comando RITAGLIAPDF ritaglia la visualizzazione di un PDF. È possibile scegliere tra una poligonale o un contorno rettangolare. Ogni istanza dello stesso sottoposto PDF file può avere un diverso contorno.

La variabile di sistema PDFOSNAP permette lo snap alla geometria del sottoposto PDF se la variabile di sistema è **On**.

La variabile di sistema PDFNOTIFY controlla se vengono emessi gli avvisi per i PDF mancanti o modificati.

La variabile di sistema XNOTIFYTIME definisce la frequenza di verifica delle risorse dei riferimento esterni modificati: Xrif, immagini e sottoposti PDF se XREFNOTIFY, IMAGENOTIFY e/o PDFNOTIFY sono **On**.

Se il valore della variabile di sistema FRAME = 3, la variabile di sistema PDFFRAME controlla la visibilità del contorno dei PDF. Le opzioni sono:

- 0: Nascondere contorni PDF.
- 1: Visualizzare e stampare i contorni PDF.
- 2: Visualizzare ma non stampare i contorni PDF.

12.19.1 Controllare la memorizzazione nella cache del sottoposto PDF

Una cache persistente di immagini multi-risoluzione viene utilizzata per visualizzare i sottoposti PDF attaccati, consentendo operazioni (molto) veloci di zoom e pan. La risoluzione più alta nella cache è 5000 x 5000 pixel. Tuttavia, quando si esegue uno zoom molto vicino, la visualizzazione del sottoposto PDF diventerà pixelata. È quindi possibile utilizzare un modus operandi ibrido che consente di passare

alla generazione in tempo reale della visualizzazione nitida del PDF quando si esegue uno zoom molto vicino. La generazione iniziale della cache delle immagini può richiedere alcuni secondi, da allora in poi l'elaborazione diventa (molto) veloce e rimane così nelle sessioni successive.

La variabile di sistema PDFCACHE controlla come viene applicata la memorizzazione dei PDF nella cache.

- 0 = non utilizza la cache PDF, utilizza sempre la generazione in tempo reale dell'immagine del sottoposto PDF. (questa modalità era l'impostazione predefinita nelle versioni precedenti di BricsCAD).
- 1 = usa la cache PDF, passa alla generazione in tempo reale quando lo zoom è molto vicino.
- 2 = usa sempre la cache PDF (predefinito).

12.19.2 Attaccare un sottoposto PDF

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Attacca PDF** ( ) sulla barra degli strumenti **Inserisci** o su **Inserisci / PDF** della barra multifunzione.
- Cliccare sul pulsante **Nuovo** ( ) della barra **Dettagli** della finestra di dialogo **Esplora Disegno - Sottoposti PDF**.
- Digitare **ALLEGAPDF** nella barra dei comandi, quindi premere Invio.

Verrà aperta la finestra di dialogo **Selezionare File Sottoposto PDF**.

2 Selezionare il file, quindi doppio clic sul file o cliccare sul pulsante **Apri** nella finestra di dialogo **Selezionare File Sottoposto PDF**.

Viene visualizzata la finestra di dialogo **Attacca Sottoposto Pdf**.

3 Nel caso di un PDF multipagina, cliccare sulla freccia verso il basso nel campo **Nome**, quindi selezionare la pagina del PDF che si desidera caricare.

4 Cliccare sull'opzione **Percorso**, quindi selezionare:

- **Percorso completo**: durante il caricamento del file, il sottoposto PDF verrà ricercato nel percorso completo. Se il sottoposto PDF non viene trovato nel percorso completo memorizzato, verrà ricercato nella cartella del disegno. Se il file sottoposto non si trova neanche qui, verrà visualizzata nel punto di inserimento della sottoposto PDF la dicitura "Riferimento mancante o non valido".
- **Percorso relativo**: durante il caricamento del file il sottoposto PDF verrà ricercato nel percorso relativo rispetto la cartella del disegno principale.. Se il sottoposto PDF non viene trovato nel percorso relativo memorizzato, verrà ricercato nella cartella del disegno. Se il file sottoposto non si trova neanche qui, verrà visualizzata nel punto di inserimento della sottoposto PDF la dicitura "Riferimento mancante o non valido".
- **Nessun percorso**: durante il caricamento del file il sottoposto PDF verrà ricercato esclusivamente nella cartella del disegno principale. Se il sottoposto non viene trovato, verrà visualizzata nel punto di inserimento della sottoposto PDF la dicitura "Riferimento mancante o non valido".

5 Scegliere se si desidera specificare il **Punto di Inserimento**, la **Scala** e l'Angolo di **Rotazione** a schermo

oppure no.

6 Cliccare sul pulsante **OK**.

La finestra di dialogo **Attacca Sottoposto Pdf** si chiude.

7 A seconda delle opzioni di inserimento scelte nel passaggio precedente, **viene richiesto** di specificare il **Punto di inserimento**, la **Scala** e/o la **Rotazione**.

12.19.3 Inserire un sottoposto PDF

Se si desidera inserire una seconda istanza di un PDF che è già collegato o un'altra pagina di un sottoposto effettuare le seguenti operazioni:

1 Avviare il comando EXPPDFS.

1 Viene visualizzata la finestra di dialogo **Esplora Disegno - Sottoposti PDF**.

2 Selezionare il sottoposto PDF.

3 Eseguire una delle seguenti operazioni:

- Cliccare il pulsante destro del mouse e scegliere **Inserisci** nel menu contestuale.
- Scegliere **Inserisci** nel menu **Modifica** della finestra di dialogo **Esplora Disegno - Sottoposti PDF**.

La finestra di dialogo **Esplora Disegno - Sottoposti PDF** verrà chiusa.

4 Continuare con passaggi da 3 a 7 della procedura precedente.

12.19.4 Gestire la visualizzazione di un sottoposto PDF

1 Cliccare sulla cornice del sottoposto PDF.

1 Viene evidenziata la cornice del PDF selezionato.

Le proprietà della sottoposto vengono visualizzate nel pannello **Proprietà**.

2 In **Varie**, cliccare su **Visualizza sottoposto** nel pannello **Proprietà**, quindi selezionare **Sì** o **No**.

12.19.5 Impostare la visualizzazione dei layer di un sottoposto PDF

1 Avvia il comando LAYERPDF.

1 **Viene richiesto**: Selezionare sottoposto PDF.

2 Cliccare sulla cornice del sottoposto PDF.

3 Viene evidenziata la cornice del PDF selezionato.

Viene visualizzata la finestra di dialogo **Layers Sottoposto**.

4 L'icona nella colonna **On** indica lo stato corrente dei layer; **On** ( ) o **Off** ( ).

5 Cliccare sull'icona nella colonna **On** per attivare o disattivare la visualizzazione di un layer.

6 (opzionale) Cliccare sul campo **Cerca layer**, quindi digitare il nome di un layer per cercarlo.

Usare i caratteri jolly (? o *) per limitare il numero di layer nell'elenco.

7 (opzionale) Cliccare sul pulsante **Nome del riferimento** per selezionare un altro sottoposto PDF.

8 Cliccare il pulsante **OK** per terminare.

12.19.6 Ritagliare un sottoposto PDF

1 Avviare il comando RITAGLIAPDF.

1 **Viene richiesto**: Selezionare sottoposto PDF:

2 Cliccare sulla cornice del sottoposto PDF.

3 Viene evidenziata la cornice del PDF selezionato.

Viene richiesto: Inserisci l'opzione di ritaglio PDF [ON/OFF/Nuovo] <Nuovo>:

4 Premere Invio per accettare l'opzione predefinita **Nuovo**.

Viene richiesto: Immettere il tipo di ritaglio PDF [Poligonale/Rettangolare] <Rettangolare>:

5 Eseguire una delle seguenti operazioni:
- Premere Invio per accettare l'opzione predefinita **Rettangolare** per definire un contorno di ritaglio rettangolare.
- Digitare P, quindi premere Invio per definire un contorno di ritaglio poligonale.

6 Definite il contorno di ritaglio.

7 I punti definiti devono trovarsi all'interno della cornice del sottoposto PDF. Cliccando all'esterno del sottoposto PDF, il punto viene posizionato sul contorno di base del PDF.

12.19.7 Commutare la visualizzazione del ritaglio di un sottoposto PDF

1 Cliccare sulla cornice del sottoposto PDF.

1 Viene evidenziata la cornice del PDF selezionato.

Le proprietà della sottoposto vengono visualizzate nel pannello **Proprietà**.

2 Nel pannello **Proprieta**, sotto **Varie**, cliccare su **Mostra ritaglio**, quindi selezionare **Si** o **No**.

12.19.8 Eliminare un contorno di ritaglio

1 Avviare il comando RITAGLIAPDF.

1 **Viene richiesto**: Selezionare sottoposto PDF.

2 Cliccare sulla cornice del sottoposto PDF.

3 Viene evidenziata la cornice del PDF selezionato.

Viene richiesto: Inserisci l'opzione di ritaglio PDF [ON/OFF/Nuovo] <Nuovo>:

4 Digitare C nella barra dei comandi, quindi premere Invio.

5 Il contorno di ritaglio verrà eliminato e quindi il sottoposto PDF verrà visualizzato completamente.

12.19.9 Regolare le impostazioni di visualizzazione di un sottoposto PDF

Il comando REGOLAPDF consente di regolare le impostazioni **Sfumatura**, **Contrasto** e **Monocromatico** di una sottoposto PDF nella barra dei comandi.

Selezionare il sottoposto PDF, quindi regolare le impostazioni di visualizzazione sotto **Regolazione Sottoposto** ne pannello **Proprietà**.

12.20 Componenti e inserimenti di componenti

Un componente è un gruppo denominato di entità, che può essere inserito in un disegno con il comando BmInsert. I componenti possono essere parametrici: ad es. finestre e porte in BricsCAD BIM e componenti meccanici standard in BricsCAD Mechanical.

Qualsiasi file .dwg può essere utilizzato come componente. Lo stesso componente può essere inserito una o più volte in uno o più altri componenti. Quando il componente viene modificato, tutti gli inserimenti del componente rifletteranno automaticamente queste modifiche. I componenti non possono essere autoreferenziali.

Un inserimento di un componente è un'entità denominata. Il nome predefinito è composto dal nome del componente ed il numero di serie dell'elemento. Internamente un inserimento di un componente è rappresentato come un riferimento ad un blocco anonimo o come riferimento esterno.

Le proprietà dei componenti inseriti, inclusi i loro parametri, possono essere modificate nel pannello **Navigatore Meccanico** e nel pannello **Proprietà**.

12.20.1 Componenti esterni e locali

I componenti possono essere locali o esterni. Un componente locale è salvato nel file .dwg dell'assieme come blocco anonimo. Un componente esterno è definito in un file .dwg separato e rappresentato da un Xrif o una copia locale memorizzata come blocco anonimo. Quando il file .dwg corrispondente viene modificato, tutti i file dei componenti esterni possono essere caricati utilizzando il comando BMUPDATE o il menu contestuale del **Navigatore Meccanico**. Se si utilizzano componenti esterni nell'assieme, ricordare sempre di trasferire tutti i file correlati quando si desidera condividere il modello con i propri colleghi.

Il comando ETRANSMIT crea un unico pacchetto contenente il file di disegno e tutte le sue dipendenze, ad esempio: riferimenti esterni, immagini, file dei font, file di configurazione del plotter, tabelle stili di stampa e file di mappaggio dei font.

È sempre possibile convertire una componente locale in uno esterno e viceversa.

Per modificare un componente locale, è necessario convertirlo in un componente esterno utilizzando il comando BMEXTERNALIZE. In alternativa è possibile utilizzare il comando BMOPENCOPY per aprire una copia di un componente, modificarlo e salvarlo. Quindi utilizzare il comando BMREPLACE per sostituire l'originale con la copia.

È possibile scegliere il tipo di inserimento predefinito per il componente: **Locale** o **Esterno**. Per modificare il tipo di inserimento, selezionare il nodo principale nel **Navigatore Meccanico** e selezionare **Locale** o **Esterno** nella proprietà **Inserisci come**.

A seconda del valore di BMAUTOUPDATE, gli inserimenti di componenti esterni vengono aggiornati automaticamente all'apertura del documento dell'assieme o manualmente con il comando BMUPDATE. È possibile modificare questo valore nella finestra di dialogo **Impostazioni**. Le opzioni sono:

- Aggiornare solo quando si avvia il comando BMUPDATE.
- Aggiornare automaticamente all'apertura del file.

12.20.2 Conversione dei componenti

Il comando BMEXTERNALIZE converte i componenti locali in componenti esterni.

Il comando BMLOCALIZE converte i componenti esterni in componenti locali.

Eseguire una delle seguenti operazioni:

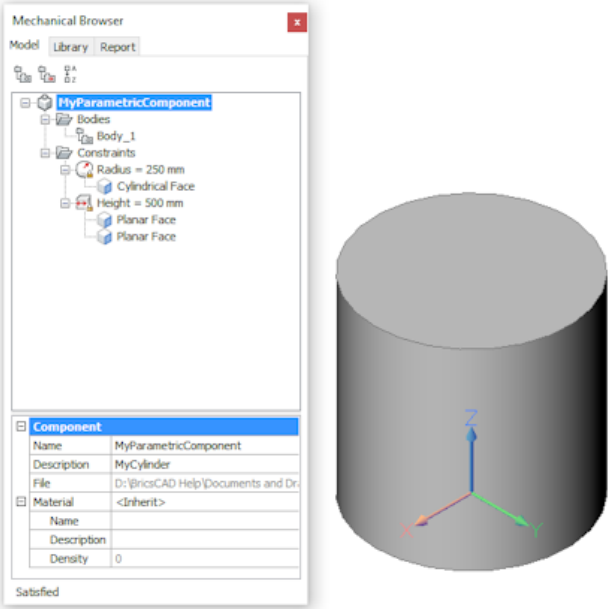
- Cliccare su **Commuta Componente in Esterno** ( ) o **Commuta Componente in Locale** nella scheda **Assemblaggio**. Gli strumenti sono disponibili anche nella barra degli strumenti **Assemblaggio** e nel menu **Assieme**.
- Cliccare il pulsante destro del mouse sull'inserimento del componente nel **Navigatore Meccanico** e selezionare **Commuta in Esterno** o **Commuta in Locale** nel menu contestuale.
- Passare il cursore del mouse sull'inserimento di un componenti in una finestra di disegno e selezionare **Commuta in Esterno** ( ) o **Commuta in Locale** nel gruppo di comando **Modello** nel Quad.

Quando si converte un componente esterno in un componente locale, il file .dwg con la definizione del componente non viene eliminata, ma le modifiche future non influiranno sul tuo assieme.

È sempre possibile distinguere tra inserimenti di componenti locali ( ) ed inserimenti di componenti esterni ( ) dalle loro icone nel **Navigatore Meccanico**.

12.20.3 Componenti Parametrici

È possibile apportare modifiche parametriche ai componenti inseriti a livello di assieme. Un componente è considerato parametrico se contiene almeno un vincolo dimensionale 2D o 3D o una serie associativa con un'espressione.



Note:

- BricsCAD mantiene automaticamente l'associatività tra la definizione di un componente parametrico ed i suoi inserimenti. Quindi si può aprire il componente parametrico per modificare la sua geometria e salvarlo. Per visualizzare il risultato, eseguire il comando BmUpdate per aggiornare la struttura meccanica dell'assieme.
- Se si necessita di tornare al valore predefinito di un parametro di un componente, basta modificare il campo **Espressione** ed eliminare il suo contenuto.

È possibile controllare se un particolare parametro di un componente sarà disponibile per la modifica in un disegno che contiene gli inserimenti di tale componente (a livello di assieme). A tale scopo, selezionare un parametro nel **Navigatore Meccanico** e modificare il valore del campo **Esposto** nella tabella delle proprietà del parametro.

Le opzioni sono:

- **Off**: Nasconde i parametri sull'assieme.
- **On**: Rende il parametro visibile a livello di assieme.
- **Auto**: Nasconde tutti i parametri i cui valori sono controllati dalle espressioni ed espone tutti gli altri parametri.

È possibile associare le unità con un parametro. Se si assegna un valore, direttamente o tramite un'espressione, ad un parametro di un componente a livello di assieme, viene automaticamente convertito dalle unità del disegno dell'assieme alle unità del parametro. I valori senza unità assegnati ai parametri non vengono convertiti.

Per impostazione predefinita tutti i parametri sono privi di unità, eccetto i parametri dei vincoli dimensionali ed i parametri che vengono usati come espressioni per i vincoli dimensionali. BricsCAD assegna automaticamente le unità di disegno a tali parametri.

12.20.4 Modificare i parametri di un inserimento componente

Quando un componente parametrico viene inserito in un modello, è possibile modificarne i parametri nel **Navigatore Meccanico** o nel pannello **Proprietà**.

L'inserimento del componente si aggiorna immediatamente in base al nuovo valore di un parametro. Si noti che questo non modifica la definizione del componente parametrico, che consiste in un file di disegno separato. Di conseguenza, nello stesso modello possono essere presenti più inserimenti dello stesso componente parametrico, con valori diversi per ciascuno dei loro parametri:

Modificare un parametro nel Navigatore Meccanico

1 Espandere il nodo **Parametri** del componente.

2 Selezionare il parametro.

3 Nella sezione **Parametri**, digitare un nuovo valore nel campo **Espressione**.

Modificare un parametro nel pannello delle Proprietà

1 Selezionare il componente nel disegno.

2 Espandere il gruppo di impostazioni **Parametri**.

3 Digitare un nuovo valore per uno dei parametri disponibili.

Note:

- Nel pannello **Proprietà** viene visualizzato il valore corrente. Se questo valore è controllato tramite un'espressione, l'espressione verrà sovrascritta dal valore appena immesso.
- Se vengono selezionati più componenti parametrici, i parametri condivisi possono essere modificati contemporaneamente. I campi di impostazione dei parametri che attualmente hanno valori diversi riportano *Varie*.

12.20.5 Sostituire i componenti

Il comando BMREPLACE sostituisce gli inserimenti dei componenti. È possibile sostituire solo gli inserimenti di livello superiore.

Nota: I valori dei parametri condivisi tra il componente sostitutivo e il componente sostituito vengono copiati automaticamente.

12.20.6 Componenti, blocchi o riferimenti esterni?

I componenti sono basati su blocchi e riferimenti esterni (Xrifs), ma il comportamento dei componenti è diverso. Un componente esterno è collegato a un file, proprio come un Xrif, ma un componente esterno può essere rappresentato da un Xrif (BMINSERT agisce come XATTACCA) o da un blocco anonimo locale (BMINSERT agisce come INSER).

I componenti possono essere costituiti da singole parti o assiemi complessi:

- Parte: composto solo da entità; nessun inserimento di sottocomponenti.
- Assieme: composto da entità e inserimenti di sottocomponenti. Anche se i componenti secondari vengono rimossi, lo stato dell'assieme viene mantenuto.

Quando un componente esterno corrisponde ad una parte e i relativi parametri non vengono modificati, il componente viene inserito come Xrif. Quest'ultimo è elencato nella sezione **Riferimenti Esterni** della finestra di dialogo **Esplora Disegno**. I layer dell'Xrif hanno un prefisso 'nome_file'.

Quando un componente esterno è un assieme, viene inserito come blocco anonimo locale. Tutti i suoi layers vengono uniti ai layers del documento principale e nessun prefisso viene aggiunto al nome del layer.

12.20.7 Componenti, blocchi e riferimenti esterni

Il comando BMMECH converte il disegno corrente in un componente meccanico. Se il disegno è già un componente meccanico BMMECH non fa nulla.

I riferimenti a blocchi vengono convertiti in inserimenti di componenti locali e i riferimenti esterni vengono convertiti in inserimenti di componenti esterni.

Il comando BMUNMECH converte il componente meccanico corrente in un disegno semplice. I componenti meccanici possono essere convertiti in blocchi e riferimenti esterni. Il comando si applica solo a disegni che sono componenti meccanici.

12.20.8 Funzionalità basate su componenti

Una caratteristica è un elemento fisico di una parte, che ha un significato ingegneristico. Esempi di caratteristiche sono fori, nervature, tasche, forme. Se applicato ad un solido 3D, una caratteristica rimuove e/o aggiunge alcuni volumi ad esso. Per creare le tue caratteristiche utilizza la tecnologia di estensione dei componenti di BricsCAD.

Un componente esteso contiene i layer BC_SUBTRACT e/o BC_UNITE. Quando si utilizza il comando BMINSERT per inserire tale componente sulla faccia di un solido 3D, il solido di destinazione viene modificato come segue:

- Tutti i solidi 3D posizionati sul layer BC_SUBTRACT del componente vengono sottratti dal solido 3D di destinazione.
- Tutti i solidi 3D posizionati sul layer BC_UNITE del componente vengono uniti al solido 3D di destinazione.

Tutte le facce create dopo le operazioni booleane formano una caratteristica basata su componenti. Vengono aggiornati quando si sposta l'inserimento del componente o quando si modificano i parametri. Quando l'inserimento viene eliminato, il solido di destinazione viene riaggiustato. Regola i solidi 3D di destinazione con il comando BMLINK.

Alcuni esempi di caratteristiche parametriche basate sui componenti sono incluse con BricsCAD. È possibile utilizzarle come campioni per creare le proprie caratteristiche.

12.20.9 Assegnare un materiale fisico ai componenti

È possibile assegnare un materiale fisico a un componente meccanico. Se un componente meccanico non ha un materiale assegnato, erediterà il materiale dal componente genitore. Se il componente genitore non dispone di un materiale assegnato, verrà utilizzato il materiale del componente più vicino. Di conseguenza, lo stesso materiale può essere assegnato a tutti i componenti che hanno un genitore comune.

1 Selezionare il nodo principale nel **Navigatore Meccanico**.

2 Selezionare la proprietà **Materiale** e premere il pulsante **Sfoglia** (...).

3 Nella finestra di dialogo **Materiali Fisici**, selezionare un materiale dall'elenco **Nel progetto** o **Nel database centrale**.

   I materiali scelti dalla **Database centrale** vengono copiati automaticamente nel database **Nel progetto**.

4 Cliccare sul pulsante **OK** per assegnare il materiale selezionato al componente.

Rimuovere il materiale dal componente:

1 Selezionare il nodo principale nel **Navigatore Meccanico**.

2 Selezionare la proprietà **Materiale** e premere il pulsante **Cancella** (X).

Inserire un componente con un materiale fisico:

Quando si inserisce un componente meccanico in un modello, tutti i suoi materiali vengono copiati nella libreria dei materiali del modello. Se nel modello di destinazione esiste già un materiale con lo stesso nome, verrà utilizzato quest'ultimo. Di conseguenza, vengono utilizzate le proprietà del materiale nel modello di destinazione. Queste sostituiscono le proprietà del materiale dal componente inserito.

Nota: I materiali copiati nella libreria materiali del documento non verranno cancellati dalle operazioni di **Annulla**.

13. Gestire le immagini

13.1 Managing images

The EXPIMAGES command displays the Drawing Explorer – Images dialog box.

The IMAGE command opens the Attachments panel.

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGEATTACH command attaches raster images to the drawing.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The IMAGEQUALITY command toggles the display quality of images between draft and high.

The IMAGENOTIFY system variable controls whether warnings for missing or modified image files are issued.

The XREFNOTIFYTIME system variable defines the frequency to check for modified externally referenced resources: Xrefs, images and pdf attachments if XREFNOTIFY, IMAGENOTIFY and/or PDFNOTIFY is ON.

You can manage images in the **Drawing Explorer - Images** dialog box and in the **Attachments** panel.

Here you can:

- Attach an image ( ) or ( ).
- Load / Unload inserted images.
- Detach images (✕).
- Edit the path to the image.
- See a preview of the attached images (**Drawing Explorer** dialog box only).

To open the **Drawing Explorer - Images** dialog box, do one of the following:

- Choose **Drawing Explorer** > **Images** in the **Tools** menu.
- Select **Images** in the **Open Drawings** sub-window of the **Drawing Explorer** dialog box.
- Launch the EXPIMAGES command.

To open the **Attachments** panel, do one of the following:

- Click the **Image Management** tool button ( ) in the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGE command.

13.1.1 Attaching an image

1 Do one of the following:

- Click the **New** tool button ( ) in the **Drawing Explorer - Images** dialog box.
- Click the **Attach Raster Image** tool button ( ) in the **Insert** toolbar or the **Insert / Images** ribbon panel.

- Launch the IMAGEATTACH command.

The **Select Image File** dialog box appears.

2 Select the image file, then double-click or click the **Open** button.

The **Attach Raster Image** dialog box appears.

3 (option) Check the **Use Geocoding Information** option, then click the **Browse** button to open an appropriate world file to be used with the selected raster image.

If the **Use Geocoding Information** option is checked, the **Insertion Point**, **Scale** and **Rotation** settings fields are not available.

4 Specify an **insertion point** or select the **Specify on-screen** check box.

5 Specify a **scale** or select the **Specify on-screen** check box.

6 Specify a **rotation** angle or select the **Specify on-screen** check box.

7 Click the **OK** button.

8 If none of the **Specify on-screen** options are selected, the image is inserted at the desired insertion point, scale and rotation angle;

else you are prompted to specify the insertion point, scale and/or rotation angle on-screen.

13.1.2 Loading and unloading images

When an image is attached and then inserted in the drawing, you can unload the image to temporarily hide it. Unloaded images are still inserted in the drawing, but they are no longer displayed. If the **Imageframe** property is ON, the frame around the image is still displayed.

To load / unload a single image

Click the **Load** column of the image in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

To load / unload multiple images

1 Hold down the Ctrl key, then select the images in the **Attachments** panel or the **Drawing Explorer – Images** dialog box. Click a selected image to de-select.

2 Click the **Load** column of a selected image.

13.1.3 Detaching images

Since an image is first attached, then inserted in the drawing, you can delete an inserted image, in the drawing without detaching it.

To detach an image:

1 Select the image(s) in the **Attachments** panel or the **Drawing Explorer – Images** dialog box.

Hold down the Ctrl key to select multiple images; click a selected image to de-select.

2 Do one of the following:

- Click the Delete or Detach (✕) tool button.
- Right-click, then choose **Delete** or **Detach** in the context menu.

13.2 Image utilities

The IMAGEADJUST command adjusts the properties of images through the Properties panel.

The IMAGECLIP command clips images.

The IMAGEFRAME command toggles the display of the frame around images; sets the value of the IMAGEFRAME system variable.

The TRANSPARENCY command toggles the transparency of images that have a transparent alpha channel. The alpha channel is a special channel that handles transparency. When an image has an alpha channel on it, you can adjust the image's opacity levels and make bits translucent or totally see-through.

Nota: When the image frames do not appear (IMAGEFRAME = 0), you must use a window selection method to select an image.

13.2.1 Clipping an image

1 Do one of the following:

- Click the **Clip Image** tool button ( ) on the **Image** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGECLIP command.

You are prompted: Select image:

2 Select the image.

3 **You are prompted**: Enter image clipping option [ON/OFf/New boundary] <New boundary>:

4 Press Enter to accept the **New Boundary** default option.

5 **You are prompted**: [Select polyline/Polygonal/Rectangular] <Rectangular>:

Press Enter to accept the **Rectangular** default option.

6 **You are prompted**: Specify first corner.

7 Click a point to define the first corner of the clipping rectangle.

8 The clipping rectangle is dynamically displayed.

You are prompted: Specify opposite corner:

9 Click a point to define the clipping rectangle.

10 The image is clipped.

13.2.2 Modifying the Image Frame setting

Using the Image Frame tool (IMAGEFRAME command):

1 Do one of the following:

- Click the **Image Frame** tool ( ) on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the IMAGEFRAME command.

You are prompted: Enter image frame setting <current value>:

2 Do one of the following:

- Type 0 to hide image frames.
- Type 1 to display and plot image frames.

- Type 2 to display but not print image frames.

Editing the IMAGEFRAME system variable:

1 Open the **Settings** dialog box.

2 Go to **Drawing / Underlays**.

3 Set the FRAME system variable = 3: Use individual settings.

4 Select the IMAGEFRAME system variable.

5 The options are:

- 0: Hide image frames.
- 1: Display and plot image frames.
- 2: Display but do not plot image frames.

13.2.3 Setting the image transparency

1 Do one of the following:

- Click the **Image Transparency** tool button ( ) on the **Images** toolbar or the **Insert / Images** ribbon panel.
- Launch the TRANSPARENCY command.

You are prompted: Select images.

2 Select the images you want to set the **Transparency** property of.

You are prompted: Enter transparency mode [ON/OFF]<current mode>:

3 Do one of the following:

- Press Enter to accept the current mode.
- Choose **On** to switch the **Transparency** property of the selected images on.
- Choose **Off** to switch the **Transparency** property of the selected images off.

Transparency of the camera image: OFF (left) and ON (right)

13.2.4 Setting the display properties of an image

1 Select an image.

2 The properties of the image appear in the **Properties** panel.

3 Go to the **Misc** properties category.

4 The following properties can be set **Yes** or **No**:
   - **Show Image**: shows or hides the image.
   - **Show clipped**: sets the clipping boundary on/off.
   - **Clip inverted**: inverts the clipping boundary.
   - **Background transparency**: sets the transparency on/off.

14. Annotazione

14.1 Dimensions

14.1.1 Introduzione agli strumenti di Quotatura

Gli strumenti di quotatura sono disponibili in:

- Barra degli strumenti **Quotatura**.
- Il menu **Quotatura**.
- Il pannello **Annota/Quotatura** della barra multifunzione.
- Nella scheda **Disegna** del **Quad su Nessuna Selezione**.

Icona	Nome strumento	Comando
	Lineare	DIMLINEARE
	Allineato	DIMALLINEATA
	Arco	ARCOQUOTA
	Raggio	DIMRAGGIO
	Diametro	DIMDIAMETRO
		DIMDISSOCIA
		DIMRIASSOCIA
	Angolare	DIMANGOLO
	LineaBase	DIMLBASE
	Continua	DIMCONTINUA

Icona	Nome strumento	Comando
	Coordinata	DIMCOORDINATA
	Ruota	DIMLINEARE
	Rendi Obliquo	DIMEDITA + O
	Multidirettrice	MULTIDIR
	Modifica Multidirettrice	MODMULTIDIR
	Aggiungi direttrice a multidirettrice	AIMLEADEREDITADD
	Rimuovi direttrice a multidirettrice	AIMLEADEREDITREMOVE
	Tolleranza	Tolleranze
	Marcatura centro	DIMCENTRO
	Edita Testo Quota	DIMEDITA + E
	Ruota Testo Quota	DIMEDITA + R
	Riposiziona Testo Quota	DIMTEDIT
	Ripristina Posizione Testo	DIMEDITA

Icona	Nome strumento	Comando
	Applica Stili di Quota	-DIMSTILE+ A
	Salva Stile di Quota	-DIMSTILE + SA
	Ripristinare lo stile	-DIMSTILE
	Stato Variabili Quota	-DIMSTILE + ST
	Direttrice	DIMDIRETTRICE
	Direttrice rapida	DIRRAPID

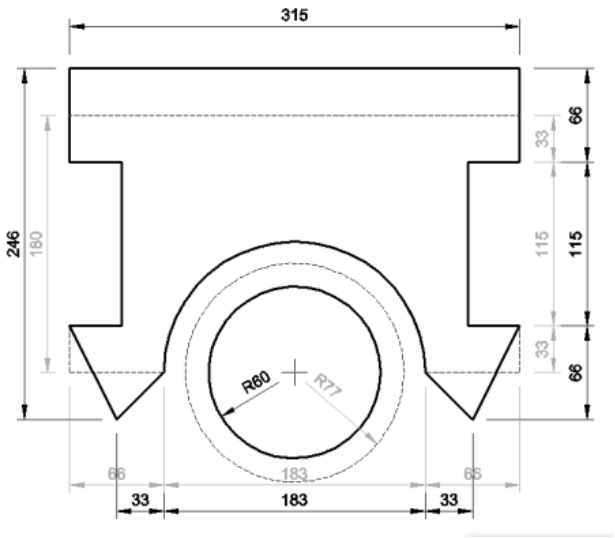
14.1.2 Quote Associative

Le quote associative si aggiornano automaticamente quando l'entità dimensionata viene spostata o modificata.

BricsCAD crea quote associative se il valore della variabile di sistema DIMASSOC è 2 (predefinito).

La proprietà associativa delle quote si applica anche alle quote posizionate nello spazio carta e sono associate alle entità dello spazio modello.

Se necessario, è possibile utilizzare il comando DIMRIGEN per aggiornare tutte le quote associative.



I seguenti snap ad entità possono essere utilizzati per creare quote associative: **punto finale**, **punto medio**, **centro**, **perpendicolare**, **quadrante**, **inserimento**, **nodo**, **intersezione** e **intersezione apparente**.

Il comando DIMDISSOCIA rimuove l'associatività dalle entità quota selezionate.

Il comando DIMRIASSOCIA riassocia o associa le quote alle entità o ai punti delle entità.

La variabile di sistema GENERATEASSOCVIEWS consente la creazione di quote associative per le viste generate dai comandi VISTABASE e VISTASEZIONE e i disegni generati dal comando BIMAGGIORNASEZIONE.

14.1.3 Impostazioni delle Quote

Le impostazioni delle quote vengono salvate in uno stile di quota. Le entità quota vengono create utilizzando le impostazioni nello stile di quota corrente.

Il comando DIMSTILE crea e modifica gli stili di quota, attraverso l'**Esplora Disegno**.

Il comando -DIMSTILE crea e modifica gli stili di quota nella barra dei comandi.

Uno stile di quota è costituito da sei gruppi di impostazioni

- **Linee e Frecce**
- **Testo**
- **Adatta**
- **Unità Primarie**
- **Unità Alternative**
- **Tolleranze**

14.1.3.1 Elecare lo stato attuale delle variabili delle quote

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Stato Variabili Quota** ( ) sulla barra degli strumenti **Quotatura**.
- Scegliere **Stato Variabili Quota** nel menu **Quotatura**.

Un elenco di tutte le variabili della quota verrà visualizzato nella barra dei comandi.

2 Premere il tasto funzione **F2** per aprire la **Cronologia dei comandi BricsCAD**.

3 Premere Invio per continuare l'elenco.

4 Premere il tasto funzione **F2** per chiudere la finestra **Cronologia dei comandi BricsCAD**.

14.1.4 Lavorare con gli stili di quota

Il comando DIMSTILE apre la finestra di dialogo **Esplora Disegno - Stili di Quota**.

Qui è possibile:

- Creare un nuovo stile di quota
- Creare stili di quota secondari
- Modificare uno stile di quota
- Definire le modiche locali degli stili di quota
- Rinominare uno stile di quota
- Impostare come corrente uno stile di quota
- Copiare uno stile di quota tra disegni
- Confrontare gli stili di quota
- Eliminare uno stile di quota

14.1.4.1 Impostare come corrente uno stile di quota

Primo metodo:

1 Cliccare il pulsante destro del mouse sul campo dello stile di quota nella **Barra di Stato**.

Aggiungere un menu contestuale.

2 Selezionare lo stile di quota nel menu contestuale.

Secondo metodo:

1 Avviare il comando DIMSTILE.

1 Lo stile di quota corrente è contrassegnato nella colonna **Corrente**.

2 Cliccare sulla colonna corrente dello stile di quota.

14.1.4.2 Creare un nuovo stile di quota

1 Avviare il comando DIMSTILE.

2 Selezionare uno stile di quota esistente, che verrà utilizzato come modello per il nuovo stile.

3 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante **Nuovo**.
   - Posizionate il cursore su uno stile di quota esistente, quindi cliccare il pulsante destro del mouse e scegliete **Nuovo** nel menu contestuale.

4 Selezionare il campo **Nome** e digitare un nome per il nuovo stile di quota.

5 Definire le impostazioni per il nuovo stile di quota.

Nota: Utilizzare l'opzione **Salva su nuovo stile** del menu contestuale per creare un nuovo stile di quota come copia dello stile di quota selezionato.

14.1.4.3 Creazione di uno stile di quota secondario

Per ogni stile di quota, è possibile creare uno stile secondario per ogni tipo di quota: **Lineare**, **Angolare**, **Raggio**, **Diametro**, **Coordinata** e **Direttrice**.

Se lo stile di quota corrente ha uno stile di quota secondario per un tipo di quota, lo stile di quota secondario verrà utilizzato automaticamente quando viene creato tale tipo di quota.

Per creare uno stile di quota secondario, eseguire le seguenti operazioni:

1 Selezionare lo stile di quota principale, quindi cliccare il pulsante destro del mouse e scegliere **Nuovo stile secondario** nel menu contestuale.

Aggiungere un menu contestuale.

2 Selezionare un tipo di entità quota nel menu contestuale.

3 Il nome dello stile secondario sarà: <Stile principale>:<Tipo entità quota>

4 Selezionare lo stile secondario e definire le proprietà che devono essere diverse dallo stile principale per il tipo di entità quota selezionata.

14.1.4.4 Modificare uno stile di quota

1 Avviare il comando DIMSTILE.

2 Selezionare uno stile di quota.

3 Personalizzare le impostazioni.

4 Chiudere la finestra di dialogo **Esplora Disegno**.

14.1.4.5 Definire le modiche locali degli stili di quota

1 Avviare il comando DIMSTILE.

2 Selezionare uno stile di quota.

3 La prima riga dell'elenco degli stili di quota riporta:<overrides on <nomestile>

4 Definire le modifiche locali.

5 Chiudere la finestra di dialogo **Esplora Disegno**.

Note:

- Si raccomanda di non creare quote con modifiche locali.
- Utilizzare le impostazioni locali delle quote per testare le modifiche, senza modificare la definizione dello stile di quota corrente, quindi utilizzare l'opzione **Salva modifiche locali su stile corrente** o **Salva su nuovo stile** per salvare le modifiche locali.
- Impostando un altro stile corrente si annulleranno le modifiche locali dello stile di quota corrente.

14.1.4.6 Rinominare uno stile di quota

1 Avviare il comando DIMSTILE.

2 Selezionare uno stile di quota.

3 Cliccare il pulsante destro del mouse, quindi selezionare **Rinomina** nel menu contestuale.

4 (opzionale) Digitare un nuovo nome nel campo **Nome**, quindi premere Invio.

14.1.4.7 Copiare uno stile di quota tra disegni

1 Aprire sia il disegno di origine sia il disegno di destinazione.

2 Avviare il comando DIMSTILE.

3 I disegni aperti sono elencati nell'elenco **Disegni Aperti** nella finestra di dialogo **Esplora Disegno**.

4 Selezionare **Stili di Quota** del disegno sorgente nell'elenco **Disegni Aperti**.

5 Selezionare gli stili di quota da copiare.

6 Tenete premuto il tasto Ctrl per selezionare più stili di quota.

7 Posizionare il cursore su uno degli stili di quota selezionati, quindi tenere premuto il pulsante sinistro del mouse e trascinare la selezione sul disegno di destinazione, quindi rilasciare il pulsante sinistro del mouse.

14.1.4.8 Confrontare gli stili di quota

1 Avviare il comando DIMSTILE.

2 Selezionare uno stile di quota.

3 Cliccare sul primo stile di quota, quindi tenete premuto il tasto Ctrl per selezionare lo stile o gli stili di quota successivi.

Vengono visualizzate le proprietà degli stili selezionati. Le impostazioni diverse sono evidenziate.

4 (opzionale) Tenere premuto il tasto Ctrl per aggiungere o rimuovere uno stile di quota dalla tabella di confronto.

14.1.5 Utilizzare gli strumenti degli stili di quota

Gli strumenti degli stili di quota presenti sulla barra degli strumenti **Quotatura** e sul pannello **Quotatura** della barra multifunzione consentono di:

- Applicare lo stile di quota corrente ad una selezione di entità quota.
- Creare un nuovo stile di quota.
- Ripristinare uno stile di quota.

- Elencare lo stato corrente delle impostazioni della quota.

14.1.5.1 Applicare lo stile di quota corrente

1 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Applica Stile di Quota** ( ).
- Scegliere **Applica Stile di Quota** nel menu **Quotatura**.
- Avviare il comando -DIMSTILE, quindi digitare **A** o **Applica** e premere Invio.

2 **Viene richiesto**: Selezionare le quote a cui applicare lo stile corrente [opzioni di selezione (?)]:

3 Selezionare le quote, quindi premere Invio.

Nota: Dato che le impostazioni dello stile di quota si applicano solo alle entità quota, è possibile, nel secondo passaggio, selezionare tutte le entità, per applicare lo stile di quota corrente a tutte le quote nel disegno.

14.1.5.2 Creare un nuovo stile di quota

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Salva Stile di Quota** ( ) sulla barra degli strumenti **Quotatura**.
- Scegliere **Salva Stile di Quota** nel menu **Quotatura**.
- Avviare il comando -DIMSTILEE, quindi digitare **SA** o **Salva** e premere Invio.

Viene richiesto: Immettere il nome dello stile da salvare o [? per elenco stili]:

2 Digitare un nome per il nuovo stile di quota, quindi premere Invio.

3 Se il nome esiste già, la barra dei comandi richiede di ridefinirlo o meno.

14.1.5.3 Ripristinare uno stile di quota

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Ripristina Stile di Quota** ( ) sulla barra degli strumenti **Quotatura**.
- Scegliere **Ripristina Stile di Quota** nel menu **Quotatura**.
- Avviare il comando -DIMSTILE, quindi digitare **R** o **Ripristina** e premere Invio.

Viene richiesto: Stile di quota da ripristinare o [? per elenco stili] <Seleziona quota>:

2 Eseguire una delle seguenti operazioni:

- Digitare un nome di stile di quota e premere Invio.
- Premere Invio, quindi cliccare su un'entità quota nel disegno.
- Lo stile di quota dell'entità selezionata è diventato lo stile di quota corrente.
- Digitare ? per vedere la lista degli stili di quota del disegno corrente.

14.1.5.4 Elencare le impostazioni dello stile di quota corrente

Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Stato Variabili Quota** ( ) sulla barra degli strumenti Quotatura.

- Scegliere **Stato Variabili Quota** nel menu **Quotatura**.
- Avviare il comando -DIMSTILEE, quindi digitare **ST** o **Stato** e premere Invio.

Lo stato delle impostazioni nello stile di quota corrente viene elencato nella finestra della **Cronologia**. Premere il tasto funzione F2 per chiudere la finestra **Cronologia**.

14.1.6 Creare quote lineari

Le quote lineari annotano distanze lineari oppure lunghezze e possono essere orientate sia orizzontalmente sia verticalmente, oppure possono essere allineate parallelamente ad un'entità esistente od ai punti di origine selezionati.

I seguenti comandi creano entità quota lineare:

- DIMLINEARE: consente di creare quote lineari orizzontali e verticali rispetto al sistema di coordinate corrente (Globale o UCS).
- DIMALLINEATA: consente di creare quote parallele a linee e segmenti di polilinea. Il comando funziona anche con segmenti di arco di polilinea, archi e cerchi, ma ne dimensiona i diametri (vedere il comando DIMDIAMETRO).
- DIMLBASE: consente di posizionare più quote lineari e angolari, tutte a partire dallo stesso punto di base di una quota esistente. Questo comando funziona solo quando nel disegno esiste già almeno un'altra quota.
- DIMCONTINUA: consente di posizionare una catena di quote continuando le quote lineari o angolari dal punto finale della quota precedente o di un'altra quota esistente.
- ARCOQUOTA: posiziona le quote che misurano le lunghezze degli archi e dei segmenti di arco polilinea.

14.1.6.1 Creare una quota lineare orizzontale o verticale

1 Avviare il comando DIMLINEARE.

1 **Viene richiesto**: Origine prima linea di estensione <Seleziona entità>:

2 Eseguire una delle seguenti operazioni:
   - Specificare l'origine della prima linea di estensione, quindi specificare l'origine della seconda linea di estensione.
   - Premere Invio, quindi selezionare la linea o il segmento lineare di una polilinea che si desidera quotare.

   La linea di quota viene visualizzata dinamicamente.

   Spostare il cursore verticalmente per creare una quota orizzontale o orizzontalmente per creare una quota verticale.

3 Specificate un punto in cui posizionare la linea di quota.

14.1.6.2 Creare una quota lineare allineata

1 Avviare il comando DIMALLINEATA.

1 **Viene richiesto**: Origine prima linea di estensione <Seleziona entità>:

2 Eseguire una delle seguenti operazioni:
   - Specificare l'origine della prima linea di estensione, quindi specificare l'origine della seconda linea di estensione.
   - Premere Invio, quindi selezionare la linea o il segmento lineare di una polilinea che si desidera quotare.

   La linea di quota viene visualizzata dinamicamente.

3 Specificate un punto in cui posizionare la linea di quota.

14.1.6.3 Creare una quota lineare ruotata

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante dello strumento **Ruotato** nella barra degli strumenti **Quotura** o nel relativo pannello della barra multifunzione.
   - Scegliere **Ruotata** nel menu **Quotatura**.

   Viene richiesto: Origine prima linea di estensione <Seleziona entità>: Eseguire una delle seguenti operazioni:

2 Eseguire una delle seguenti operazioni:
   - Specificare l'origine della prima linea di estensione, quindi specificare l'origine della seconda linea di estensione.
   - Premere Invio, quindi selezionare la linea o il segmento lineare di una polilinea che si desidera quotare.

   Viene richiesto: Angolo della linea di quota <0.0>:

3 Eseguire una delle seguenti operazioni:
   - Digitare un angolo di rotazione nella barra dei comandi, quindi premere Invio.
   - Inserire l'angolo di rotazione specificando due punti.

   La linea di quota viene visualizzata dinamicamente.

4 Specificate un punto in cui posizionare la linea di quota.

14.1.6.4 Creare quote linea di base

1 Avviare il comando DIMLBASE.

1 **Viene richiesto**: Lineabase: Origine della prossima linea di estensione o [Selezionare la quota di partenza/Annulla] <Selezionare la quota di partenza>:

2 Eseguire una delle seguenti operazioni:
   - Andare al passaggio 2 per accettare la quota della linea di base visualizzata dinamicamente, a partire dalla quota lineare o angolare aggiunta più di recente.
   - Premere Invio, quindi selezionare l'entità quota iniziale.

3 Specificate un punto per creare la prima quota base.

4 Continuare a specificare i punti o premere Invio per interrompere.

Note:

- Quando si seleziona la quota iniziale accertarsi di cliccare la prima linea di estensione oppure la prima metà della linea di quota al fine di utilizzare il primo punto d'origine della quota iniziale come punto di origine per la quota linea di base.
- Se si seleziona una quota angolare, verranno create quote angolari impilate.
- La distanza tra le linee delle quote impilate è controllata dalla variabile di sistema DIMDLI (Proprietà della spaziatura della linea di base della quota nello stile di quota).

14.1.6.5 Creare quote continue

1 Avviare il comando DIMCONTINUA.

1 **Viene richiesto**: Continua: Origine della prossima linea di estensione o [Selezionare la quota di partenza/Annulla] <Selezionare la quota di partenza>:

2 Eseguire una delle seguenti operazioni:
   - Passare al punto 2 per accettare la quota continua visualizzata dinamicamente, a partire dalla quota lineare o angolare aggiunta più di recente.
   - Premere Invio, quindi selezionare l'entità quota iniziale.

3 Specificare un punto per creare la prima quota continua.

4 Continuare a specificare i punti o premere Invio per interrompere.

Note:

- Quando si seleziona una quota iniziale accertarsi di cliccare la seconda linea di estensione oppure la seconda metà della linea di quota al fine di utilizzare il secondo punto di origine della quota iniziale come punto di origine della quota continua. Diversamente la nuova linea di quota sovrapporrà parzialmente la quota iniziale.
- Se si seleziona una quota angolare, verranno create quote angolari continue.

14.1.6.6 Creare una quota di lunghezza arco

1 Avviare il comando ARCOQUOTA:

1 **Viene richiesto**: Selezionare arco o segmento arco di polilinea:

2 Selezionare il segmento dell'arco o della polilinea.

3 La quota della lunghezza dell'arco viene visualizzata dinamicamente.

4 Specificate un punto in cui posizionare la linea di quota.

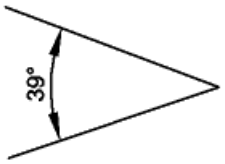
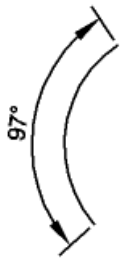
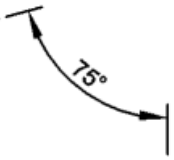
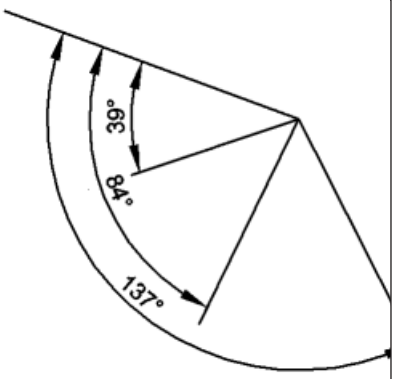
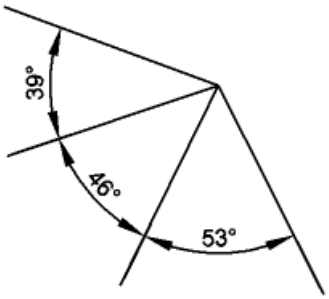
14.1.7 Creare quote angolari

Il comando DIMANGOLO crea una quota angolare.

Le quote angolari annotano l'angolo misurato tra due linee, due segmenti lineari di una polilinea o l'angolo di un arco circolare. È anche possibile quotare un angolo selezionando un vertice dell'angolo e due punti finali.

A partire da una quota angolare esistente, è possibile aggiungere una quota linea di base o una quota continua.

- Una quota linea di base angolare inserisce una quota aggiuntiva da un'origine comune della prima linea di estensione.
- Una quota continua angolare continua una quota angolare dalla seconda linea di estensione di una quota angolare precedente.

		
quota angolare misurata tra due linee	quota angolare di un arco	quota angolare definita da un vertice e due punti finali
		
quote angolari linea di base	quote angolari continue	

14.1.7.1 Quotare un angolo tra due linee

1 Avviare il comando DIMANGOLO.

1 **Viene richiesto**: Selezionare linea, arco o cerchio [opzioni di selezione (?)] <Premere INVIO per specificare angolo>:

2 Selezionare un segmento di linea o polilinea lineare.

3 **Viene richiesto**: Altra linea della quota angolare [opzioni di selezione (?)]:

4 Selezionare una seconda linea o un segmento lineare di polilinea.

5 La quota angolare viene visualizzata dinamicamente.

Viene richiesto: Posizione dell'arco di quota [Angolo/Testo]:

6 Specificare un punto per posizionare la quota.

7 Spostare il cursore per quotare l'angolo interno tra le linee o il loro prolungamento.

14.1.7.2 Quotare l'angolo di un arco

1 Avviare il comando DIMANGOLO.

1 **Viene richiesto**: Selezionare linea, arco o cerchio [opzioni di selezione (?)] <Premere INVIO per specificare angolo>:

2 Selezionare un arco.

3 La quota angolare viene visualizzata dinamicamente.

Viene richiesto: Posizione dell'arco di quota [Angolo/Testo]:

4 Specificare un punto per posizionare la quota.

14.1.7.3 Creare una quota angolare definita da un vertice e due punti finali

1 Avviare il comando DIMANGOLO.

1 **Viene richiesto**: Selezionare linea, arco o cerchio [opzioni di selezione (?)] <Premere INVIO per specificare angolo>:

2 Premere Invio.

3 **Viene richiesto**: Vertice dell'angolo:

4 Specificare un punto.

5 **Viene richiesto**: Primo lato dell'angolo:

6 Specificare un punto.

7 **Ti viene richiesto**: altro lato dell'angolo:

8 Specificare un punto.

9 La quota angolare viene visualizzata dinamicamente.

Viene richiesto: Posizione dell'arco di quota [Angolo/Testo]:

10 Specificare un punto per posizionare la quota.

11 Spostare il cursore per quotare l'angolo interno o esterno.

14.1.8 Creare quote diametro e raggio

I comandi DIMCENTRO e DIMDIAMETRO creano quote diametro e raggio per annotare i raggi e i diametri di cerchi e archi circolari.

14.1.8.1 Creare una quota diametro

1 Avviare il comando DIMDIAMETRO.

1 **Viene richiesto**: Selezionare arco o cerchio da quotare [opzioni di selezione (?)]:

2 Selezionare un arco o un cerchio.

3 La quota diametro viene visualizzata dinamicamente.

4 Specificare un punto in cui posizionare l'entità quota.

14.1.8.2 Creare una quota raggio

1 Avviare il comando DIMRAGGIO.

1 **Viene richiesto**: Selezionare arco o cerchio da quotare [opzioni di selezione (?)]:

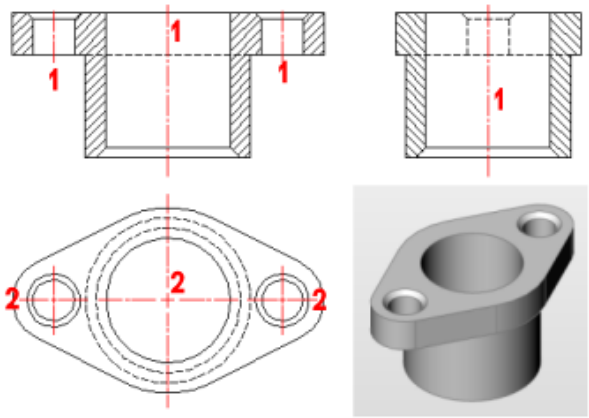
2 Selezionare un arco o un cerchio.

3 La quota raggio viene visualizzata dinamicamente.

4 Specificare un punto in cui posizionare l'entità quota.

14.1.9 Linee d'asse e marcatori centro

Le linee d'asse e i marcatori centro disegnano entità di riferimento per indicare assi di simmetria e centri di fori circolari. Le linee d'asse e marcatori centro sono entità associative: quando le linee di riferimento, i cerchi o gli archi circolari vengono modificati, vengono automaticamente riposizionate.



1 Lineaasse

2 Marcatore Centro

Il comando LINEAASSE crea un'entità di linea d'asse associata a due linee selezionate.

Il comando CENTRO crea un'entità marcatore centro associata a un cerchio o un arco selezionato.

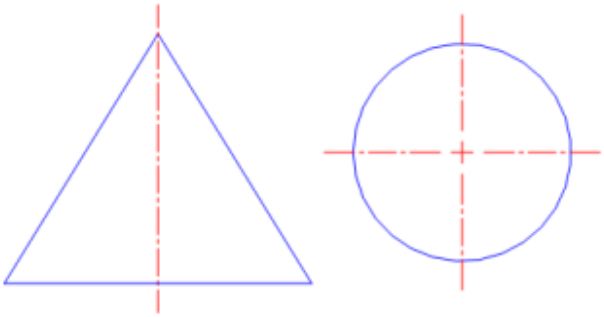
Il comando RIASSOCIACENTRO ricrea l'associazione tra un'entità linea d'asse e due linee selezionate, o tra un marcatore centro ed un cerchio o arco selezionato.

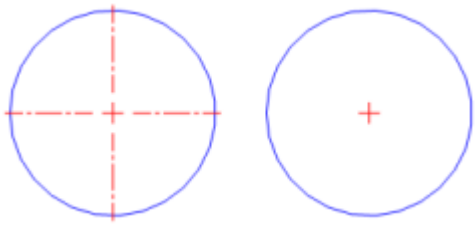
Il comando DISSOCIACENTRO interrompe l'associazione delle entità linee d'asse e marcatori centro.

Il comando REIMPOSTACENTRO ripristina le linee d'asse e le entità dei marcatori centro.

14.1.9.1 Variabili di sistema per entità linee d'asse e marcatori centro

Le proprietà e la geometria delle entità linea d'asse e marcatori centro vengono controllate tramite variabili di sistema.

Nome	Titolo	Descrizione
CENTERCROSSGAP	Vuoto marcatore centro	Determina il divario tra il marcatore centro e le sue linee d'asse. Digitare un valore per specificare la dimensione assoluta dell'intervallo in unità di disegno. Digitare un valore seguito da x per impostare la dimensione del divario rispetto al diametro del cerchio o dell'arco circolare. Il valore predefinito 0.05x crea un divario di 5/100 del diametro. . (punto) non crea spazi vuoti.
CENTERCROSSSIZE	Dimensione della croce del marcatore centro	Determina la dimensione della croce del marcatore centro. Digitare un valore per specificare la dimensione assoluta delle linee del marcatore centro in unità di disegno. Digitare un valore seguito da x per impostare la dimensione del marcatore centro rispetto al diametro del cerchio o dell'arco circolare. Il valore predefinito 0.1x crea marcatori centro 1/10 del diametro. Digitare un . (punto) per creare marcatori centro con linee di lunghezza zero.
CENTEREXE	Lunghezza estensioni linea d'asse	Definisce la lunghezza delle estensioni della linea d'asse. Il valore è espresso in unità di disegno. 

Nome	Titolo	Descrizione
CENTERLAYER	Layer predefinito per marcatore centro o linea d'asse	Specifica il layer predefinito per le nuove linee d'asse e marcatori centro. Digitare un nome. Digitare un nome. Se tale layer non esiste, il layer verrà creato durante la creazione della prima Linea d'Asse o Marcatore Centro. Digirare un . (punto) per creare linee d'asse e marcatori centro sul layer corrente.
CENTERLTYPE FILE	File del tipo di linea per il marcatore centro o linea d'asse	Specifica il file del tipolinea utilizzato per creare linee d'asse e marcatori centro. Predefinito nei disegni con unità imperiali: **default.lin** Predefinito nei disegni con unità metriche: **iso.lin**
CENTERLTYPE	Tipo di linea per il marcatore centro o linea d'asse	Specifica il tipolinea per le linee d'asse e marcatori centro appena creati. Digitare un nuovo nome del tipolinea o premere Invio per accettare il nome corrente. Digitare un . (punto) per linee continue.
CENTERLTSCALE	Scala tipo di linea per il marcatore centro o linea d'asse	Specifica la scala del tipolinea delle linee d'asse e marcatori centro appena creati.
CENTERMARKEXE	Estensione automatica per marcatore centro o linea d'asse	Determina se le linee d'asse si estendono (a sinistra) o meno (a destra) per i nuovi marcatori centro. 

14.1.9.2 Modificare le entità linee d'asse e marcatori centro

Le proprietà delle entità centro possono essere modificate nel pannello **Proprietà**.

14.1.10 Creare quote coordinata

Una quota coordinata quota la distanza perpendicolare da un'origine oppure da un punto base (l'origine del sistema coordinate utente corrente (WCS o UCS). Le quote coordinate consistono in una coordinata-x od -y ed una direttrice. Una quota coordinata-x misura la distanza lungo l'asse x, mentre una quota coordinata-y misura la distanze lungo l'asse y.

Dopo aver selezionato i punti della quota coordinata, il programma determina automaticamente se il punto è una coordinata-x o una coordinata-y sulla base della direzione di trascinamento utilizzata per il secondo punto. È, comunque, possibile specificare se la coordinata rappresenta una coordinata-x o una coordinata-y. Il testo delle quote coordinata è sempre allineato con la linea della direttrice, indipendentemente dall'orientamento del testo specificato dallo stile di quota corrente.

14.1.10.1 Creare una quota coordinata

1 Impostare l'origine del sistema di coordinate nel punto di riferimento per le quote coordinata.

2 Avviare il comando DIMCOORDINATA.

3 **Viene richiesto**: Seleziona punto per quota coordinata:

4 Specificare un punto.

5 La quota coordinata viene visualizzata dinamicamente.

Viene richiesto: Punto finale direttrice [X/Y/Testo/Angolo]:

6 Cliccare per posizionare la quota coordinata.

14.1.11 Modificare le quote

È possibile modificare le entità quota:

- Utilizzando le grips.
- Modificando il testo della quota.
- Ruotando il testo della quota.
- Riposizionando il testo della quota.
- Ripristinando la posizione del testo.
- Rendendo oblique le linee di estensione.

14.1.11.1 Utilizzare i grips per modificare una quota lineare

1 Selezionare un'entità quota.

1 Verranno visualizzate 5 grips:

2 grips sui punti di origine (1)

2 grips sulla linea di quota (2)

1 grip sul testo di quota (3)

2 (opzionale) Trascinare le grips del punto di origine per regolare i punti di origine della quota.

3 Il testo della quota viene regolato automaticamente.

4 (opzionale) Trascinare i grips della linea di quota per spostare la linea di quota.

5 (opzionale) Trascinare il grip del testo per spostare il testo della quota.

Note:

- A seconda dell'impostazione **Sposta Testo** (variabile di sistema DIMTMOVE), la linea di quota si sposta lungo il testo oppure il testo si sposta in modo indipendente. Verrà aggiunta una direttrice se l'impostazione **Sposta Testo** è impostata su **Aggiungi un direttrice quando il testo quota viene spostato**.
- Se due quote adiacenti (quote continue) vengono selezionate insieme, non è possibile spostare simultaneamente i loro grips coincidenti.

14.1.11.2 Modificare il testo di quota

Il testo di quota può essere modificato:

- nell'editor TestoM:
  - utilizzando il comando DDEDIT
  - doppio clic sull'entità della quota
- nel campo **Sovrascrivi testo** nel pannello **Proprietà**.
- utilizzando l'opzione **Edita testo** del comando DIMEDITA.

Note:

Per formattare il testo della quota: possono essere utilizzate le seguenti operazioni:

- **\L** (barra rovesciata L maiuscola) avvia il testo sottolineato, **\l** (barra rovesciata L minuscola) per arrestare il testo sottolineato
- **<>** per aggiungere il testo predefinito della quota (= distanza misurata)
- **[]** per aggiungere le unità alternative, quando le unità alternative sono attualmente spente nello stile di quota.
- **%%d** per aggiungere il simbolo dei gradi (°)
- **%%c** per aggiungere il simbolo del diametro (Ø)
- **%%p** per aggiungere il simbolo più/meno (±)

ad es. se si digita **%%c\L<> cm** nel campo **Sovrascrittura testo**, il testo della quota viene visualizzato come:

Ø24,43cm

14.1.11.3 Ruotare il testo della quota

1 Eseguire una delle seguenti operazioni:
   - Cliccare sullo strumento **Ruota Testo Quota** ( ) nella barra degli strumenti **Quotatura** o nel barra multifunzione.
   - Scegliere **Ruota Testo Quota** nel menu **Quotatura**.
   - Avviare il comando DIMEDITA, quindi scegliere l'opzione **Ruota testo**.

   Viene richiesto: Angolo del testo di quota:

2 Digitare un valore, quindi premere Invio.

3 **Viene richiesto**: Selezionare entità quota per ruotare il testo [opzioni di selezione (?)]:

4 Selezionare le quote e premere Invio.

14.1.11.4 Riposizionare il testo della quota

1 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Riposiziona Testo Quota** ( ) nella barra degli strumenti **Quotatura** o nella barra multifunzione.
- Scegliere **Riposiziona Testo Quota** nel menu **Quotatura**.
- Avviare il comando DIMTEDIT.

Viene richiesto: Selezionare quote per riposizionamento testo:

2 Selezionate la quota.

3 Il testo della quota si muove dinamicamente.

4 Cliccare per riposizionare il testo della quota.

14.1.11.5 Ripristinare la posizione del testo

1 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Ripristina Posizione Testo** ( )
- Scegliere **Ripristina Posizione Testo** nel menu **Quotatura**.
- Avviare il comando DIMEDITA e scegliere l'opzione **Ripristina testo**.

Viene richiesto: Selezionare quote per ripristinare il loro testo nella posizione predefinita [opzioni di selezione (?)]:

2 Selezionare le quote, quindi premere Invio.

14.1.11.6 Rendere oblique le linee di estensione

1 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Rendi Obliquo** ( ) nella barra degli strumenti **Quotatura** o nella barra multifunzione.
- Scegliere **Rendi Obliquo** nel menu **Quotatura**.
- Avviare il comando DIMEDITA e scegliere l'opzione **Linee Oblique**.

Viene richiesto: Selezionare quote lineari da rendere oblique [opzioni di selezione (?)]:

2 Selezionare le entità quota, quindi premere Invio.

3 **Viene richiesto**: Inserisci angolo obliquo:

4 Digitare un valore, quindi premere Invio.

Nota: Per ripristinare le linee di estensione non ruotate, eseguire la procedura descritta qui sopra e digitare 0 (zero) alla richiesta 'Inserisci angolo obliquo'.

14.1.12 Creare direttrici ed annotazioni

Una direttrice consiste in una freccia, una linea, polilinea oppure una spline ed una annotazione.

La procedura generale per posizionare una direttrice è:

- Specificare la posizione della punta della freccia.
- Creare i segmenti di linea della direttrice.
- Digitare il testo della direttrice.

14.1.12.1 Creare una direttrice ed una annotazione

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante dello strumento **Direttrice** ( ) nella barra degli strumenti **Quotatura** o nel pannello della barra multifunzione.
   - Scegliere **Direttrice** nel menu **Quotatura**.
   - Avviare il comando DIRETTRICE o DIMDIRETTRICE.

   Viene richiesto: Inizio direttrice:

2 Specificare la posizione della punta della freccia.

3 **Viene richiesto**: Punto successivo:

4 Specificare il punto finale del primo segmento della direttrice.

5 **Viene richiesto**: Specificare il punto successivo o [Formato/Annulla/Nota]<Nota>:

6 Aggiungere altri segmenti alla direttrice, quindi premere Invio o cliccare il pulsante destro del mouse per interrompere l'aggiunta di segmenti.

7 **Viene richiesto**: Prima riga del testo di annotazione <opzioni>:

8 Digitare la prima riga del testo dell'annotazione, quindi premere Invio.

9 **Viene richiesto**: Riga successiva del testo di annotazione:

10 Eseguire una delle seguenti operazioni:
   - Premere Invio oppure cliccare il pulsante destro del mouse per creare la direttrice e l'annotazione.
   - Continuare l'aggiunta di righe al testo di annotazione, quindi premere Invio oppure cliccare il pulsante destro del mouse per creare la direttrice e l'annotazione.

14.1.12.2 Creare solo una direttrice

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante dello strumento **Direttrice** ( ) nella barra degli strumenti **Quotatura** o nel pannello della barra multifunzione.
   - Scegliere **Direttrice** nel menu **Quotatura**.
   - Avviare il comando DIRETTRICE o DIMDIRETTRICE.

   Viene richiesto: Inizio direttrice:

2 Specificare la posizione della punta della freccia.

3 **Viene richiesto**: Punto successivo:

4 Specificare il punto finale del primo segmento della direttrice.

5 **Viene richiesto**: Specificare il punto successivo o [Formato/Annulla/Nota]<Nota>:

6 Aggiungere altri segmenti alla direttrice, quindi premere Invio o cliccare il pulsante destro del mouse per interrompere l'aggiunta di segmenti.

7 **Viene richiesto**: Prima riga del testo di annotazione <opzioni>:

8 Premere Invio.

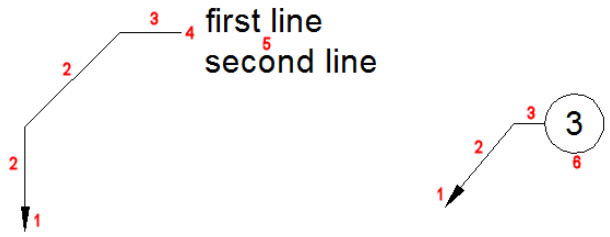
9 **Ti viene richiesto**: Opzioni testo di quota [Blocco/Copia/Nessuna/Tolleranza/testoM] <testoM>:

10 Scegliere l'opzione **Nessuna**.

14.1.13 Creare Multidirettrici

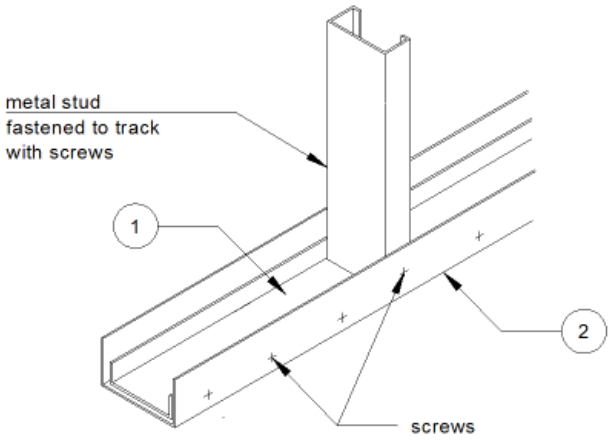
Una multidirettrice consiste in:

- una punta di freccia (1)
- una o più linee direttrici (2), che possono essere dritte o curve
- facoltativamente una linea di collegamento orizzontale (3) ed una distanza dal testo (4)
- contenuto: un testo multilinea (5) oppure un blocco (6)



Le Multidirettrici vengono create utilizzando lo stile multidirettrice corrente. Ogni disegno contiene uno stile **Standard** della multidirettrice, che non può essere rinominato o eliminato; lo stile multidirettrice **Standard** può essere, però, personalizzato. Il comando MULTIDIR permette di definire le modifiche locali dello stile multidirettrice corrente.

Le Multidirettrici possono essere create partendo dalla punta della freccia, oppure dalla linea di collegamento o dal contenuto.



14.1.13.1 Creare una multidirettrice

1 (opzionale) Avviare il comando STILEMULTIDIR, quindi impostare l'attuale stile multidirettrice.

2 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Multidirettrice** ( ) nella barra degli strumenti **Quotatura** o nel pannello della barra multifunzione.
- Scegliere **Multidirettrice** nel menu **Quotatura**.
- Avviare il comando MULTIDIR.

Viene richiesto: Specificare la posizione della punta della freccia direttrice o [Linea di collegamento direttrice prima/Contenuto prima/OPzioni] <Opzioni>:

3 (opzionale) Selezionare un'opzione di posizionamento:

- punta della freccia prima
- linea collegamento prima
- contenuto prima.

L'opzione di posizionamento utilizzata più di recente verrà ripetuta.

4 La sequenza predefinita è: punta della freccia - linea di collegamento - contenuto.

5 (opzionale) Scegliere **Opzioni** se si desidera definire modifiche locali sullo stile multidirettrice corrente.

Viene richiesto: Digitare un'opzione [tipo di DIrettrice/linea di COllegamento/tipo di Contenuto/n. max Punti/priMo angolo/seconDO angolo/opzioni di UScita] <opzioni di UScita>:

Specificare le opzioni, quindi scegliere **Uscita** per continuare.

6 Specificate un punto per definire la posizione della punta della freccia della direttrice.

7 **Viene richiesto**: Specificare la posizione della linea di collegamento:

8 Specificare un punto per definire la posizione della linea di collegamento.

9 A seconda del tipo di contenuto:

- **TestoM**: viene lanciato il comando TestoM.
- **Blocco**: viene inserito un blocco.
- **Nessuno**: il comando verrà terminato.

14.1.13.2 Aggiungere vertici a una linea di direttrice

1 Se necessario, premere il tasto funzione **F12** per attivare il menu cursore **Quad**.

2 Posizionare il cursore sulla linea della direttrice a cui si desidera aggiungere un vertice, quindi tenere premuto il tasto Ctrl.

3 Viene visualizzato il menu del cursore **Quad**.

4 Spostare il cursore sul menu del cursore **Quad** ed espandere la scheda **Edita**.

5 Cliccare sullo strumento **Aggiungi Vertice** ( ).

Un nuovo vertice viene aggiunto dinamicamente.

6 Specificare la posizione del vertice.

14.1.13.3 Rimozione di vertici da una linea di direttrice

1 Se necessario, premere il tasto funzione **F12** per attivare il menu cursore **Quad**.

2 Posizionare il cursore sulla linea della direttrice a cui si desidera aggiungere un vertice, quindi tenere premuto il tasto Ctrl.

3 Viene visualizzato il menu del cursore **Quad**.

4 Spostare il cursore sul menu del cursore **Quad** ed espandere la scheda **Edita**.

5 Cliccare sullo strumento **Rimuovi Vertice** ( ).

Il vertice viene rimosso.

14.1.13.4 Aggiungere linee di direttrice ad una multidirettrice

1 Se necessario, premere il tasto funzione **F12** per attivare il menu cursore **Quad**.

2 Appoggia il cursore sull'entità multidirettrice.

3 Viene visualizzato il menu del cursore **Quad**.

4 Spostare il cursore sul menu del cursore **Quad** ed espandere la scheda **Edita**.

5 Cliccare sullo strumento **Aggiungi Direttrice** ( ).

Viene visualizzata una nuova linea di direttrice dinamicamente.

6 Specificare l'origine della nuova linea di direttrice.

7 Viene visualizzata in modo dinamico un'altra nuova linea di direttrice.

8 Eseguire una delle seguenti operazioni:

- Specificare l'origine della nuova linea di direttrice.
- Cliccare il pulsante destro del mouse o premere Invio per arrestare l'aggiunta di linee di direttrice.

(*) Quando la proprietà N.massimo di punti direttrice dello stile multidirettrice è superiore a 2, cliccare il pulsante destro del mouse per interrompere l'aggiunta di vertici per la nuova linea di direttrice.

14.1.13.5 Rimuovere linee di direttrici da una multidirettrice

1 Se necessario, premere il tasto funzione **F12** per attivare il menu cursore **Quad**.

2 Appoggiare il cursore sulla linea della direttrice che si desidera rimuovere.

3 Viene visualizzato il menu del cursore **Quad**.

4 Spostare il cursore sul menu del cursore **Quad** ed espandere la scheda **Edita**.

5 Cliccare sullo strumento **Rimuovi Direttrice** ( ).

La linea della direttrice viene rimossa.

14.1.14 Aggiungere Tolleranze Geometriche

Le Tolleranze Geometriche specificano la variazione massima che è consentita in forma o posizione rispetto la geometria reale. La tolleranza geometrica è, in sostanza, lo scarto di tolleranza all'interno del

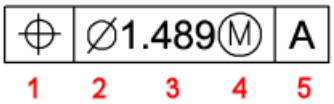
quale una superficie o asse di un foro o di un cilindro può essere considerato corretto, e quindi accettabile per il corretto funzionamento.

BricsCAD disegna la tolleranza geometriche utilizzando una cornice di controllo costituita da un rettangolo suddiviso in sezioni.

Ogni cornice di controllo della caratteristica consiste in almeno due sezioni. La prima sezione contiene un simbolo di tolleranza geometrica che indica la caratteristica geometrica a cui viene applicata la tolleranza, come la posizione, l'orientamento, o la forma. Ad esempio, una tolleranza della forma può indicare la planarità o rotondità di una superficie. Ad esempio, una tolleranza di forma può indicare la planarità o la rotondità di una superficie.

La seconda sezione contiene il valore della tolleranza. Quando appropriato, il valore della tolleranza è preceduto da un simbolo di diametro e seguito da un simbolo di condizione materiale. Le condizioni dei materiali applicano le caratteristiche che possono variare di dimensione.

Il valore della tolleranza può quindi essere seguito da lettere riferite ai dati che possono essere primari, secondari o terziari, in concomitanza con le condizioni del materiale di ogni dato. Le lettere riferite ai dati sono generalmente utilizzate come tolleranze di riferimento da uno fino a tre piani perpendicolari dai quali viene effettuata una misura. Inoltre le lettere riferite ai dati possono anche indicare un punto esatto oppure un'asse.



1 Simbolo della caratteristica geometrica

2 Simbolo del diametro

3 Valore della tolleranza

4 Simbolo della condizione del materiale

5 Riferimento dato

Quando due tolleranze vengono applicate alla stessa geometria, è possibile anche aggiungere una tolleranza composita che consiste in un valore di tolleranza primaria seguita da un valore di tolleranza secondaria. Per rendere una tolleranza più specifica, è possibile che essa contenga una tolleranza proiettata che consiste in un valore di altezza seguito da un simbolo di tolleranza proiettata. Per esempio, è possibile utilizzare una tolleranza proiettata per indicare la perpendicolarità di una parte incorporata.

14.1.14.1 Creare una cornice di tolleranza geometrica

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Tolleranza** (⊥) nella barra degli strumenti **Quotatura** o nel pannello della barra multifunzione.
- Scegliere **Tolleranza** nel menu **Quotatura**.
- Digitare **tolleranza** nella barra dei comandi, quindi premere Invio.

Verrà visualizzata la finestra di dialogo Tolleranza geometrica.

2 Comporre la cornice della tolleranza geometrica nella finestra di dialogo **Tolleranza geometrica**.

3 Cliccare sul pulsante **OK**.

La finestra di dialogo **Tolleranza geometrica** viene chiusa.

4 Specificare un punto per inserire la cornice della tolleranza geometrica nel disegno.

14.1.14.2 Modifica una cornice di tolleranza geometrica

1 Doppio clic sulla cornice della tolleranza geometrica.

1 Viene visualizzata la finestra di dialogo **Tolleranza geometrica**.

2 Modificare le impostazioni.

3 Cliccare sul pulsante **OK**.

La finestra di dialogo **Tolleranza geometrica** viene chiusa.

Il contenuto della cornice della tolleranza geometrica viene aggiornato.

14.2 Texts

14.2.1 Variabili di sistema per il testo

Le variabili di sistema possono essere impostate mediante la finestra di dialogo **Impostazioni**.

Nome Variabile	Titolo Variabile	Descrizione
FONTALT	Font alternativo	Definisce il font da usare nel caso un font testo non venga trovato.
HIDETEXT	Nascondi testo con NASCONDI	Specifica se il comando NASCONDI tratta entità di testo.
MIRRTEXT	Specchiatura del Testo	Determina se il testo viene specchiato o meno dal comando SPECCHIO.
MTEXTED	Editor testo multilinea	Imposta l'editor di testo primario e secondario da usare per le entità di testo multilinea.
MTEXTFIXED	Posizione di visualizzazione del testo multilinea	Controlla se BricsCAD deve eseguire uno zoom, una rotazione e/o un pan della vista per adattarla al TestoM in fase di modifica.
QTEXTMODE	Modalità testo rapido	Controlla come vengono visualizzate le entità testo. Se attivato, il testo viene visualizzato come riquadro.
TEXTANGLE	Angolo testo	Archivia l'angolo dell'ultima entità testo aggiunta

Nome Variabile	Titolo Variabile	Descrizione
TEXTFILL	Riempimento testo	Determina se i caratteri True Type vengono visualizzati come riempiti o vuoti. Filled Text Outlined Text
TEXTQLTY	Qualità testo	Definisce la nitidezza dei font True Type per la stampa ed il rendering.
TEXTSIZE	Dimensioni testo	Imposta l'altezza predefinita per le nuove entità testo. TEXTSIZE non ha effetto se lo stile di testo corrente ha un'altezza fissa.
TEXTSTYLE	Stile di testo	Salva il nome dello stile di testo corrente.
TSPACEFAC	Fattore spazio testo	Specifica l'interlinea delle righe del testo multilinea, misurata come fattore dell'altezza del testo. Sono accettati valori compresi tra 0,25 e 4.
TSPACETYPE	Tipo spazio testo	Specifica il tipo di interlinea utilizzata per il testo multilinea. Almeno: regola la spaziatura delle linee in base ai caratteri più alti in una linea. Esatto: utilizzata l'interlinea specificata, indipendentemente dalle dimensioni dei singoli caratteri.
TSTACKALIGN	Allineamento impilamento testo	Determina l'allineamento verticale del testo impilato: basso, centro, alto.
TSTACKSIZE	Dimensione impilamento testo	Consente di specificare la percentuale dell'altezza del testo impilato rispetto all'altezza attuale del testo selezionato. Sono accettati valori compresi tra 25 e 125.

14.2.2 Lavorare con gli stili di testo

Quando si aggiunge un testo ad un disegno, viene utilizzato lo stile di testo corrente, che determina il font, la dimensione, l'orientamento ed altre proprietà della nuova entità testo.

Ogni disegno ha uno stile di testo predefinito, denominato **Standard**, che inizialmente utilizza il carattere **Arial**. Non è possibile eliminare o rinominare lo stile **Standard**, ma è possibile modificarlo.

Cambiando il **font** o l'**orientamento** di uno stile esistente, tutte le entità di testo esistenti create utilizzando quello stile verranno automaticamente aggiornate per riflettere il cambiamento. Il cambiamento di una qualsiasi altra proprietà non ha invece alcun effetto sul testo esistente.

E' possibile creare ed utilizzare un numero illimitato di stili di testo aggiuntivi.

Gli stili di testo annotativi creano entità di testo annotative. In uno stile di testo annotativo, l'**Altezza** definisce l'altezza del testo nello spazio carta; l'altezza nello spazio modello dipende dalla scala annotazione corrente definita dalle variabili di sistema CANNOSCALE e CANNOSCALEVALUE.

14.2.2.1 Aprire la finestra di dialogo Esplora Stili di Testo

Eseguire una delle seguenti operazioni:

- Nel menu **Strumenti** > **Esplora Disegno**, scegliere **Stili di Testo...**
- Cliccare sul campo **Stile** nella **Barra di Stato**.
- Avviare il comando STILE.

14.2.2.2 Creare uno stile di testo

1 Aprire la finestra di dialogo **Esplora Stili di Testo**.

2 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Nuovo** ( ) nella sezione **Dettagli** della finestra **Esplora Disegno**.
- Selezionare uno stile esistente, quindi cliccare il pulsante destro del mouse e scegliere **Nuovo** nel menu contestuale.

Viene creato uno stile di testo **NuovoStile**, che eredita le proprietà dello stile attualmente selezionato.

3 Digitare un nome nel campo **Nome Stile di Testo**, sostituendo il nome predefinito **NuovoStile**.

4 Definire le proprietà dello stile di testo.

5 Chiudere la finestra di dialogo **Esplora Stile di Testo**.

14.2.2.3 Modificare uno stile di testo

1 Aprire la finestra di dialogo **Esplora Stili di Testo**.

2 Selezionare lo stile di testo.

3 Definire le proprietà dello stile di testo.

4 Chiudere la finestra di dialogo **Esplora Stile di Testo**.

14.2.2.4 Impostare come corrente uno stile di testo

Eseguire una delle seguenti operazioni:

- Cliccare il pulsante destro del mouse sul campo **Stile di Testo Corrente** nella **Barra di Stato**, quindi cliccare sullo stile del testo.
- Aprire la finestra di dialogo **Esplora Stili di Testo**.

  Lo stile del testo corrente è indicato nella colonna **Corrente**.

  Eseguire una delle seguenti operazioni:

  - Cliccare sul campo **Corrente** dello stile di testo.
  - Cliccare il pulsante destro del mouse sullo stile del testo, quindi scegliere **Imposta Corrente** nel menu contestuale.

14.2.2.5 Eliminare uno stile di testo

1 Aprire la finestra di dialogo **Esplora Stili di Testo**.

2 Selezionare lo stile di testo.

3 Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Elimina** (✕).
- Selezionare lo stile del testo, quindi scegliere **Elimina** nel menu contestuale.

Se lo stile di testo non è utilizzato nel disegno o in uno stile di quota, lo stile di testo verrà eliminato, altrimenti verrà visualizzato un messaggio.

4 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Elimina** per eliminare lo stile di testo e tutte le entità che lo stanno utilizzando.
- Cliccare sul pulsante **Cambia...** per eliminare lo stile di testo ed assegnare uno stile di testo diverso alle entità, che stanno correntemente utilizzando lo stile di testo selezionato.
- Cliccare il pulsante **Annulla** per annullare la procedura di eliminazione.

Nota: Non è possibile eliminare lo stile di testo corrente.

14.2.3 Creare un testo

Il comando TESTO inserisce singole righe di testo nel disegno; se lo si desidera, valuta le espressioni LISP.

Il comando TESTOM inserisce il testo in un paragrafo formattato in un riquadro di delimitazione che limita l'estensione del testo.

14.2.3.1 Creare testo a riga singola

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Testo** (A) sulla barra degli strumenti **Disegno** o sul pannello **Annota/Testo** della barra multifunzione.
- Scegliere **Testo** nel menu **Disegna**.
- Avviare il comando TESTO.

2 **Viene richiesto**: Punto iniziale del testo o [usa Stile definito/ALlinea su linea/adaTta tra punti/Centra orizzontalmente/Medio (oriz/vert)/giustifica a Destra/opzioni Giustificazione...]:

3 Specificare un punto.

4 **Viene richiesto**: Altezza del testo <altezza corrente>:

5 Eseguire una delle seguenti operazioni:

- Premere Invio o cliccare il pulsante destro del mouse per accettare l'altezza del testo corrente.
- Digitare un valore.
- Cliccare su un punto per definire graficamente l'altezza.

Viene richiesto: Angolo di rotazione del testo <angolo corrente>:

6 Eseguire una delle seguenti operazioni:

- Premere Invio o cliccare il pulsante destro del mouse per accettare l'angolo di rotazione

corrente.

- Digitare un valore.
- Cliccare su un punto per definire graficamente l'angolo di rotazione.

7 Digitare il testo, quindi premere Invio.

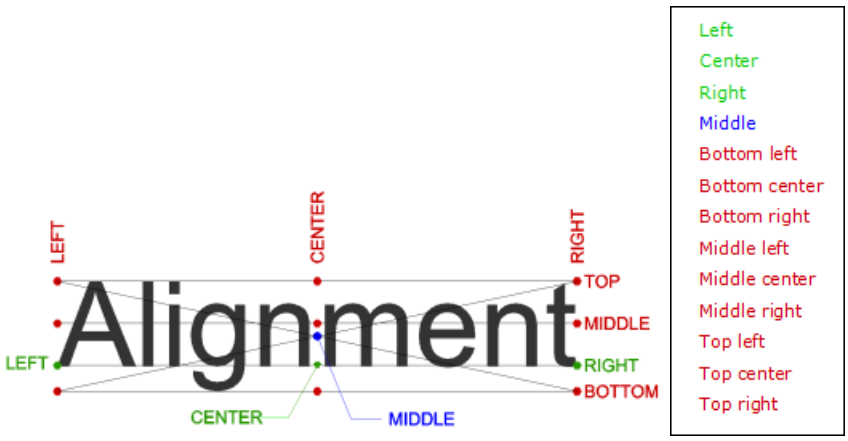
8 Viene creato un nuovo testo vuoto.

9 Eseguire una delle seguenti operazioni:
- Digitare il testo.
- Premere Invio per interrompere.

10 (opzionale) Continua ad aggiungere testi, quindi premere Invio due volte per interrompere.

Nota: Una volta creato il testo nell'attuale sessione BricsCAD, l'entità di testo più recente si evidenzia quando si ripete il comando TESTO. Premere Invio o cliccare il pulsante destro del mouse per creare un nuovo testo sotto il testo creato più di recente, utilizzando la stessa altezza e angolo di rotazione del testo.

Opzioni di giustificazione del testo a riga singola:



14.2.3.2 Creare testo multilinea

1 Eseguire una delle seguenti operazioni:
- Cliccare sul pulsante dello strumento **Testo Multilinea** (A) sulla barra degli strumenti **Disegno** o sul pannello **Annota/Testo** della barra multifunzione.
- Scegliere **Testo Multilinea** nel menu **Disegna**.
- Avviare il comando TESTOM.

Viene richiesto: Testo Multilinea: Primo angolo per blocco di testo:

2 Specificate il punto di inserimento dell'entità testo multilinea.

3 **Viene richiesto**: Selezionare angolo opposto per blocco di testo o [Giustificato/angolo di Rotazione/Stile di testo/Altezza testo/Direzione/Larghezza/SPaziatura linea/Colonne]:

4 Eseguire una delle seguenti operazioni:
- Specificare l'angolo opposto del blocco di testo per definire la larghezza massima del testo multilinea.

- Se le **Quote dinamiche** sono attive, l'altezza e la larghezza del campo di testo vengono visualizzate dinamicamente. Digitare l'altezza e la larghezza nei campi delle quote dinamiche. Premere il tasto TAB per passare tra i campi. Premere Invio per confermare.
- Digitare @ nella barra dei comandi, quindi premere Invio per il testo non a capo (larghezza zero) (= larghezza della linea illimitata).

Sopra al testo verrà visualizzata la barra **Formattazione Testo**.

5 Digitare il tuo testo.

6 La larghezza massima della riga è definita dalla larghezza del blocco di testo come definito nel passaggio 3.

Premere Invio per iniziare un nuovo paragrafo.

7 Per concludere il comando e chiudere la barra di **Formattazione Testo** effettuare una delle seguenti operazioni:
- Cliccare sul pulsante **OK**.
- Premere Ctrl + Invio.
- Cliccare all'esterno dell'entità TestoM.

Barra degli Strumenti ed Impostazioni per la formattazione del testo

Icona o impostazione	Nome	Descrizione
Standard	Stile di Testo	Imposta lo stile di testo
[Arial]	Font del Testo	Imposta il font del testo
2.5 (Default)	Altezza Testo	Imposta l'altezza del testo
B	Grassetto	Crea un testo in grassetto
I	Italic	Crea un testo in corsivo
T	Sottilinea	Crea un testo sottolineato
T	Sopralineato	Creare un testo sopralineato
aA	Maiuscolo	Converte il testo selezionato in lettere maiuscole
Aa	Minuscolo	Converte il testo selezionato in lettere minuscole

Icona o impostazione	Nome	Descrizione
	Annotativo	Attiva o disattiva la proprietà annotativa
	Interlinea	Imposta l'interlinea
	Colonne	Dispone il testo in colonne
ByLayer	Colore	Imposta il colore
0	Angolo obliquo	Imposta l'angolo obliquo
1	Fattore larghezza	Imposta il fattore di larghezza
1	Spaziatura del Carattere	Imposta la spaziatura tra i caratteri
	Annulla	Annulla azioni recenti
	Ripristina	Ripristina azioni precedentemente annullate
	Frazione Impilata	Crea frazioni impilate
@	Carattere Speciale	Inserisce caratteri speciali
	Campo	Inserisce un campo variabile
	Giustificazione	Imposta la giustificazione del testo
	Maschera di Sfondo	Aggiunge uno sfondo colorato
	Righello	Attiva o disattiva la visualizzazione del righello
OK	OK	Chiude la barra degli strumenti **Formattazione Testo** ed applica le modifiche

Icona o impostazione	Nome	Descrizione
Cancel	Annulla	Chiude la barra **Formattazione Testo** e lascia invariata l'entità testo multilinea

Giustificazione testo multilinea

(Il punto rosso indica il punto di inserimento dell'entità testo multilinea).

This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multilin entity to illustrate the va justification settings t apply to multiline tex	This is a sample multi entity to illustrate the justification setti apply to multili
Alto Sinistra	Alto Centro	Alto Destra
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Centro-sinistra	Mezzo Centro	Mezzo Destra
This is a sample multiline text entity to illustrate the various justification settings that apply to multiline texts.	This is a sample multilin entity to illustrate the va justification settings t apply to multiline tex	This is a sample multi entity to illustrate the justification setti apply to multili
Basso Sinistra	Basso Centro	Basso Destra

14.2.3.3 Includere caratteri e simboli di testo speciali

Puoi usare codici di controllo e stringhe unicode per includere dei caratteri speciali nelle entità testo.

Per includere codici di controllo, durante l'inserimento del testo, digitare due simboli di percentuale (%%) seguiti dal codice di controllo o dal carattere speciale. Un singolo simbolo di percentuale viene trattato come un carattere di testo normale.

Caratteri di testo speciali

Codice di Controllo	Stringa Unicode	Risultato
%%d	\U+00B0	Disegna un simbolo "gradi" (°)
%%p	\U+00B1	Disegna un simbolo più-o-meno (±)

Codice di Controllo	Stringa Unicode	Risultato
%%c	\U+2205	Disegna un simbolo diametro (Ø)
%%k	\U+0336	Inizia a barrare il testo. Ripeti il codice per interromperlo.
%%u	\U+005F	Inizia a sottolineare il testo. Ripeti il codice per interromperlo.
%%o	\U+203E	Inizia a sovralineare il testo. Ripeti il codice per interromperlo.

Simboli e stringhe Unicode

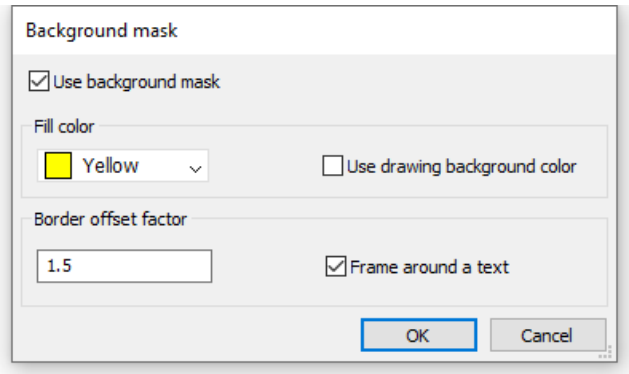
Nome	Simbolo (*)	Stringa Unicode
Quasi uguale	≈	\U+2248
Angolo	∠	\U+2220
Centro linea	℄	\U+2104
Delta	Δ	\U+0394
Fase elettrica	ϕ	\U+0278
Identità	≡	\U+2261
Non uguale	≠	\U+2260
Omega	Ω	\U+03A9
Quadrato	²	\U+00B2
Cubo	³	\U+00B3

(*) Se supportato dal font di testo.

14.2.3.4 Aggiungere una maschera di sfondo

1 Quando si crea o si modifica un'entità testo multilinea, cliccare sul pulsante dello strumento **Maschera di Sfondo** ( ) sulla barra degli strumenti **Formattazione Testo**.

Viene visualizzata la finestra di dialogo **Maschera di Sfondo**:



2 Selezionare l'opzione **Utilizza la maschera di sfondo**.

3 Eseguire una delle seguenti operazioni:
   - Scegliere un colore nell'elenco a discesa **Colore di riempimento**.
   - Scegliere **Seleziona colore...** nell'elenco a discesa **Colore di riempimento** e scegliere un colore nella finestra di dialogo **Seleziona Colore**.
   - Selezionare l'opzione **Usa colore di sfondo del disegno**.

4 Definire il **fattore offset dal bordo**, che imposta il margine attorno al testo per lo sfondo. Il valore è basato sull'altezza del testo e deve essere compreso tra 1 e 5. Un fattore 1.0 si adatta esattamente all'entità del testo multilinea.

5 (opzionale) Selezionare l'opzione **Cornice attorno al testo** per creare una cornice del testo.

6 Cliccare su **OK** per creare la maschera di sfondo.

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.

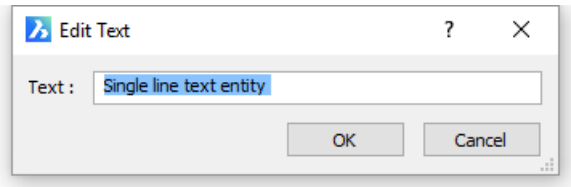
The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.

14.2.4.1 To edit a Text entity

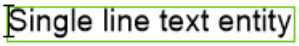
1 Double click the text entity.

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

Nota: To restore the built-in text editor, delete the content of the MTEXED system variable.

14.2.5 Esplodere il testo

Il comando ESPLTESTO esplode le entità di Testo e Testo Multilinea in polilinee 2D.

14.2.5.1 Esplodere il testo

1 Seleziona i testi

2 Eseguire una delle seguenti operazioni:
   - Scegliere **Esplodi** nella scheda **Edita** del **Quad**.
   - Avviare il comando ESPLTESTO.

Note:

- Il riempimento solido di alcuni font TTF viene rimosso una volta esploso il testo, lasciando il testo vuoto (solo bordi).
- Utilizzare il comando UNISCIper creare polilinee chiuse dai contorni del testo esploso.
- Lo strumento **Esplodi** nella scheda **Edita** del **Quad** lancia il comando ESPLTESTO. Il comando ESPLODI non può esplodere le entità di testo.

14.2.6 Trovare e sostituire un testo

Il comando TROVA consente di specificare il testo che si desidera cercare. È possibile trovare o sostituire il testo. Facoltativamente, è possibile eseguire lo zoom sulle varie istanze delle entità testo contenenti il testo ricercato.

14.2.6.1 Impostare le opzioni Trova e Sostituisci

1 Avviare il comando TROVA.

1 Viene visualizzata la finestra di dialogo **Trova e Sostituisci**.

2 Fare clic sul pulsante **Opzioni**.

3 Selezionare le opzioni nella finestra di dialogo **Opzioni Trova e Sostituisci**.

4 Cliccare sul pulsante **OK**.

14.2.6.2 Cercare un testo

1 Avviare il comando TROVA.

1 Viene visualizzata la finestra di dialogo **Trova e Sostituisci**.

2 (opzionale) Impostare le opzioni **Trova e Sostituisci**.

3 (opzione) Cliccare il pulsante **Seleziona Entità** ( ) per comporre un'insieme di selezione.

   Se non è stata selezionata alcuna entità, viene eseguita la ricerca nell'intero disegno.
   - La finestra **Trova e Sostituisci** verrà chiusa temporaneamente per permettere la selezione delle entità.
   - Cliccare il pulsante destro del mouse per concludere la selezione delle entità.

   Il campo **Cerca in** riporta **Selezione corrente** anziché **Disegno intero**.

4 Digitare il testo nel campo **Trova Stringa Testo**.

5 Cliccare sul pulsante **Trova**.

6 Il contenuto della prima entità di testo contenente la stringa di ricerca viene visualizzato nel riquadro **Testo Trovato**.

7 (opzione) Fare clic sul pulsante **Zoom In**.

Il disegno viene ingrandito sull'entità testo.

8 Eseguire una delle seguenti operazioni:

9 Ripetere i passaggi da 5 e 6 per trovare più istanze della stessa stringa di ricerca.

10 Ripetere i passaggi da 2 fino a 6 per trovare un'altra stringa di testo.

11 Premere il tasto Esc o fare clic sul pulsante **Chiudi** per interrompere.

14.2.6.3 Sostituire un testo

1 Avviare il comando TROVA.

1 Viene visualizzata la finestra di dialogo **Trova e Sostituisci**.

2 (opzionale) Impostare le opzioni **Trova e Sostituisci**.

3 (opzione) Cliccare il pulsante **Seleziona Entità** ( ) per comporre un'insieme di selezione.

Se non è stata selezionata alcuna entità, viene eseguita la ricerca nell'intero disegno.

- La finestra **Trova e Sostituisci** verrà chiusa temporaneamente per permettere la selezione delle entità.
- Cliccare il pulsante destro del mouse per concludere la selezione delle entità.

Il campo **Cerca in** riporta **Selezione corrente** anziché **Disegno intero**.

4 Digitare il testo nel campo **Trova Stringa Testo**.

5 Digitare il testo sostitutivo nel campo **Sostituisci con**, quindi premere Invio.

Vengono elencate tutte le entità di testo contenenti la stringa di ricerca.

6 Eseguire una delle seguenti operazioni:

- Cliccare su un elemento nell'elenco per evidenziare l'entità di testo nel disegno, quindi cliccare sul pulsante **Sostituisci**.
- Fare clic sul pulsante **Sostituisci**.

14.2.7 Controllo ortografia

Il correttore ortografico controlla l'ortografia del testo nel disegno corrente. Puoi controllare l'ortografia del testo su una o più entità selezionate o nell'intero disegno.

Durante il controllo ortografico, il comando ORTOGRAF verifica le parole presenti nel disegno o nell'insieme di selezione con quelle presenti nel dizionario principale corrente e nel dizionario personalizzato corrente. Se una parola non viene trovata in nessuno dei due dizionari è possibile selezionare l'ortografia corretta nel campo suggerimenti o aggiungere la parola al dizionario personale corrente. Il dizionario personalizzato può essere utilizzato per parole specifiche e tecniche, ad esempio parole specifiche di medicina o meccanica.

14.2.7.1 Impostazioni dizionario

I dizionari utilizzati dal comando ORTOGRAF sono definiti dalle variabili di sistema DCTMAIN (dizionario principale) e DCTCUST(dizionario personalizzato).

Nota: Installiamo solo il dizionario inglese in quanto la versione inglese può essere distribuita liberamente, mentre tutte le altre lingue non sono gratuite. Tuttavia, puoi scaricare altre lingue e utilizzarle gratuitamente (vedi Aggiungere un dizionario principale di seguito).

14.2.7.2 Aaprire la finestra di dialogo Controllo Ortografia

Eseguire una delle seguenti operazioni:

- Scegliere **Ortografia** nel menu **Strumenti**.
- Cliccare sullo strumento **Controllo Ortografia** nel pannello **Annotazione/Testo** della barra multifunzione.
- Avviare il comando ORTOGRAF.

14.2.7.3 Controllare l'ortografia

1 Aprire la finestra di dialogo **Controllo Ortografia**.

2 (opzionale) Cliccare l'icona **Seleziona entità** ( ) per selezionare un gruppo di entità.

Se non è stata selezionata alcuna entità, viene eseguita la ricerca nell'intero disegno.

- La finestra di dialogo **Controllo Ortografia** si chiude temporaneamente per consentire la selezione delle entità.
- Cliccare il pulsante destro del mouse per concludere la selezione delle entità.

Il campo **Dove controllare** riporta **Selezione corrente** anziché **Intero disegno**.

3 (opzionale) Cambiare il dizionario corrente (vedi sotto).

4 Cliccare sul pulsante **Avvia**.

La prima parola con errori di ortografia viene visualizzata nella sezione **Parola corrente**.

5 Il contenuto dell'entità di testo in cui viene trovata **la parola corrente** viene visualizzato nel campo **Contesto**.

6 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Ignora** per lasciare invariata questa istanza della **parola corrente**.
- Cliccare sul pulsante **Ignora Tutto** per lasciare invariate tutte le istanze della **parola corrente**.
- Accettare la parola nel campo di **Suggerimenti** o selezionare una parola nell'elenco **Suggerimenti**, quindi cliccare sul pulsante **Cambia** per sostituire questa istanza della **parola corrente** con la parola del campo **Suggerimenti**.
- Accettare la parola nel campo **Suggerimenti** o selezionare una parola nell'elenco **Suggerimenti**, quindi cliccare sul pulsante **Cambia Tutto** per sostituire tutte le istanze della **parola corrente** con la parola del campo **Suggerimenti**.
- Cliccare sul pulsante **Aggiungi** per aggiungere la **parola corrente** al dizionario personalizzato corrente.

- Cliccare sul pulsante **Ricerca** per elencare le parole simili alla parola nel campo **Suggerimenti**.

7 Ripetere il passaggio precedente fino a quando non viene visualizzato il messaggio **Il controllo ortografico è stato completato**.

14.2.7.4 Cambiare i dizionari

1 Aprire la finestra di dialogo **Controllo Ortografia**.

2 Cliccare sul pulsante **Cambia Dizionari ...**.

Viene visualizzata la finestra di dialogo **Cambia Dizionari**.

3 Espandere l'elenco **Dizionario Principale** in modo da cambiare il dizionario principale.

4 Selezionare un nuovo dizionario principale.

5 (opzionale) Cliccare il bottone **sfoglia...** per selezionare un nuovo **Dizionario personalizzato**.

6 (opzionale) Digitare una parola nel campo **Personalizza parole dizionario**, e successivamente cliccare il bottone **Aggiungi** per aggiungere una parola al dizionario personalizzato corrente.

7 Cliccare il pulsante **Applica e Chiudi** per confermare.

Note:

- I dizionari personalizzati vengono salvati nella cartella **Support** della **cartella principale di installazione**.
- Se si desidera condividere il dizionario personalizzato con MS Word, fare riferimento al dizionario personalizzato MS Word al punto 5 della procedura precedente. Per impostazione predefinita, CUSTOM.DIC è il dizionario personalizzato in MS Word, disponibile in *C:\<\Users>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC*.

14.2.7.5 Aggiungere un nuovo dizionario principale

1 Aprire la finestra di dialogo **Controllo Ortografia**.

2 Cliccare sul pulsante **Cambia Dizionari ...**.

3 Cliccare sul pulsante di **Scarica...**.

Si aprirà il browser: https://extensions.openoffice.org/dictionaries

4 Scorrere fino alla lingua scelta, quindi cliccare sul collegamento per aprire la pagina della lingua selezionata.

5 Cliccare sul pulsante **Download extension** per iniziare a scaricare il file ***.oxt**.

6 Modificare l'estensione **oxt** in **zip**.

7 Estrarre i file ***.aff** e ***.dic** dal file ZIP nella cartella **C:\Program Files\Bricsys\BricsCAD Vx\Support**.

Il dizionario è ora disponibile nella lista **Dizionario Principale** sulla finestra di dialogo **Cambia Dizionari**.

14.2.7.6 Aggiungere un nuovo dizionario personalizzato

1 Aprire l'editor di testo, ad esempio **Blocco Note**.

2 (opzionale) Digitare le parole che si desidera utilizzare nel dizionario personalizzato.

3 Ogni parola deve essere posizionata su una riga separata.

4 Salvare il file nella cartella **Support** della cartella principale **Roamable** utilizzando l'estensione file **.cus**.

14.2.8 Lavorare con i Campi

Un campo è un'entità di testo che contiene istruzioni per visualizzare lo stato o la proprietà di un'altra entità o impostazione nel disegno. Quando un campo viene aggiornato viene visualizzato l'ultimo valore della sorgente di dati a cui si riferisce. I campi possono essere usati per inserire proprietà del disegno, proprietà personalizzate o proprietà di un oggetto all'interno di testi multilinea o tabelle o come attributi di un blocco. I campi negli attributi sono particolarmente utili nei cartigli: utilizzando il disegno, gruppo di fogli, sottogruppo e le proprietà del foglio i cartigli possono essere compilati automaticamente.

Vedere il comando PROPDIS per saperne di più circa le proprietà predefinite e personalizzate del disegno.

Vedere il comando GRUPPOFOGLI per saperne di più circa i gruppi di fogli predefiniti e personalizzati, proprietà di sottoinsiemi e del foglio.

Se nessun valore è disponibile, un campo visualizza dei trattini (----).

Se la variabile di sistema FIELDDISPLAY è ON, i campi verranno visualizzati con uno sfondo grigio, il quale non verrà stampato.

La variabile di sistema FIELDEVAL controlla quando i campi devono essere aggiornati.

14.2.8.1 Procedura generale per creare i campi

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Inserisci campo** ( ) sulla barra degli strumenti **Inserisci** e sul pannello **Annotazione/Testo** della barra multifunzione.
- Scegliere **Inserisci Campo...** nel menu **Inserisci**.
- Avviare il comando CAMPODATI.
- Cliccare sul pulsante **Campo** ( ) sulla barra degli strumenti **Formattazione Testo** durante la creazione o la modifica di un Testo Multilinea o del contenuto di una cella di una tabella.

Viene visualizzata la finestra di dialogo **Campo**.

2 In **Nomi campo** espandere un gruppo, quindi scegliere un nome di campo.

Appaiono le impostazioni del nome del campo selezionato.

3 Regolare le impostazioni, quindi cliccare sul pulsante **OK**.

La finestra di dialogo **Campo** si chiude.

Il campo viene creato nel Testo Multilinea o nella cella della tabella in corso di modifica.

o

Viene richiesto: Specificare punto iniziale o [Altezza/Giustificazione]:

Eseguire le seguenti operazioni:

- (opzionale) Digitare **A** o scegliere **Altezza** nel menu contestuale per definire l'altezza del testo.
- (opzionale) Digitare **G** o scegliere **Giustificazione** nel menu contestuale per impostare la

giustificazione del testo.

- Specificare il punto di inserimento del campo nel disegno.

Viene creata un'entità Testo Multilinea contenente il campo.

14.2.8.2 Visualizzare una proprietà di un'entità in un campo

1 Avviare il comando CAMPODATI.

1 Viene visualizzata la finestra di dialogo **Campo**.

2 Espandere **Oggetti** nell'elenco **Nomi campi** e selezionare **Oggetto**.

3 Cliccare sul pulsante **Seleziona Entità** nella finestra di dialogo **Campo**.

La finestra di dialogo **Campo** si chiude temporaneamente.

Viene richiesto: Seleziona entità:

4 Cliccare sull'entità di cui si desidera visualizzare una proprietà.

5 Viene nuovamente visualizzata la finestra di dialogo **Campo**.

6 Selezionare la proprietà dell'entità nell'elenco **Proprietà**.

7 Impostare il layout di visualizzazione:

- **Formato**: scegliere un formato di unità.
- **Precisione**: selezionare una precisione nel pulsante elenco
- **Converti**: questa opzione consente di ricalcolare il valore del campo.
  - Scegliere una formula nell'elenco; ad es. f*#, dove f è il fattore che si digita nel campo **Converti** e # è il valore del campo.
  - Digitare un fattore nel campo **Converti**, ad es. 0.0001 per esprimere l'area di una polilinea in metri quadrati in un disegno in cm.

8 Cliccare sul pulsante **OK**.

9 Specificare un punto per inserire il campo nel disegno.

14.2.8.3 Modificare un campo

1 Doppio clic sulla cella di tabella o entità Testo Multilinea che contiene il campo.

2 Doppio clic sul campo.

3 Viene visualizzata la finestra di dialogo **Campo**.

4 Regolare le proprietà del campo.

5 Cliccare sul pulsante **OK** per applicare le modifiche.

14.2.8.4 Aggiornare un campo

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante **Aggiorna Campi...** sulla barra degli strumenti **Ridisegna / Rigenera**.
- Scegliere **Aggiorna Campi...** nel menu **Strumenti**.
- Avviare il comando AGGCAMPODATI.

Viene richiesto: Seleziona campi da aggiornare [opzioni di selezione (?)]:

2 Selezionare i campi che si desideri aggiornare, quindi cliccare il pulsante destro del mouse o premere Invio.

3 I campi selezionati vengono aggiornati.

Nota: Se è impostato il codice bit 16 della variabile di sistema FIELDEVAL, l'esecuzione del comando RIGEN aggiorna tutti i campi nel disegno.

14.2.8.5 Convertire un campo di testo

1 Doppio clic sulla cella di tabella o entità Testo Multilinea che contiene il campo.

2 Selezionare il campo.

3 Cliccare il pulsante destro del mouse, quindi selezionare **Converti Campo in Testo** nel menu contestuale.

14.2.9 Lavorare con le tabelle

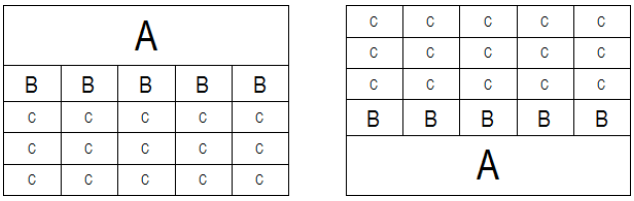
Una tabella è un'entità composta da righe e colonne, proprio come un foglio di calcolo Microsoft Excel. Un'entità tabella può essere creata come una tabella vuota oppure è possibile importare dati da un file CSV (Comma Separated Values). L'aspetto del testo della tabella e della griglia è controllato da uno stile di tabella. Gli stili tabella vengono salvati nel file DWG.

Una tabella predefinita è composta da:

- una cella **Titolo** (A)
- una riga di celle di **intestazione** (B)
- una o più righe di celle **Dati** (C).

Una tabella può avere una delle seguenti due direzioni:

- **Giù** (riga del titolo in alto)
- **Su** (riga del titolo in basso).



14.2.9.1 Creare una tabella vuota

1 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Tabella** ( ) sulla barra degli strumenti **Disegno** o sul pannello **Inizio/Annotazioni** della barra multifunzione.
- Scegliere **Tabella** nel menu **Disegna**.
- Avviare il comando TABELLA.

Viene visualizzata la finestra di dialogo **Inserisci Tabella**.

2 Scegliere uno Stile Tabella dalla lista **Stile Tabella**.

3 (opzionale) Impostare lo stile della cella per la prima, la seconda e le altre righe.

4 Per impostazione predefinita, lo stile della cella per la prima riga è **Titolo**, lo stile della cella per la seconda riga è **Intestazione** e lo stile della cella per le altre righe è **Dati**.

5 In **Opzioni tabella**, scegliere **Inizia da una Tabella Vuota**.

6 Sotto **Modalità di Inserimento**, eseguire una delle seguenti operazioni:
   - Scegliere **Specificare il Punto di Inserimento**, quindi specificare il numero di colonne, la larghezza delle colonna, il numero di righe per i dati e l'altezza delle righe.
   - Scegliere **Specificare Finestra**, quindi eseguire una delle seguenti operazioni:
   - Specificare il numero di colonne e righe di dati.
   - Specificare la larghezza della colonna e l'altezza della riga.
   - Specificare la larghezza della colonna ed il numero di righe di dati.
   - Specificare il numero di colonne e l'altezza delle righe.

7 Cliccare sul pulsante **OK**.

   A seconda della scelta nel passaggio 5, **viene richiesto** di specificare un punto di inserimento o una finestra.

   Viene creata la tabella e viene visualizzata la barra degli strumenti **Formattazione Testo** (vedere Creazione di testi).

8 Eseguire una delle seguenti operazioni:
   - Iniziare ad aggiungere testo nella tabella.
   - Cliccare all'esterno della tabella per chiudere la barra degli strumenti **Formattazione Testo**.

14.2.9.2 Creare una tabella di dati

Quando si importano i dati da un file CSV (Comma Separated Values), assicurarsi che il carattere separatore di elenco sul vostro sistema sia uguale a quello utilizzato nel file CSV (vedi Definire il carattere separatore di elenco).

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante dello strumento **Tabella** ( ) sulla barra degli strumenti **Disegno** o sul pannello **Inizio/Annotazioni** della barra multifunzione.
   - Scegliere **Tabella** nel menu **Disegna**.
   - Avviare il comando TABELLA.

   Viene visualizzata la finestra di dialogo **Inserisci Tabella**.

2 Sotto **Opzioni Tabella**, scegliere **Da Dati**.

3 Fare clic sul pulsante **Sfoglia** ().

4 Viene visualizzata la finestra di dialogo **Seleziona file di origine**.

   Scegliere un file CSV, quindi cliccare sul pulsante **Apri**.

5 Cliccare sul pulsante **OK** nella finestra di dialogo **Inserisci Tabella**.

6 Cliccare su un punto nel disegno per posizionare la tabella.

7 Viene creata la tabella e viene visualizzata la barra degli strumenti **Formattazione Testo** (vedere

Creazione di testi).

8 Eseguire una delle seguenti operazioni:
   - Iniziare a modificare il testo nella tabella.
   - Cliccare all'esterno della tabella per chiudere la barra degli strumenti **Formattazione Testo**.

9 (opzionale) Selezionare la tabella e trascinare i grip per regolare le dimensioni della tabella.

14.2.9.3 Creare una tabella da un foglio Excel

1 Aprire il foglio di calcolo in **Microsoft Office Excel**.

2 Selezionare le celle da cui si desidera creare una tabella.

3 Per copiare la selezione negli appunti, effettuate una delle seguenti operazioni:
   - Premere **Ctrl + C**.
   - Cliccare il pulsante destro del mouse, quindi selezionare **Copia** nel menu contestuale.

4 In BricsCAD, scegliere **Incolla Speciale...** nel menu **Modifica** o avviare il comando INCOLLASPEC.

5 Viene visualizzata la finestra di dialogo **Incolla Speciale**.

6 Scegliere **Foglio di calcolo XML** nell'elenco delle opzioni **Come**.

7 Cliccare nel disegno per inserire la tabella.

14.2.9.4 Modificare una tabella utilizzando i grip

Quando viene selezionata una tabella, vengono visualizzate più grips:

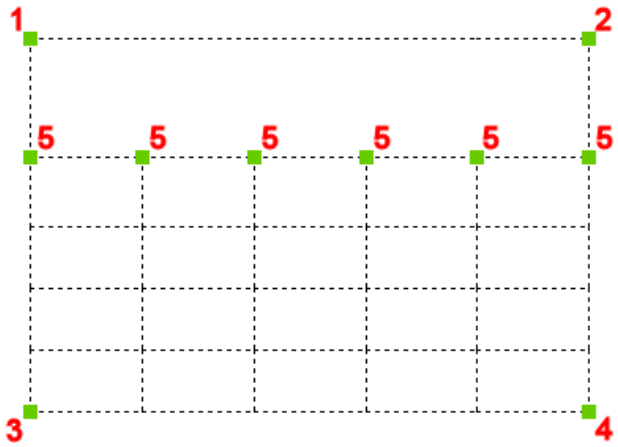
1 Sposta la tabella.

2 Modifica la larghezza in modo uniforme.

3 Modifica l'altezza in modo uniforme.

4 Modifica la larghezza e l'altezza in modo uniforme.

5 Modifica la larghezza di una colonna.



14.2.9.5 Modificare una tabella utilizzando la barra degli strumenti Tabella

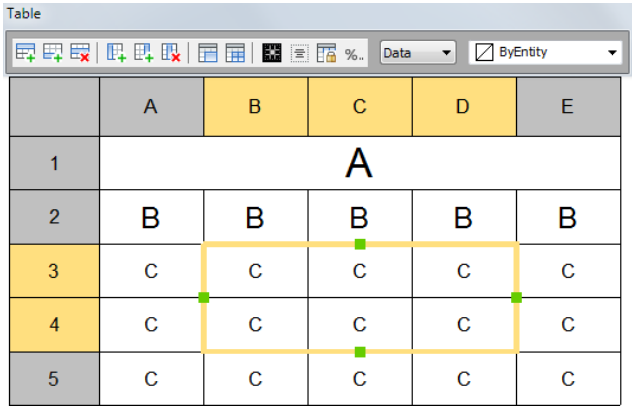
1 Per aprire la barra degli strumenti **Tabella** effettuare una delle seguenti operazioni:
   - Cliccare sulla cella che si desidera modificare.

- Premere e tenere premuto il pulsante sinistro del mouse, quindi definire un rettangolo per selezionare più celle.

2 Vengono visualizzate le lettere delle colonne e i numeri delle righe.

3 Le colonne e le righe della selezione sono evidenziate.

Intorno alla selezione verrà disegnato un rettangolo con quattro grips.



4 (opzionale) Trascinare i grip della selezione per modificare le dimensioni.

5 (opzionale) Cliccare su uno strumento nella barra degli strumenti **Tabella**.

6 (opzionale) Cliccare il pulsante destro del mouse per aprire un menu contestuale:
- **Elimina contenuto della cella**: elimina il testo nelle celle selezionate.
- **Pulisci modifiche locali cella**: reimposta tutte le proprietà delle celle selezionate con le impostazioni dello **Stile Cella** come definito nello **Stile Tabella**.
- **Pulisci modifiche locali tabella**: reimposta tutte le proprietà di tutte le celle con le impostazioni dello **Stile Cella** come definito nello **Stile Tabella**.

7 (opzionale) Ripetere i passaggi 3 e 4 per continuare a modificare la tabella.

8 (opzionale) Doppio clic su una cella per modificare il testo della cella.

9 (opzionale) Eseguire una delle seguenti operazioni:
- Cliccare su un'altra cella.
- Tenere premuto il tasto sinistro del mouse, quindi trascinare il cursore per selezionare più celle.

10 Cliccare all'esterno della tabella per chiudere la barra degli strumenti **Tabella**.

Strumenti Tabella

Icona	Nome strumento	Descrizione
	Inserisci Riga Sopra	Inserisce una riga vuota sopra la selezione.
	Inserisci Riga Sotto	Inserisce una riga vuota sotto la selezione.

Icona	Nome strumento	Descrizione
	Elimina Riga	Elimina le righe selezionate.
	Inserisci Colonna a Sinistra	Inserisce una colonna vuota a sinistra della selezione.
	Inserisci Colonna a Sestra	Inserisce una colonna vuota a destra della selezione.
	Elimina colonna	Elimina le colonne selezionate.
	Unisci	Unisce le celle selezionate.
	Separa Celle	Divide celle precedentemente unite.
	Stile Bordo	Imposta lo stile del bordo della selezione (non ancora implementato).
	Allineamento testo	Consente di impostare l'allineamento del testo della selezione.
	Blocca / Sblocca	Blocca / Sblocca la selezione.
%..	Imposta Formato Cella	Imposta il formato delle celle selezionate (non ancora implementato).
Title	Imposta Stile Cella	Imposta lo stile delle celle selezionate: **Titolo**, **Intestazione** o **Dati**.
ByEntity	Imposta sfondo cella	Imposta il colore di sfondo della selezione.

14.2.9.6 Modificare una tabella nel pannello Proprietà

1 Cliccare su una delle linee della griglia.

1 Le proprietà della tabella vengono visualizzate nel pannello **Proprietà**.

2 (opzionale) Applicare un diverso **Stile Tabella**.

3 (opzionale) Modificare la direzione della tabella.

4 (opzionale) Modificare la larghezza e/o l'altezza della tabella.

5 (opzionale) Modificare il margine verticale e/o orizzontale delle celle.

6 Premere Esc per interrompere le modifiche.

14.2.9.7 Modificare una cella nella barra delle Proprietà

1 Cliccare sulla cella.

1 Le proprietà della cella vengono visualizzate nel pannello **Proprietà**.

2 (opzionale) Modificare la larghezza e/o l'altezza della tabella.

3 La larghezza e l'altezza della colonna e/o della riga della cella vengono modificate.

4 (opzionale) Modificare la rotazione del testo.

5 Le opzioni sono: 0°, 90°, 180° e 270°.

6 (opzionale) Modificare il contenuto della cella.

7 (opzionale) Scegliere una diversa opzione di allineamento del testo.

8 Premere Esc per interrompere le modifiche.

14.2.9.8 Valutare le formule

Attualmente le formule sono supportate solo parzialmente. Esistono formule più complesse che non sono ancora supportate, ma nella maggior parte dei casi le formule nei fogli Excel rimarranno funzionanti quando il foglio Excel viene importato come tabella. La modifica e la creazione di formule tramite la finestra di dialogo Campo non è ancora supportata, ma è possibile modificarle/crearle sul posto. Ad esempio: se si digita =SUM(A2:A4) in una cella, verrà interpretato come una formula e verrà calcolata la somma del valore nelle celle dalla A2 alla A4.

14.2.9.9 Modificare il contenuto della tabella

1 Eseguire una delle seguenti operazioni:
- Doppio clic sulla cella che si desidera modificare.
- Avviare il comando MODIFTABELLA.
- **Viene richiesto**: Selezionare una cella di una tabella da modificare.

  Cliccare sulla cella che si desidera modificare.

La barra degli strumenti **Formattazione Testo** viene visualizzata sopra la cella selezionata (vedere Creazione di testi).

2 Modificare la cella.

3 (opzionale) Premere Alt + Invio per creare una seconda linea nella stessa cella.

4 Modificare un'altra cella:
- Premere il tasto Tab per passare alla cella successiva nella stessa riga.
- Premere Maiusc + Tab per passare alla cella precedente nella stessa riga.
- Premere Invio per passare alla cella successiva nella stessa colonna.
- Premere Maiusc + Invio per passare alla cella precedente nella stessa colonna.
- Premere uno dei tasti freccia.

5 Cliccare all'esterno della tabella per interrompere la modifica.

14.2.9.10 Esportare una tabella

Durante l'esportazione di dati in un file CSV (Comma Separated Values), viene utilizzato il carattere separatore di elenco definito nel tuo sistema (vedi Definizione del carattere separatore di elenco).

1 Avviare il comando ESPORTATABELLA.

1 **Viene richiesto**: Seleziona Tabella da esportare:

2 Cliccare sulla tabella di cui si desidera esportare i dati.

3 Viene visualizzata la finestra di dialogo **Esporta Dati**.

4 Selezionare una cartella per il salvataggio del file di dati.

5 Digitare un nome nel campo **Nome file**.

6 Cliccare sul pulsante **Salva**.

I dati nella tabella selezionata vengono salvati come file CSV.

14.2.9.11 Impostare il carattere separatore elenco

1 Aprire la finestra di dialogo **Impostazioni** di Windows.

2 Aprire la categoria delle impostazioni **Area Geografica**.

3 Cliccare su **Data e ora**.

4 Cliccare su **Cambia i formati di data e ora**.

5 Cliccare su **Impostazioni aggiuntive**.

6 Imposta il carattere **separatore di elenco**.

7 Cliccare su **OK** per confermare.

14.3 Annotative Entities

14.3.1 Entità Annotative

Le annotazioni sono tutte le entità del disegno che non hanno una geometria grafica, come le quote, simboli, testi, ecc. Un problema comune alle annotazioni è che spesso sono dipendenti dalla scala di stampa. Alcune annotazioni devono essere scalate per la scala di stampa, o sono destinate ad essere visualizzate solo con alcune scale di stampa.

Gli stili e le entità annotative modificano il modo in cui vengono visualizzati (dimensione, posizione e orientamento) in base al fattore di scala corrente ed alla finestra dello spazio carta.

Le entità che possono avere la proprietà annotativa sono le seguenti:

- testo e testo multilinea
- tratteggi
- blocchi e attributi
- quote e tolleranze
- direttrici e multidirettrici

Gli stili che possono avere la proprietà annotativa sono i seguenti:

- stili di testo (vedi il comando STILE)
- stili di quota (consultare il comando DIMSTILE)
- stili multidirettrice (vedi il comando STILEMULTIDIR)

Note:

- Entità create utilizzando uno stile annotativo, ottengono automaticamente la scala annotazione corrente.
- Quando uno stile annotativo viene applicato ad un'entità non annotativa, la proprietà annotativa non viene assegnata automaticamente.

14.3.1.1 Comandi

RIPRISTANN: ripristina tutte le rappresentazioni in scala delle entità annotative selezionate alla posizione della loro attuale rappresentazione in scala, quindi sincronizza tutte le rappresentazioni in scala.

AGGIORNANNOT: aggiorna le entità annotative selezionate per abbinarle alla loro attuale definizione di stile.

SCALAOGG: aggiunge o rimuove le scale supportate per le entità annotative attraverso una finestra di dialogo.

-SCALAOGG: aggiunge o rimuove le scale supportate per le entità annotative dalla barra dei comandi.

MODIFELENCOSCALE: modifica l'elenco delle scale di un disegno, utilizzato nella sezione **Scala di stampa** delle finestre di dialogo **Stampa** e **Impostazioni Pagina** e la proprietà **Scala standard** di una finestra dello spazio carta. Definisce le scale annotazione disponibili per le entità annotative.

-MODIFELENCOSCALE: modifica l'elenco delle scale di un disegno dalla barra dei comandi.

14.3.1.2 Variabili di sistema

ANNOALLVISIBLE: Attiva o disattiva la visualizzazione delle entità annotative che non supportano la scala annotazione corrente.

ANNOTATIVEDWG: Specifica se il disegno deve comportarsi come un blocco annotativo quando inserito in un altro disegno.

CANNOSCALE: Imposta il nome della scala annotazione corrente per lo spazio corrente. Sono accettate sola le scale esistenti nell'attuale elenco scale. L'impostazione CANNOSCALE viene salvata nello spazio modello ed in ogni finestra di layout dei vari layout. Quando si crea un'entità annotativa la scala annotativa corrente viene applicata automaticamente.

CANNOSCALEVALUE: Visualizza il valore della scala annotazione corrente. Ad esempio, il valore della scala 1:2 è 0.5.

HPANNOTATIVE: Controlla se le nuove entità tratteggio sono annotative.

MSLTSCALE: Se ON, i tipolinea nello spazio modello vengono scalati alla scala annotazione corrente.
E' necessaria una rigenerazione quando si attiva il valore della variabile MSLTSCALE. Questa variabile è l'equivalente della variabile PSLTSCALE (scala del tipolinea dello spazio carta) nello spazio carta.

MSOLESCALE: Controlla la dimensione di un oggetto OLE contenente testo quando incollato nello spazio modello. La variabile di sistema MSOLESCALE riguarda solo l'adattamento iniziale. Gli oggetti OLE esistenti non sono interessati se la variabile viene modificata.

SAVEFIDELITY: Controlla se la visualizzazione corrente del disegno rimane conservata quando viene aperto in un programma che non supporta le entità annotative.

SELECTIONANNODISPLAY: Controlla se tutte le scale di rappresentazione vengono visualizzate quando viene selezionata un'entità annotativa.

14.3.1.3 Impostare la scala annotazione corrente

Per impostare la scala annotazione corrente, eseguire una delle seguenti procedure:

- Cliccare sul campo **Scala annotazione** sotto **Varie** nel pannello **Proprietà**, quindi premere il pulsante freccia giù e selezionare una scala nell'elenco.
- Cliccare il pulsante destro del mouse sul campo **Scala Annotazione** nella **Barra di Stato**, quindi selezionare una scala nell'elenco. Se il campo **Scala Annotazione** non è disponibile, cliccare sulla freccia verso il basso sul lato destro della **Barra di Stato** e scegliere **Scala Annotazione** nell'elenco.

Nota: Sono disponibili solo le scale presenti nell'elenco **Scale** del disegno. Utilizzare il comando MODIFELENCOSCALE per modificare l'elenco **Scale**.

14.3.1.4 Rendere annotativa un'entità annotazione

Le entità annotazione che vengono create utilizzando uno stile non annotativo possono essere rese annotative utilizzando la seguente procedura:

1. Selezionare l'entità annotazione.
2. Nella pannello **Proprietà**, cliccare sulla proprietà **Annotativo**.
3. Selezionare **Sì**.

   La scala annotazione corrente viene applicata all'entità selezionata.

14.3.2 Scala annotazione

Le scale annotazione vengono utilizzate per calcolare il fattore di scala delle entità annotative. Esse definiscono l'altezza delle entità testo annotativo od il ridimensionamento complessivo dei blocchi annotativi. In generale la scala annotazione permette di creare entità annotative nello spazio modello per essere ridimensionate automaticamente in modo corretto nello spazio carta.

- Testi, MultiTesti, Quote, Multidirettrici: La scala annotazione corrente viene applicata al momento della creazione, se lo stile corrente è annotativo.
- Tratteggi e Riempimenti Sfumati: la scala annotazione corrente viene applicata se la variabile di sistema HPANNOTATIVE è ON.
- Blocchi: La scala di annotazione corrente definisce la scalatura di blocchi annotativi durante l'inserimento.

Le entità annotative possono supportare più scale annotazione. Quando la variabile di sistema SELECTIONPREVIEW è attiva o quando si utilizza il menu del cursore **Quad**, viene visualizzata un'icona

accanto all'entità evidenziata, a indicare se una singola scala annotazione ( ) si applica a una o più entità annotative( ).

14.3.2.1 Aggiungere o rimuovere scale annotazione

La seguente procedura consente di:

- Verificare quali scale annotazione sono attualmente applicate.
- Aggiungere scale annotazione.
- Rimuovere scale annotazione.

1 Selezionare l'entità o le entità annotative.

2 Nel pannello **Proprietà**, selezionare il campo **Scala annotativa**.

3 Cliccare sul pulsante **Sfoglia** ( ) del campo **Scala annotativa**.

Viene visualizzata la finestra di dialogo **Scala Oggetto Annotazione**.

4 (opzionale) Se sono state selezionate più entità, scegliere se elencare tutte le scale della selezione o solo le scale in comune.

5 (opzionale) Eseguire una delle seguenti operazioni:

- Selezionare una o più scale nell'elenco, quindi cliccare sul pulsante **Elimina**. Le scale selezionate vengono rimosse dall'elenco.
- Cliccare sul pulsante **Aggiungi...**, quindi selezionare una o più scale nell'elenco e cliccare sul pulsante **OK**.

  Le scale selezionate vengono aggiunte all'elenco.

6 Cliccare sul pulsante **OK** per chiudere la finestra di dialogo **Scala Oggetto Annotazione**.

Le modifiche vengono applicate al gruppo di selezione.

Nota: Sono disponibili solo le scale presenti nell'elenco **Scale** del disegno. Utilizzare il comando MODIFELENCOSCALE per modificare l'elenco **Scale**.

14.3.3 Testi Annotativi

Le entità testo vengono create utilizzando l'attuale stile di testo:

- Se lo stile di testo è annotativo, l'entità testo ottiene la proprietà Annotativo automaticamente e viene assegnata la scala annotazione corrente.
- Se lo stile di testo non è annotativo, è necessario rendere l'entità del testo annotativa manualmente.

I testi annotativi hanno due proprietà di altezza:

- **Altezza testo su modello**: l'altezza di visualizzazione sullo schermo nello spazio modello, per quanto riguarda la scala di annotazione corrente.
- **Altezza testo su foglio**: l'altezza sulla carta se stampato con la scala annotazione supportata.

14.3.3.1 Rendere annotativa un'entità testo

1 Selezionare l'entità testo.

1 Le proprietà dell'entità testo selezionata vengono visualizzate nel pannello **Proprietà**.

2 Nella pannello **Proprietà**, cliccare sulla proprietà **Annotativo**.

3 Scegliere **Sì** dall'elenco delle opzioni.

È stata assegnata la scala di annotazione corrente.

Il testo non è scalato: l'altezza attuale del testo diventa l'**Altezza testo del modello**.

14.3.3.2 Rimuovere la proprietà annotativa

1 Selezionare l'entità testo.

1 Le proprietà dell'entità testo selezionata vengono visualizzate nel pannello **Proprietà**.

2 Nella pannello **Proprietà**, cliccare sulla proprietà **Annotativo**.

3 Scegliere **No** nell'elenco delle opzioni.

Se l'entità di testo supporta più scale annotazione, il testo viene ridimensionato alla sua scala iniziale.

14.3.4 Tratteggi e riempimenti sfumati annotativi

La variabile di sistema HPANNOTATIVE controlla se le nuove entità tratteggio o riempimento sfumato sono annotative.

HPANNOTATIVE è disattivato per impostazione predefinita. La variabile di sistema non viene salvata e viene reimpostata sul valore predefinito quando chiude BricsCAD.

14.3.4.1 Impostare la variabile di sistema HPANNOTATIVE

Eseguire una delle seguenti operazioni:

- Nella finestra di dialogo **Impostazioni**, sotto **Disegno** > **Disegnare** > **Creazione entità** > **Tratteggi**.
- Selezionare l'opzione **Annotativo** nella finestra di dialogo **Tratteggio e Sfumatura**.
- Scegliere l'opzione **Annotativo** nella barra dei comandi del comando -TRATTEGGIO.

14.3.4.2 Rendere annotativo un'entità tratteggio o sfumatura

1 Selezionare il tratteggio o l'entità di riempimento sfumato.

1 Le proprietà dell'entità testo selezionata vengono visualizzate nel pannello **Proprietà**.

2 Selezionare la proprietà **Annotativo**.

3 Scegliere **Sì** dall'elenco delle opzioni.

La scala annotazione corrente viene applicata al tratteggio o all'entità riempimento sfumato selezionata e il tratteggio viene ridimensionato di conseguenza.

14.3.4.3 Rimuovere la proprietà annotativa

1 Selezionare il tratteggio o il riempimento sfumato.

1 Le proprietà dell'entità testo selezionata vengono visualizzate nel pannello **Proprietà**.

2 Selezionare la proprietà **Annotativo**.

3 Scegliere **No** nell'elenco delle opzioni.

14.3.5 Blocchi annotativi

La proprietà annotativa di un blocco viene impostata durante la creazione della definizione del blocco utilizzando il comando BLOCCO o successivamente nella finestra di dialogo **Esplora Disegno/Blocchi**.

I blocchi annotativi possono avere una proprietà opzionale per abbinare sempre l'orientamento di un layout, indipendente dalla rotazione della finestra dello spazio carta che lo contiene.

14.3.5.1 Rendere annotativa una definizione di blocco

1 Avviare il comando EXPBLOCKS.

1 Verrà visualizzata la finestra di dialogo **Esplora Disegno / Blocchi**.

2 Se necessario, scegliere **Visualizza Dettagli**.

Le definizioni dei blocchi annotativi hanno l'icona **Annotativo** ( ) nella colonna **Annotativo**.

3 Cliccare nel campo **Annotativo** di una definizione di blocco per attivare o disattivare la proprietà Annotativo.

14.3.5.2 Impostare la proprietà Corrispondenza orientamento con Layout

1 Avviare il comando EXPBLOCKS.

1 Verrà visualizzata la finestra di dialogo **Esplora Disegno / Blocchi**.

2 Se necessario, scegliere **Visualizza Dettagli**.

3 Selezionare la casella di controllo **Corrispondenza orientamento con Layout** di una definizione di blocco per impostare la proprietà.

14.3.6 Quote annotative

Le entità quota vengono create utilizzando l'attuale stile di quota:

- Se lo stile di quota è annotativo, le quote ottengono automaticamente la proprietà Annotativo e la scala annotazione corrente.
- Se lo stile di quota non è annotativo, è necessario rendere le quote annotative manualmente.

Note:

- L'impostazione **Scala generale della quota** (variabile di sistema DIMSCALE) non è modificabile per gli stili di quota annotativi ed è trascurata per le quote annotative.
- Si consiglia di impostare **Scala generale della quota** = 1 per gli stili di quota annotativi.

14.3.6.1 Rendere annotativa un'entità quota

1 Selezionare un'entità quota.

1 Le proprietà dell'entità quota selezionata vengono visualizzate nel pannello **Proprietà**.

2 Selezionare la proprietà **Annotativo**.

3 Scegliere **No** nell'elenco delle opzioni.

L'entità quota viene scalata rispetto al valore corrente della variabile di sistema DIMSCALE.

14.3.7 Direttrici annotative

Le Multidirettrici vengono create utilizzando lo stile multidirettrice corrente:

- Se lo stile Multidirettrice è annotativo, l' entità Multidirettrice ottiene automaticamente la proprietà annotativa e viene assegnata la scala dell'annotazione corrente.
- Se lo stile Multidirettrice non è annotativo, è necessario rendere manualmente annotative le entità Multidirettrice.

14.3.7.1 Rendere annotativa un'entità Multidirettrice

1 Selezionare l'entità **Multidirettrice**.

Le proprietà dell'entità **Multidirettrice** selezionata vengono visualizzate nel pannello **Proprietà**.

2 Selezionare la proprietà **Annotativo**.

3 Scegliere **Sì** nell'elenco delle opzioni.

È stata assegnata la scala di annotazione corrente.

L'entità **Multidirettrice** viene scalata rispetto alla scala annotazione corrente.

14.3.7.2 Rimuovere la proprietà annotativa

1 Selezionare l'entità **Multidirettrice**.

Le proprietà dell'entità **Multidirettrice** selezionata vengono visualizzate nel pannello **Proprietà**.

2 Selezionare la proprietà **Annotativo**.

3 Scegliere **No** nell'elenco delle opzioni.

15. Personalizzazione

15.1 Startup options

When starting up BricsCAD, it is possible to add arguments or switches. For example:

- The Command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" starts BricsCAD and loads the specified drawings.
- The Command line "bricscad.exe /P MyProfile" starts BricsCAD using the MyProfile user profile.

"/P MyProfile" is called a Command line switch. Both a slash or a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will works. The space is optional, e.g. "/PMyProfile" would also work, and the switch character is case insensitive, e.g. "-p MyProfile" is also fine.

Switch	Description
/P [userProfileName]	Starting without the /P option, BricsCAD uses the current profile (= the most recently used profile) which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, then a new empty user profile will be created with that name. If the /P argument is an .arg file the arg file is used to initialize the profile settings on the first run (See Starting BricsCAD with a specific profile).
/B [scriptFileName.scr]	Runs a script (*.scr) or lisp (*.lsp) at startup. Opens all the file types that can be dragged and dropped on the application main frame window. If you pass filenames without a parameter, they are opened as if they had been passed with /B. Therefore, bricscad.exe mylisp.lsp is exactly the same as bricscad.exe /B mylisp.lsp, and bricscad.exe my.dwg is identical to bricscad.exe /B my.dwg.
/L	Suppresses the startup splash screen.
/LD [application.arx/ brx/drx/lsp]	Loads an application on startup and on every new or opened drawing depending on the file extension. If the file extension is .lsp/.des/.mnl, it is treated as a lisp file and loaded in every drawing. Other file types are loaded only once at startup.
/Safemode	Prevents all third party plug-in code, enablers and user lisp files from loading in the current session. Starting in a clean environment can prevent potential crash causes. See Launching BricsCAD in safe mode.
/S [path]	Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C: \BricsCAD\
/T [TemplateFileName]	Creates a new drawing based on the template drawing specified.

Switch	Description
regserver unregserver	To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/".

Nota:

- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile".
- Switches can be combined, for example **/P MyProfile /L /S C:\BricsCAD** starts BricsCAD using the **MyProfile** user profile without the splash screen and sets the SRCHPATH system variable to **C:\BricsCAD** .
- Switches /B and /LD enable all sorts of files to be loaded, some maybe not expected. For example, .cui, .dsd, .txt, .pdf types are also processed.
- If GETSTARTED = ON when you launch BricsCAD, the **BricsCAD Launcher** dialog box appears, which lets you choose the workspace.

Besides these switches, BricsCAD supports file names as Command line arguments.

Supported file types are:

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

15.2 File utente di BricsCAD

In BricsCAD i file utente sono archiviati in cartelle specifiche:

- **PlotConfig**: file di configurazione del plotter (*.pc3)
- **PlotStyles**: file delle tabelle degli stili di stampa (*.stb) e file delle tabella stili di stampa dipendenti dai colori (*.ctb)
- **Supporto**:
  - file di definizione dei modelli di tratteggio (*.pat)
  - file tipo di linea (*.lin)
  - file alias (*.pgp)
  - file delle unità (*.unt)
  - file di personalizzazione interfaccia utente (*.cui)
  - file della libreria BIM (*.bsyslib), vere il comando BIMINFOPROGETTO.

- file di configurazione dell'albero della struttura (*.cst), vedere il comando PANNELLOSTRUTTURA.
- file di configurazione del pannello Libreria (*.ccf), vedere il comando APRIPANNELLOLIBRERIA.

- **Templates**: disegni de modelli (*.dwt)

Piattaforma	Cartelle	Sottocartelle
Windows	C:\\Users<UserName>\AppData\Local \Bricsys \BricsCAD\<Version>\it_IT	\Modelli
Windows	C:\\Users<UserName>\AppData\Roaming \Bricsys\BricsCAD\<Version>\it_IT	\PlotConfig \PlotStyles \Supporto
Mac	/Users/<UserName>/Library/Preferences/ Bricsys/BricsCAD/<Version>/it_IT	/PlotConfig /PlotStyles /Supporto /Modelli
Linux	/home/<UserName>/Bricsys/BricsCAD/ <Version>/it_IT	/PlotConfig /PlotStyles /Supporto /Modelli

15.2.1 Utilizzare il Gestore file utente BricsCAD

Quando esegui l'aggiornamento, la prima volta che avvii BricsCAD dopo l'aggiornamento, il contenuto della sottocartella **UserDataCache** della cartella di installazione di BricsCAD viene confrontato con il contenuto delle **cartelle dei file utente**. Se in queste cartelle sono presenti file personalizzati e uno o più dei file corrispondenti nella cartella **UserDataCache** sono più recenti, viene visualizzata la finestra di dialogo **Gestore file utente BricsCAD** che chiede se mantenere o sovrascrivere i file con la nuova versione.

Eseguire una delle seguenti operazioni:

1. Selezionare un file nella lista dei file, quindi selezionare **Sì** per sovrascrivere o selezionare **N**o per mantenere i propri file.
   - Ripetere questa procedura per ciascun file nella lista.
   - Selezionare **Si a Tutti** per sostituire tutti i propri file con i file più recenti di BricsCAD.
   - Cliccare sul pulsante **Ferma Copia** per mantenere tutti i propri file.
2. Se si è deciso di non aggiornare uno o più file, viene visualizzata una finestra di avviso che elenca tutti i file che si desidera conservare e che quindi non verranno aggiornati. Viene chiesto se ricevere o meno un promemoria la prossima volta che avvii BricsCAD.

3 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante **Sì** per aggiornare uno di questi file in un secondo momento.

     In questo caso questa procedura viene riavviata al successivo avvio di BricsCAD.
   - Cliccare sul pulsante **No** per non visualizzare il promemoria.

15.3 Loading and unloading CUI files

When you install BricsCAD, the default CUI files are copied to the **Support** folder (see BricsCAD User Files).

When you launch BricsCAD the first time, the **default.cui** file is loaded automatically.

There are two procedures to load a CUI file:

- Load main CUI file: replaces the currently loaded main CUI file.
- Load partial CUI file: partially replaces the content to the currently loaded main CUI file.

15.3.1 Loading a main CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load main CUI file ...** .

   The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

   The selected CUI file is loaded.

4 Close the **Customize** dialog.

15.3.2 Loading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 In the **File** menu, choose **Load partial CUI file ...** .

   The **Choose a customization file** dialog box displays the content of the **Support** folder.

3 Select a CUI file, then double-click or click the **Open** button.

   The content of the selected CUI file is appended: definitions which are different from the ones in the main CUI are used.

4 Close the **Customize** dialog.

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

15.3.3 Unloading a partial CUI file

1 Launch the CUSTOMIZE command.

1 The **Customize** dialog box appears.

2 Click the **Menus** tab.

3 Right-click the partial CUI name and choose **Unload <PartialCuiName>** in the context menu.

4 Close the **Customize** dialog.

15.3.4 Managing customization groups

1 Launch the MENULOAD command.

1 The **Customization groups** dialog opens.

2 (Option) To load a partial CUI file, click the browse button (...) to select a partial CUI file, then click the **Load** button.

3 (Option) To unload a partial CUI file, select the partial CUI file in the **Loaded Customization Groups** list, then click the **Unload** button.

4 Click the **Close** button to stop.

15.4 Creating a partial CUI file

1 Launch the CUSTOMIZE command.

2 Do one of the following:
   - In the **File** menu, choose **Create new partial CUI file ...**
   - Place the cursor on a Menu group, then right-click and choose **Create new partial CUI file ...** in the context menu.

   The **Create a Customization File** dialog opens.

   By default the new partial CUI file will be created in the **Support** folder.

3 (Option) Select a different folder to save the CUI file in.

4 Type a name for the new CUI file in the **File name** field.

5 Click the **Save** button.

   The newly created **Partial CUI file** is loaded and a **Menu Group** of the same name is created in the **Menus**, **Toolbars**, **Ribbon**, **Keyboards**, **Mouse**, **Tablet** and **Quad** tab and in the **Available Tools** pane of the different tabs.

15.5 Managing toolboxes

A toolbox contains a set of tools, which can be added to menus, toolbars, ribbon panels, Quad tabs and keyboard or mouse shortcuts. Toolboxes can be created in the main CUI or in a partial CUI.

15.5.1 Creating a toolbox in a menu group

1 In the **Customize** dialog box, select a menu group in the **Available Tools** panel:
   - BricsCAD (= default main CUI)
   - <PartialCuiName> (= partial CUI)

2 Right-click and choose **Append toolbox** in the context menu.

   The **Add Toolbox** dialog box appears.

3 Type a name for the new toolbox in the **Name** field, then click the **OK** button.

The toolbox is added in the **Available Tools** tree.

15.5.2 Creating a tool in a toolbox

1 In the **Customize** dialog box, select a toolbox in the **Available Tools** panel.

2 Right-click and choose **Append tool** in the context menu.

The **Add tool** dialog box appears.

3 **Title**: type a name for the new tool.

4 **Help**: type a help string.

5 **Command**: type a command or a command macro*.

6 **Image**: Click the **Browse** button (...).

The **Tool Image** dialog box appears.

7 Do one of the following:

- Select **Builtin**, then choose one of the built-in icons.
- Select **Bitmap file**, first choose to select one image file or a small and a large image file, then click the **Browse** button (...) to select the image file(s).

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

* Syntax to create command macros:

- Start the macro with **^c^c**, the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- Type an underscore (_) in front of a command name if you want to use the macro in a non-English version of BRICSCAD.
- A period (.) in front of a command name ensures that the macro still works if this command is temporarily undefined.
- A minus sign (-) in front of a command launches the Command line version of a command, thus preventing the command dialog window to open.
- A semicolon (;) is the equivalent of pressing the Enter key, e.g. to accept a default prompt option.
- A backslash (\) temporarily pauses the macro to allow user input, e.g. specifying a point.

In the following macro: ^c^c_-layer;m;InteriorWalls;;mline

- **^c^c** concludes a running command, if any.
- **-layer;** launches the -LAYER command.
- **m;** selects the **Make new current layer** option.
- **InteriorWalls** makes the InteriorWalls layer current and creates it first if necessary.
- **;;mline** concludes the -LAYER command and launches the MLINE command.

15.6 Creating toolbars

1 In the **Customize** dialog box, click the **Toolbars** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append toolbar** in the context menu.

     The new toolbar will be added below the last existing toolbar.
   - Select an existing toolbar in a menu group, then right-click and choose **Insert toolbar** in the context menu.

     The new toolbar will be added above the selected toolbar.

   The **Add toolbar** dialog box appears.
3 Type a name in the **Title** field.
4 Click the **OK** button.

   The toolbar is created.
5 Define the properties of the toolbar:
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Position**: Defines the display position of the toolbar when the CUI file is loaded.

     The options are: **Floating**, **Top**, **Left**, **Bottom** or **Right**. The latter four refer to the window edge where the toolbar will be docked.
   - **Display**: Defines whether the toolbar appears or not when the CUI file is loaded.
   - **Rows**: Specifies the number of rows when its position is floating. Toolbars with controls cannot have multiple rows.
   - **Xval**: When the position is Floating, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
   - **Yval**: When the position is Floating, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.
6 Add items to the toolbar.

15.6.1 To add a flyout to a toolbar

1 In the **Customize** dialog box, click the **Toolbars** tab.
2 Do one of the following:
   - Select a toolbar, then right-click and choose **Append Flyout** in the context menu, to add the flyout at the end of the toolbar.
   - Expand the toolbar, right-click a tool then choose **Insert flyout** in the context menu, to add the flyout above the selected tool.
3 The **Add flyout** dialog box appears.
4 Type a name in the **Title** field of the **Add flyout** dialog.
5 Click the **OK** button.

   A new empty toolbar is created and inserted as a flyout in the selected toolbar.

6 Define the flyout properties of the toolbar.
   - **Title**: The toolbar name must include alphanumeric characters with no punctuation other than a dash (–) or an underscore (_). The **Title** along with the **Alias** enables the toolbar to be referenced programmatically.
   - **Target alias**: Identifies the referenced toolbar.
   - **Own Icon**:
   - **No**: The icon of the first tool or the icon of the most recently used tool is displayed in the parent toolbar.
   - **Yes**: The selected icon is displayed in the parent toolbar.
7 Define the properties of the toolbar when the flyout is teared off and inserted as a floating toolbar.
8 Add items to the toolbar.
9 Items added to the flyout are automatically added to the toolbar and vice versa.

15.7 Creating menus

Main menus sit in the menu bar at the top of the BricsCAD application window. Submenus show as a flyout of a main menu.

The MENUBAR system variable controls the display of menus.

15.7.1 Creating a main menu

1 In the **Customize** dialog, click the **Menus** tab.
2 Do one of the following:
   - Click the **Main Menus** subgroup of a menu group, then right-click and choose **Append main menu** in the context menu.

     The new menu will be added below the last existing menu.
   - Select an existing menu in the **Main Menus** subgroup, then right-click and choose **Insert main menu** in the context menu.

     The new menu will be added above the selected menu.

   The **Add Main Menu** dialog box appears.
3 Type a name in the **Title**.
4 Click the **OK** button.

   The main menu is created.
5 Add items to the menu.

Nota: The new main menu is not visible in the menu bar until at least one menu item is added to it.

15.7.2 Adding a submenu to a main menu

1 In the **Customize** dialog, click the **Menus** tab.

   If collapsed, expand the menu group.

   If collapsed, expand the **Main menus** group of the menu group.

2 Do one of the following:
   - Select a main menu, then right-click and choose **Append submenu** in the context menu.

     The new submenu will be added below the last existing submenu.
   - Select an existing submenu, then right-click and choose **Insert submenu** in the context menu.

     The new submenu will be added above the selected submenu.

   The **Add submenu** dialog box appears.
3 Type a name in the **Title** field.
4 Click the **OK** button.

   The submenu is created.
5 Add items to the submenu.

15.7.3 Adding a submenu to a submenu

1 In the **Customize** dialog, click the **Menus** tab.
2 Expand the submenu.
3 Select an item in the submenu, then right-click and choose **Insert submenu** in the context menu.

   The **Add submenu** dialog box appears.
4 4. Type a name in the **Title** field.
5 Click the **OK** button.

   The submenu is created.
6 Add items to the submenu.

15.8 Creating context menus

The availability of context aka shortcut menus is controlled through the SHORTCUTMENU system variable. There are three context menu modes:

- **Default mode**: Displays the **Default** context menu when no command is active and nothing is selected.
- **Edit mode**: Displays the **Edit** context menu when a selection set exists. This mode is context sensitive. If a specific shortcut menu for the selected entity or entities exists, its content is added to the **Edit** context menu. For example, the content of the **Multiline Text Object** context menu is available in the **Edit** context menu when an MText entity is selected.
- **Command mode**: Displays the **Command** context menu when a command is running.

Shortcut menus appear when you right-click. However:

- When bitcode 16 of the SHORTCUTMENU system variable is set, a right-click repeats the previous command and you need to hold down the right mouse button longer (= long right-click) to display shortcut menus.
- When bitcode 4 of the QUADDISPLAY system variable is set (= Show Quad on right-click), shortcut menus only show when bitcode 16 of the SHORTCUTMENU system variable is set.

Special shortcut menus are:

- **Grips**: Appears on a right-click when using grips to edit an entity.
- **Entity Snap**:
  - MBUTTONPAN = OFF: appears when you click the mouse wheel (middle mouse button).
  - MBUTTONPAN = ON: displays on Shift + Right-Click

15.8.1 Creating shortcut menus

1 In the **Customize** dialog, click the **Menus** tab.

2 Do one of the following:
  - Click the **Context Menus** subgroup of a menu group, then right-click and choose **Append context menu** in the context menu.

    The new context menu will be added below the last existing context menu.
  - Select an existing context menu in the **Context Menus** subgroup, then right-click and choose **Insert context menu** in the context menu.

    The new context menu will be added above the selected context menu.

  The **Add Context Menu** dialog box appears.

3 Type a name in the **Menu title** field.

4 (option) To create a regular context menu:
  - Choose **As regular context menu**.
  - Click the **Add Context…** button.

    The **Add context alias** dialog box appears.
  - Select a context alias. The options are: CMCOMMAND, CMDEFAULT, CMEDIT, GRIPS and SNAP.
  - This controls when the context menu is available. See above.
  - (option) add another context alias.
  - Click the **OK** button to create the context menu.

5 (option) To create a context sensitive context menu:
  - Choose **As context menu on specific entities**.
  - Click the **Add Entity Types…** button.

    The **Add Entity Alias** dialog appears.
  - Tick the **One entity** and/or **Multiple entities** boxes.
  - Select one or more entities in the list, then click the **OK** button.
  - (option) Click the **Add Custom…** button add a custom alias.
  - Click the **OK** button to create the context menu.

6 Add tools to the context menu.

15.9 Creating keyboard shortcuts

1 In the **Customize** dialog box, click the **Keyboard** tab.

2 Do one of the following:
   - Select a menu group, then right-click and choose **Append shortcut** in the context menu.

     The new shortcut will be added below the last shortcut.
   - Select an existing shortcut in a menu group, then right-click and choose **Insert shortcut** in the context menu.

     The new shortcut will be added above the selected shortcut.

   The **Add keyboard shortcut** dialog box appears.
3 Do one of the following:
   - Select **Add an existing tool**, then select an existing tool.
   - Select **Add a new tool**, then create a new tool.

     See the **Create a tool in a toolbox** procedure in **Managing Toolboxes**.
4 In the **Properties** grid of the new shortcut, select the **Key** field, then press the key combination you want to assign to the shortcut.

15.10 Customizing the Quad

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

- Right-click the **QUAD** field in the **Status** bar and choose **Customize Quad items...** in the context menu.
- Run the CUSTOMIZE command, then select the **quad** tab.

15.10.1 Creating a quad tab

1 Do one of the following:
   - Right-click a menu group and choose **Append quad tab** in the context menu.

     The quad tab will be added below the last existing quad tab in the menu group.
   - Right-click an existing quad tab and choose **Insert quad tab** in the context menu.

     The quad tab will be added above the selected quad tab.

   The **Add Quad Tab** dialog box appears.
2 Type a name in the **Label** field.

   The label appears in the Quad cursor menu.
3 Type a name in the **Title** field.

   The tab title must be unique in the menu group.
4 Click the **OK** button.
5 Add tools to the tab.

15.10.2 Copying a quad tab

You can insert a copy of a quad tab in the same menu group or to another menu group.

- To insert a copy of a quad tab, select the tab, then right-click and choose **Insert Copy** in the context menu.

The **Add Quad Tab** dialog box appears.

Specify a label and title (see above), then click the **OK** button.

- To insert a copy of quad tab in another menu group: drag-and-drop the quad tab onto the target menu group.

15.10.3 Setting the Entity Filter property of a tool

The Quad cursor menu is context sensitive. The available tools depend on the entity under the cursor or the current selection set. When no selection exists, the 'No Selection Quad' appears on a right-click if bitcode 4 of the QUADDISPLAY system variable is selected.

1 Select the tool.

1 The **Quad Button** properties are displayed in the bottom section of the **Customize** dialog box.

2 Select the **Entity Filter** property.

The default setting is NO_SELECTION.

3 Click the **Browse** button (...) at the right hand side of the field.

The **Edit Quad Button Filter** dialog box appears.

4 (option) Click the **Add Filter...** button.

The **Add quad button filter** dialog box appears.

Click the down arrow at the right hand side of the field and choose a filter:

- NO_SELECTION: available when nothing is selected.
- OBJECT_ANY: available when a single entity is selected.
- OBJECTS_ANY: available when multiple entities are selected.

Click the **OK** button.

5 (option) Repeat the previous step to add a filter or remove a filter.

6 (option) Click the **Add Entity Type(s)...** button.

The **Add Entity Alias** dialog box appears.

- Tick the **One entity** and/or **Multiple entities** boxes.
- Tick the box the entity type(s) the tool applies to.

Click the **OK** button.

Nota: It is not possible to modify the Entity Filter of a built-in Quad Button.

15.11 Adding tools to user interface controls

Tools can be drag-and-dropped from the toolboxes in the **Available Tools** section of the **Customize** dialog box onto menus, context menus, toolbars, rows in a ribbon panel, keyboard shortcuts and Quad tabs.

1 Select the target in the **Customize** dialog box.

2 In the **Available Tools** section, expand the source toolbox.

3 Select the tool and drag-and-drop it on the target item.

4 (option) Expand **Controls** in the **Available Tools** section, then drag-and-drop a control onto the target item.

5 (option) Drag-and-drop tools to reposition them or to move them to a different target item.

15.12 Personalizzare la barra multifunzione

La barra multifunzione è composta da una serie di schede, che a loro volta sono composte da pannelli. I pannelli della barra multifunzione sono costituiti da righe con strumenti, controlli e sottopannelli.

Le schede della barra multifunzione controllano quali pannelli vengono visualizzati sulla barra multifunzione.

Per visualizzare una scheda della barra multifunzione sulla barra multifunzione, è necessario aggiungere la scheda a un'area di lavoro.

15.12.1 Creazione di una scheda della barra multifunzione

1 Nella finestra di dialogo **Personalizza**, cliccare sulla scheda **Barra Multifunzione**.

2 Eseguire una delle seguenti operazioni:

- Espandere un gruppo di menu, quindi cliccare il pulsante destro del mouse sulle **schede barra multifunzione** e scegliere **Aggiungi scheda barra multifunzione** nel menu contestuale.

  La nuova scheda barra multifunzione verrà aggiunta sotto l'ultima scheda barra multifunzione esistente.

- Selezionare una scheda barra multifunzione esistente in un gruppo di menu, Cliccare il pulsante destro del mouse e scegliere **Inserisci scheda barra multifunzione** nel menu contestuale.

  La nuova scheda della barra multifunzione verrà aggiunta sopra la scheda della barra multifunzione selezionata.

Viene visualizzata la finestra di dialogo **Aggiungi Scheda Barra Multifunzione**.

3 Definire la scheda della barra multifunzione:

- **Etichetta**: Nome della scheda come mostrato nella barra multifunzione.
- **Titolo**: Nome lungo della scheda.
- **ID**: Tag univoco per identificare la scheda nel gruppo di menu. L'ID viene utilizzato per fare riferimento alla scheda della barra multifunzione a livello di programmazione.

4 Cliccare sul pulsante **OK** per creare la scheda della barra multifunzione.

15.12.2 Creazione di un gruppo della barra multifunzione

1 Nella finestra di dialogo **Personalizza**, cliccare sulla scheda **Barra Multifunzione**.

2 Eseguire una delle seguenti operazioni:

- Espandere un gruppo di menu, quindi cliccare il pulsante destro del mouse su **Pannelli barra multifunzione** e scegliere **Inserisci pannello barra multifunzione** nel menu contestuale.

  Il nuovo pannello barra multifunzione verrà aggiunto sotto l'ultimo pannello barra multifunzione esistente.

- Selezionare un pannello barra multifunzione esistente in un gruppo di menu, cliccare il pulsante destro del mouse e scegliere **Inserisci pannello barra multifunzione** nel menu contestuale.

  Il nuovo pannello della barra multifunzione verrà aggiunto sopra il pannello della barra multifunzione selezionato.

Viene visualizzata la finestra di dialogo **Aggiungi Pannello Barra Multifunzione**.

3 Definire il pannello della barra multifunzione:
   - **Etichetta**: Nome del pannello visualizzato nella barra multifunzione.
   - **Titolo**: Nome lungo del pannello.
   - **ID**: Tag univoco per identificare il pannello nel gruppo di menu. L'ID viene utilizzato per fare riferimento al pannello della barra multifunzione a livello di programmazione.

4 Cliccare sul pulsante **OK** per creare il pannello della barra multifunzione.

15.12.3 Aggiungere un pannello barra multifunzione a una scheda

1 Nella finestra di dialogo **Personalizza**, cliccare sulla scheda **Barra Multifunzione**.

2 Espandere **Schede della barra multifunzione** in un gruppo di menu.

3 Eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse sulla scheda della barra multifunzione e scegliere **Aggiungi pannello** nel menu contestuale.

     Il pannello della barra multifunzione verrà aggiunto sotto l'ultimo pannello barra multifunzione esistente.
   - Espandere una scheda della barra multifunzione, quindi cliccare il pulsante destro del mouse e scegliere **Inserisci pannello** nel menu contestuale.

     Il pannello della barra multifunzione verrà aggiunto sopra il pannello della barra multifunzione selezionato.

Viene visualizzata la finestra di dialogo **Seleziona Pannello Barra Multifunzione**.

Sono elencati tutti i pannelli barra multifunzione esistenti nel gruppo di menu.

4 Selezionare un pannello nell'elenco, quindi doppio clic o cliccare sul pulsante **OK**.

Il pannello viene aggiunto alla scheda.

15.12.4 Aggiungere una riga a un pannello

1 Nella finestra di dialogo **Personalizza**, cliccare sulla scheda **Barra Multifunzione**.

2 Espandere **Pannelli barra multifunzione** in un gruppo di menu.

3 Selezionare un pannello, quindi eseguire una delle seguenti operazioni:
   - Cliccare il pulsante destro del mouse e scegliere **Aggiungi riga** nel menu contestuale.

     La riga verrà aggiunta sotto l'ultima riga esistente nel pannello selezionato.
   - Selezionare una riga esistente, quindi cliccare il pulsante destro del mouse e scegliere **Inserisci riga barra multifunzione** nel menu contestuale.

     La riga verrà aggiunta sopra la riga selezionata.

15.12.5 Aggiungere elementi a una riga

Cliccando il pulsante destro del mouse su una riga, sono disponibili le seguenti opzioni in un menu contestuale:

- **Aggiungi riga pannello**: aggiunge un sottopannello contenente tre righe del sottopannello per impostazione predefinita. È quindi possibile aggiungere uno o più elementi a queste righe e aggiungere/rimuovere righe.
- **Aggiungi pulsante di comando**: vedere **Aggiunta di strumenti ai controlli dell'interfaccia utente**.
- **Aggiungi pulsante a discesa**: aggiunge un pulsante a cui è possibile aggiungere più elementi. Viene visualizzato solo il primo elemento o l'elemento utilizzato più di recente.
- **Aggiungi pulsante interruttore**: aggiunge un pulsante per attivare o disattivare un'impostazione o un'opzione. Ad esempio snap di entità o priorità di selezione. Un pulsante interruttore indica lo stato dell'impostazione o dell'opzione.
- **Aggiungi separatore**: aggiunge uno spazio tra le righe.

15.12.6 Definire le proprietà di un elemento

Selezionare l'elemento, quindi definire le proprietà nella tabella delle proprietà nella parte inferiore della finestra di dialogo **Personalizza**.

15.12.7 Riposizionare gli elementi

Per riposizionare qualsiasi elemento in un pannello o in una riga: selezionare l'elemento, quindi tenere premuto il pulsante sinistro del mouse e spostare l'elemento nella posizione desiderata. Rilascia il pulsante del mouse per posizionare l'elemento.

15.12.8 Elimino elementi

Selezionare l'elemento, quindi cliccare il pulsante destro del mouse e scegliere **Elimina** nel menu contestuale.

15.13 Tool palettes

On a tool palette you can add blocks, hatches and commands in a tabbed window.

Drag a tool from the palette to the graphic screen to launch the related tool.

15.13.1 Opening the Tool Palettes panel

Do one of the following:

- Hover over a toolpanel (Windows) or an empty toolpanel area (Windows and Linux), then right-click and choose **Palettes** > **Tool Palettes** in the context menu.
- Launch the TOOLPALETTES command.

15.13.2 Setting the view options

1 Right-click an empty space of the icon area on the **Tool Palettes** panel and choose **View Options...** in the context menu.

The **View Options** dialog appears.

2 Drag the **Image Size** slider to adjust the image size.

3 Set the viewstyle: **Icon with text**, **Icon only** or **List view**.

4 Choose either **Current Palette** or **All Palettes** from the **Apply to** list button.

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

15.13.3 Customizing tool palettes

Nota: The default folder for tool palettes is defined by the TOOLPALETTEPATH system variable.

15.13.4 Opening the Customize dialog box

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

2 Choose **Customize Palettes** in the context menu.

The **Customize** dialog box appears.

15.13.5 Importing a tool palette

1 In the **Palettes** pane, right-click and choose **Import** in the context menu.

The **Import Palette** dialog box appears.

2 Select a folder, then select a palette file (*.atc, *.btc or *.xtp).

3 Click the **Open** button or double-click to import the palette.

The palette is added in the **Palettes** list.

4 Click the **Close** button to close the **Customize** dialog box.

15.13.6 Creating a palette group

1 Place the cursor on the **Palette Groups** root folder name.

2 Right-click and choose **New Group** in the context menu.

A new group folder is created.

3 Type a name for the new group.

15.13.7 Creating a palette subgroup

1 Place the cursor on an existing group name.

2 Right-click and choose **New Palette** in the context menu.

A new group folder is added as a subgroup of the selected group.

3 Type a name for the new group.

15.13.8 Adding a palette to a group

1 Place the cursor on a palette.

2 Select a palette, then drag the palette onto a palette group name.

15.13.9 Removing a palette from a group

1 Select the palette in the group.

2 Right-click and choose **Remove** in the context menu.

15.13.10 Setting a palette group current

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Set Current** in the context menu.

15.13.11 Deleting a palette group

1 Select the palette group in the palette groups tree.

2 Right-click and choose **Delete** in the context menu.

15.13.12 Renaming a tool palette

1 Right-click tool palette tab and choose **Rename Palette** in the context menu.

2 Type a new name for the palette in the **Name** field.

3 Press Enter to confirm.

15.13.13 Deleting a tool palette

1 Right-click tool palette tab and choose **Delete Palette** in the context menu.

2 Click the **OK** button on the **Confirm Palette Deletion** box.

15.13.14 Managing tools

1 Right-click the tool.

1 A context menu appears.

2 Choose an option:

- **Cut**: Removes the item from the palette. Open another panel, then right-click and choose **Paste** in the context menu.
- **Copy**: Copies the tool. Open another panel, then right-click and choose **Paste** in the context menu.
- **Delete**: Deletes the tool. A dialog box appears, asking you to confirm the deletion.
- **Rename**: Type a new name for the tool.
- **Update tool image**: Updates the image if the image source file has been modified.
- **Specify Image...**: Allows you to replace the default image.

  Choose a new image in the **Select Image File** dialog box, then double-click or click the **Open** button.
- **Remove Image...**: Restores the default image.
- **Properties**: When you click an item on a tool palette, a command is launched. The properties of the tool palette item control the execution of the command.

  The available properties on the **Tool Properties** dialog box depend on the tool type: Command, Hatch, Block or Component.

Edit the tool properties, then click the **OK** button.

15.13.15 Creating a new tool palette

1 Move the cursor to the icon area of the **Tool Palettes** panel, then right-click.

A context menu appears.

2 Choose **New Palette** in the context menu.

A new tab is added.

3 Type a name for the new palette in the **Name** field.

15.13.16 Adding items to a tool palette

The following items can be added to a tool palette:

- Hatch patterns
- Blocks
- Components
- Commands

15.13.17 Adding hatch patterns

1 Select the palette you want to add a hatch pattern to in the **Tool Palettes** panel.

2 Open the **Hatch and Gradient** dialog box.

3 Select either **Predefined** or **Custom** from the **Type** list button.

4 Double-click the **Swatch** field on the **Hatch and Gradient** dialog box.

The **Hatch Pattern Palette** appears.

5 Select the hatch pattern you want to add, then right-click and choose **Add to Current Tool Palette**.

The hatch pattern is added on the selected tool palette.

6 (option) Repeat steps 4 and 5 to add more hatch patterns.

7 (option) Drag-and-drop patterns to change their order on the palette.

15.13.18 Adding blocks

Blocks can be added from:

- the current drawing.
- any drawing.
- all drawings in a selected folder.

15.13.19 Adding blocks from the current drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Blocks**.

3 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

4 Right-click and choose **Add to Current Tool Palette** in the context menu.

15.13.20 Adding blocks from any drawing

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in the **Local Folders** list.

5 Click the **Expand** icon (+) of the containing drawing and click the **Blocks** icon.

All blocks in the selected drawing are displayed in the **Details** pane.

6 Select the block(s) you want to add. Hold down the Ctrl key to select multiple blocks.

7 Right-click and choose **Add to Current Tool Palette** in the context menu.

15.13.21 Adding a drawing as a block

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the drawing in **Local Folders** list.

5 Right-click and choose one of the following in the context menu.

- **Add to Current Tool Palette as block**.

  Blocks are inserted using the INSERT command.

- **Add to Current Tool Palette as component**.

  Components are inserted using the BMINSERT command.

15.13.22 Adding all blocks from multiple drawings

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

1 Open the **Drawing Explorer** dialog box.

2 Select **Folders** in the left hand pane.

3 If not already available, click the **Add local folder...** button to add the containing folder to the **Folders** list.

4 Select the folder, then right-click and choose **Add blocks to Tool Palettes** in the context menu.

15.13.23 Adding commands

1 Open the **Customize** dialog box.

2 In the **Available Tools** pane, on the **Customize** dialog, expand a menu group (e.g. BricsCAD).

All toolboxes in the selected menu group are listed.

3 Expand a toolbox.

4 All commands in the selected toolbox are listed.

5 Right-click a tool and choose **Add to Current Tool Palette** in the context menu.

15.14 Command aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the Command line. There is no need to type the complete name. After typing the command alias the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for the RECTANG command, whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands using the keyboard.

Command aliases are saved in a *.pgp file (P roG ram P arameters file). By default the **default.pgp** file in the **Support** folder of the Roamable Root folder is used.

If a command can be executed transparently (= while another command is running), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

Nota:

- Unlike commands, aliases are only local and cannot be used as global commands preceded by a underscore (_).
- Command aliases cannot be used in scripts (*.scr).

15.14.1 Editing the alias file in a text editor

In order to modify existing aliases or to create new ones, the **default.pgp** file can be edited using an ASCII text editor (e.g. Notepad).

The first section of the **default.pgp** file defines the **Shell** commands. The second section of the **default.pgp** file defines command aliases.

The following syntax is used to define a command alias:

alias, *command

alias = the character string to be entered at the command prompt. The command alias must be followed by a comma (,).

command = the BricsCAD command. The command name must be preceded by an asterisk (*).

Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom.

Some commands, such as HATCH or BOUNDARY, can be preceded by a hyphen (-) to access the Command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.:

vi, *-view

Execute the REINIT command to reload the alias file when it was externally edited.

15.14.2 Using the Customize dialog to edit the alias file

1 Click the **Command Aliases** tab on the **Customize** dialog.

2 (option) To create a new alias,
   - Do one of the following:
   - Click the **Add** button.
   - Right-click an alias and choose **Add Alias** in the context menu.
   - Right-click a command and choose **Add Alias** in the context menu to create an alias for the selected command.

     The **Add Alias** dialog box appears.
   - Type the command alias in the **Alias** field.
   - Click the down arrow at the right hand side of the **Command** field, then select a command. You can start typing the command name in the **Command** field to easily find the command in the list.
   - Click the **OK** button.

3 (option) To edit an existing alias,
   - Select the alias in the list.
   - Click the **Edit** button, or right-click the alias and choose **Edit Alias** in the context menu.

     The **Edit Alias** dialog box appears.
   - Type a new alias in the **Alias** field.

     If the alias already exists, you are prompted to overwrite the existing alias.
   - Click the **OK** button.

4 (option) To delete an alias,
   - Select the alias in the list.
   - Click the **Delete** button, or right-click and choose **Delete** in the context menu.

     A **Confirm** dialog box appears.
   - Click the **OK** button on the **Confirm** dialog box to delete the alias.

Nota: If you use these procedures to edit the .pgp file, it is not necessary to execute the REINIT command.

15.15 User profiles

In BricsCAD, you can customize the settings that control your drawing environment, which can then be saved in a **user profile**. You can create multiple user profiles, e.g. a profile for each of your drawing types or workflows. User profiles can also be used when multiple users share the same computer, and each user wants to create their own drawing environment.

User profiles can be exported to a file, which can then be imported on a different computer. This allows you to copy your settings to a new computer or to install the same settings on multiple computers.

If you use third party applications on top of BricsCAD, you might want to create a user profile to use with each of these applications.

The **User Profile Manager** is a standalone tool that lists the existing profiles and provides all tools needed to **Load**, **Create**, **Set Current**, **Copy**, **Rename**, **Delete**, **Export** and **Import** user profiles.

15.15.1 Content of a user profile

User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment, such as the following.

- Main Cui file.
- Current Workspace. BricsCAD has six standard workspaces: Drafting, Drafting (toolbars), Modeling, Modeling (toolbars), Mechanical and BIM.
- Configuration settings: all system variables which are saved in the Registry and user preferences.
- Default plot settings.
- Settings in dialog windows, for example, search options in the **Settings** dialog.
- Project settings.
- Recent paths.
- Status bar settings.
- Properties panel settings.
- Toolbar and menu configuration.
- Worksets.

15.15.2 The User Profile Manager

To open the **User Profile Manager** dialog box:

- When BricsCAD is running: launch the PROFILEMANAGER command.
- Execute **profilemanager_app.exe** in the BricsCAD program folder (e.g. *C:\Program Files\Bricsys \BricsCAD Vx*).

In the **User Profile Manager** dialog:

- The **Current Profile** is marked.
- **Create**: Creates a new profile, using default settings.
- **Set Current**: Loads the selected profile.
- **Copy**: Creates a copy of the selected profile in the **Copy profile** dialog box.
- **Delete**: Deletes the selected profile.
- **Export...**: Exports the selected profile to a .arg file.
- **Import...**: Imports a **.arg** file as a new profile.
- **Start**: Starts BricsCAD with the selected user profile. This option is available only when the dialog box is opened using **profilemanager_app.exe** in the BricsCAD installation folder.
- **OK**: Closes the dialog box.

15.15.3 Starting BricsCAD with a specific user profile

When you restart BricsCAD, the user profile that was current when you closed the previous session is used.

To create a desktop shortcut to launch BricsCAD using a specific user profile, do one of the following:

- Add the **/P option** to the **Target** settings of the BricsCAD shortcut. To open the shortcut settings dialog: right-click the shortcut icon on the desktop and choose **Properties** in the context menu.

  A typical BricsCAD shortcut could be: **"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P UserProfileName**
- Execute **profilemanager_app.exe**, then either double-click a user profile or select a user profile and click the **Start** button.

Nota:

- The **/P** option is case insensitive.
- A space is needed between the file location and /P option.
- When BricsCAD is already running, a second instance of BricsCAD is launched, at least on condition that the SINGLETON variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog page for more information about variables and settings.
- If the user profile in the argument of the **/P** option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. /P "background white".

15.16 Workspaces

The WORKSPACE command sets the current workspace and lets you create, modify and save workspaces.

The WSSAVE command saves the current workspace under a different name, thus creating a new workspace.

The WSSETTINGS command opens the **Customize** dialog box at the Workspaces tab.

Workspaces define settings and preferences regarding a specific working environment.

The following workspaces are available:

- Drafting
- Drafting (toolbars)
- Modeling
- Modeling (toolbars)
- Mechanical
- BIM

The name of the workspace is stored in the WSCURRENT system variable of the current User Profile.

If the GETSTARTED system variable is ON, you can select the workspace on the **BricsCAD Launcher** dialog box on startup.

The availability of a startup option depends your license level.

Startup Option	Workspace	License Level
2D Drafting	Drafting	Lite or higher
3D Modeling	Modeling	Pro or higher
Mechanical design	Mechanical	Mechanical or Ultimate
BIM	BIM	BIM or Ultimate

15.16.1 Managing workspaces

Workspaces are defined in the **Workspace** tab page on the **Customize** dialog box.

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand a workspace.

3 A workspace definition is composed of:

- **Menus**: defines which menus are available.
- **Toolbars**: defines which toolbars are displayed by default.
- **Panels**: controls the visibility of dockable panels.

  Right-click a palette, then choose an option in the context menu:

  - **Show**: the panel is visible.
  - **Hide**: the panel is not visible.
  - **Don't change**: keep the current state.
- **Ribbon**: defines the available ribbon tabs.
- **Quad Tabs**: defines the order and availability of the Quad tabs.
- **On Switch**: defines the system variables and user preferences to be set, and the commands to be executed, when switching to the workspace.

15.16.2 Setting the current workspace

Do one of the following:

- Right-click the **Current Workspace** field in the **Status** bar, then select a workspace in the context menu.
- Choose a workspace in the **Workspaces** toolbar.
- Type **WSCURRENT** at the command prompt.

  You are prompted: New value for WSCURRENT <"2d DRAFTING">:

  Type the name of the new current workspace, then press Enter.

Nota:

- If the WSAUTOSAVE system variable is ON, upon switching to another workspace, the toolbar positions of the current workspace are saved in the CUI file. The default value for WSAUTOSAVE is ON.

- It is not recommended to switch workspaces if the name of the user profile refers to a workspace.

15.16.3 Creating a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Do one of the following:

- Select Workspaces, then right-click and choose **Append workspace** in the context menu.

  The new workspace will be added below the last existing workspace.

- Select an existing workspace, then right-click and choose **Insert workspace** in the context menu.

  The new workspace will be added above the selected workspace.

The **Add workspace** dialog box appears.

3 Type a name in the **Name** field then press Enter or click the **OK** button.

A newly created workspace contains the following items:

- A standard menu set
- A standard toolbar set
- All palettes set to **Don't change**
- An empty ribbon
- A default set of Quad tabs
- An empty **On Switch** sequence

Default properties. See the **Settings** dialog box under **Program Options / Workspace**.

15.16.4 Editing a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Expand the workspace.

3 (option) Edit the menu set:

- Change the order: Drag a menu.
- Delete a menu: select the menu, then right-click and choose **Remove** in the context menu.
- Insert a menu:
  - Right-click a menu in the list and choose **Insert menu** in the context menu.

    The **File** menu is inserted by default.
  - In the **Workspace Menu** grid, select the **ID** field and select the desired menu in the list.

4 (option) Edit the toolbar set:

- Change the order: Drag a toolbar.
- Delete a toolbar: select the toolbar, then right-click and choose **Remove** in the context menu.
- Insert a toolbar:
  - Right-click a toolbar in the list, and choose **Insert toolbar** in the context menu.

    The **Standard** toolbar is inserted by default.
  - In the **Workspace Toolbar** grid, select the **ID** field and select the desired toolbar in the list.

5 Edit the properties of the panels:
   - Set the desired 'on switch' behavior: **Show**, **Hide** or **Don't change**.
   - Select a panel, then edit its properties in the property grid.

6 Edit the composition of the ribbon:
   - To add a tab: right-click an existing tab, then choose **Insert tab** in the context menu and select the tab in the **Select Ribbon Tab** dialog.
   - To remove a tab: right-click the tab and choose **Remove** in the context menu.
   - To change the position of a tab: drag the tab.

7 Edit the Quad settings:
   - To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.
   - Right-click a Quad command group, then choose **Switch off** in the context menu to remove the command group from the Quad cursor menu.
   - Right-click a currently switched off Quad command group, then choose **Switch on** in the context menu to add the command group to the Quad cursor menu.

8 Edit the **On Switch** procedure:

   To add a command to the **On Switch** sequence: select the command in the available commands pane and drag it to the **On Switch** node.

15.16.5 Deleting a workspace

1 In the **Customize** dialog box, click the **Workspaces** tab.

2 Right-click the workspace, then choose **Delete** in the context menu.

   A **Confirm** dialog box appears.

3 Press the **Yes** button to delete the selected workspace.

Nota: Do not delete default workspaces, as they are used in the **BricsCAD Launcher** dialog box. When one of the default workspaces is deleted, use the **Importing a Workspace** procedure below to restore the missing workspace from Default.cui in the .../UserDataCache\Support\en_US subfolder of the BricsCAD installation folder.

15.16.6 Importing a workspace

The option in the **File** menu on the **Customize** dialog box can import workspaces from another CUI file into the current main CUI.

1 In the **Customize** dialog box, choose **Import workspaces...** in the **File** menu.

   The **Choose a Customization File** dialog appears.

2 Browse to the folder where the CUI file, you want to import the workspace from, is saved.

3 By default, BricsCAD saves CUI files in the **Support** folder of the Roamable Root folder. See BricsCAD User Files.

4 Select the CUI file.

5 A dialog box appears, listing all workspaces in the selected CUI file.

6 Select the workspace(s) to import in the list, then click the **OK** button.

Press and hold the Ctrl key to select multiple workspaces.

15.17 Express tools

15.17.1 About Express tools

Express tools is a collection of productivity tools that can be added to BricsCAD. You might already know these tools from AutoCAD.

The tools are available for you to provide additional options or possibilities but they are unsupported. BricsCAD is not responsible for the successful operation of these tools.

If you want to know more about how these tools work, go to CADwiesel.

15.17.2 Adding Express tools to bricsCAD

1 Go to the Bricsys application page.

2 If Express Tools is not present on the application page, enter **Express tools** in the search field.

3 Click the Express Tools tile.

The Express Tools detail page displays.

4 Do the following:

- Select the BricsCAD version.
- Tick the **I agree to the extended privacy statement** box.
- Click the Download button.

You will receive a zip file.

5 Unzip this file to a location of your choice.

6 Without BricsCAD running, double click on the installation file and install the Express Tools.

- Select a language.
- Specify the installation folder or accept the default folder.
- Click the **Install** button.

The Express tools are installed.

The **Express.cui** file is added in the Support folder of the roamable root folder. By default this is *C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\<version>\en_US\Support*

7 The next time you launch BricsCAD, the Express tools are available in:

- The Express menu.
- Four Express tools toolbars.
- The Express Tools tab on the ribbon.

15.17.3 Unloading the Express tools

1 Launch the CUIUNLOAD command.

The Customization Groups dialog box displays.

2 Select EXPRESS.

3 Click the Unload button.

15.17.4 Reloading the Express tools

Type **expressmenu** in the Command line, then press Enter.

The **Express.cui** file is reloaded as a partial cui and the Express tools menu, toolbars and ribbon tab are available again.

15.18 Lisp

How to create LISP routines for drawing setup.

15.18.1 BricsCAD Lisp

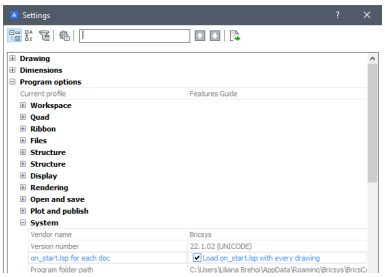
Migrating from AutoCAD

When you start migrating to BricsCAD, you'll find that there are virtually no differences in BricsCAD LISP versus OtherLISP. Your code loads and runs, and the functionality is identical. The primary differences when "porting" your apps will be a few minor setup steps, Command line structure, and possibly file locations. To start, let's review the setup and launch of your existing programs in BricsCAD.

If you are auto-launching your programs using **Acad.lsp** or **Acaddoc.lsp**, you will simply need to rename them as **on_start.lsp** or **on_doc_load.lsp** respectively.

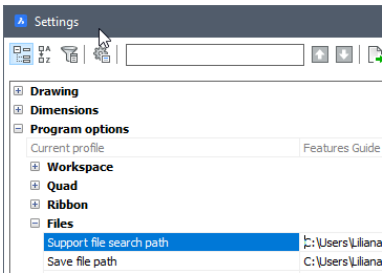
BricsCAD Settings Panel

To enable auto-launching your program set the ACADLSPASDOC system variable to 1 or simply toggle the option in *Settings / Program options / System*:



It is also recommended that you spend a little time familiarizing yourself with the Options/Setting menu in BricsCAD, as there are settings and options specific to BricsCAD environment.

The next step in the settings dialog would be to add your support folders to the Files search path, which is found within Settings / Program Options / Files:



It is recommended to use the support paths added as in the manner above and/or located one level below the current project folder. It is a good practice to use a “findfile” function to validate the file exists before proceeding with your commands, as in the method below:

```
(if (findfile "custom-code.lsp") (load "custom-code"))
```

This is one of the methods you may use to demand-load your functions within the **on_start.lsp** file:

```
(defun c:my-function ()                    ; this defines the command
(if (findfile "custom-code.lsp")           ; check if the file exists
(load "custom-code")                       ; if True, launch
(alert "Custom-code not in Support Path!") ; not True, so alert)
)                                          ; end if statement
)                                          ; end function
```

The command defun (DEfine FUNction) is the same name as the function inside of your **on_start.lsp** file and that function redefines the calling function when it’s launched.

15.18.2 Drawing Setup Lisp Routines for BricsCAD

Code sample

The below simple drawing setup routine could be adapted to your needs with minimal changes.

```
(defun C:MYSETUP ( / )                  ;define the command line function "mysetup"
  (setvar "cmdecho" 0)                  ;set display of prompts and input to OFF
  (command "_audit" "_y")               ;audit current drawing
  (command "_-purge" "_all" "*" "_n")   ;purge all unused styles, etc
  (command "_-layer" "_m" "1" "_c" "1" "" "")  ; create layer "1" color "1"
  (command "_-layer" "_m" "2" "_c" "2" "" "")
  (command "_-layer" "_m" "3" "_c" "3" "" "")
  (command "_-layer" "_m" "4" "_c" "4" "" "")
  (command "_viewres" "_y" "20000")     ;set view resolution to maximum
  (setvar "facetres" 10)                ;set facet resolution to max
  (setvar "filedia" 1)                  ;turn file dialogs on
  (setvar "attdia" 1)                   ;turn attribute dialogs on
  (setvar "attreq" 1)                   ;enable attribute editing on placement
  (setvar "cecolor" "bylayer")          ;set current color bylayer
  (setvar "clayer" "0")                 ;set current layer to "0"
  (setvar "maxsort" 5000)               ;set maximum layer sort to 5,000
  (command "_zoom" "_e" "_zoom" "0.95x")  ;zoom extents, then out a little
  (setvar "cmdecho" 1)                  ;set display of prompts and input to ON
  (princ)
)
```

Each individual line in this drawing setup routine could have been typed in at the command line in BricsCAD. By wrapping them into a Lisp routine, the **defun** (*define function*) makes the drawing setup routine “callable” from a menu or by typing its name on the BricsCAD command line.

If you don’t include the **defun**, the program will run once, when you load it by its file name. Using the **defun** and loading this routine within your **lsp** will make the function available at any time by typing its function name: **mysetup**.

The commands with the preceding dashes (e.g. **-layer** and **-purge**) are used to prevent the command’s dialog boxes from launching.

> ***Nota:*** Always use English command and options names to avoid fails on localized (non-English) BricsCAD versions.

A good practice when opening files from others is to execute an **audit** followed by a **purge** to remove any unused items.

Purge command could be run multiple times to remove nested items. See the below example:

```
(repeat 3 (command "-purge" "all" "*" "n"))
```

> ***Nota:*** **Purge** is done before adding the new, empty layers, as they would be removed if we did it after.

The **command** calls for setting layers could be combined into a single command statement, but it’s easier to read when each line stands alone.

The **setvar** calls for setting a system variable to a specified value, e.g. style settings for dimensions and text, unit settings, etc.

The **maxsort** setting was introduced for two reasons. First, **Maxsort** controls the maximum number of dialog box entries that will be sorted. The second reason is that the “default value” for **maxsort** is 1,000. With today’s computers and often found larger project files, that number can be too low. If you open drawings where the layer names won’t sort into alphabetical order, the default **maxsort** value could be the reason!

A more versatile Layer Maker for your drawing setup routine

The following example could be added to your version of mysetup.lsp. By creating a simple text file “Layers.txt” with an editor like Notepad, you can add a list of layer names to be created when the routine is run. This is an alternative to the hard-coded process shown above. By using the findfile function, BricsCAD will first search the current directory, then sequentially through your defined support folders to find the Layers.txt file. You can place your default layer setup text file in the support path and place alternative versions of it in individual project folders.

```
(if (not (findfile "layers.txt"))    ;"not" = looking for no result from findfile
(alert "can't find layers.txt")      ;No file? yell at the user and exit
(progn                               ;Otherwise, do the stuff in the progn group
(setq layerfile (open (findfile "layers.txt") "r"))  ;open layers.txt to "read"
(while (setq layername (read-line layerfile))        ;while reading each line, store the text to layername
(command "-layer" "m" layername "")                  ;make layer with the layername
)                                    ;end the while loop
(close layerfile)                    ;close the layers.txt file
)                                    ;end the progn
)                                    ;end the if
```

16. Modificare Entità

16.1 Modifying commands overview

Icon	Command	Description
	3DARRAY	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	ARRAY	Creates 2D and 3D arrays of entities in rectangular or polar patterns, or along a path.
	ARRAYCLASSIC	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	ARRAYPATH	Associatively distributes entity copies evenly along a path into multiple rows, and levels.
	ARRAYPOLAR	Associatively distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows, and levels.
	ARRAYRECT	Associatively distributes copies of entities into any number of rows, columns, and levels.
	BREAK	Splits an entity into two entities.
	CHAMFER	Creates chamfers at intersections of linear entities, defined by two lengths, or a length and an angle.
	CHANGE	Changes the properties of entities, through the Command line. This command has largely been superseded by the **Properties** panel.
	COPY	Draws duplicates of the selected entities.
	COPYGUIDED	After selecting the entities, their relation to their reference curves will automatically guide you to inserting the copy in similar locations.
	DIVIDE	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

Icon	Command	Description
	EXPLODE	Reduces complex entities to simpler ones, such as blocks into constituent parts, or polylines into lines and arcs.
	EXTEND	Lengthens lines, arcs, polylines or rays to meet another entity.
	FILLET	Fillets intersections; joins intersecting lines with an arc of any suitable radius.
	FLATTEN	Projects 2D and 3D entities onto the XY-plane of the current view.
	HATCHEDIT	Edits hatches and gradient fills in a dialog box.
	JOIN	Joins lines, lightweight and old-style 2D polylines, 3D polylines, arcs, elliptical arcs, polyarcs, splines, and helices at their common endpoints.
	LENGTHEN	Changes the length of a line, polyline, freehand entity or arc.
	MANIPULATE	Launches the Manipulator widget; rotates, moves, mirrors and scales entities. Keep the Ctrl key pressed while starting an action to manipulate a copy of the selected entities.
	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	MIRROR	Moves or copies the reflected image of selected entities about a line.
	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	MOVE	Moves entities to another location in the same drawing.
	MOVEGUIDED	After selecting the entities, their relation to their reference curves will automatically guide you to moving the entities in a similar location.
	OFFSET	Creates a parallel or offset copy of linear entities.
	OVERKILL	Deletes duplicate entities and overlapping lines, arcs, and polylines, and unifies partly overlapping or contiguous ones; optionally moves duplicates to a dedicated layer. Combines duplicate blocks; optionally purges duplicate block definitions.

Icon	Command	Description
	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	PROPERTIES	Opens the BricsCAD **Properties** panel.
	ROTATE	Rotates entities around a specified point.
	ROTATE3D	Rotates entities around an axis.
	SCALE	Changes the scale of existing entities, either enlarging them or reducing them proportionately in X, Y, and Z directions.
	SIMPLIFY	Reduces the number of vertices of polylines and hatch boundaries without changing their general shape. Simplified entities are easier to manipulate and can significantly reduce file size.
	SPLINEDIT	Edits a two-dimensional or three-dimensional spline.
	STRETCH	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	TRIM	Erases the portions of selected entities that cross a specified boundary.

16.2 Entity modification settings

Entity modification settings are found in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity modification**.

16.3 Match properties

The MATCHPROP command (short for "match properties") assigns the properties of one entity to other entities.

16.3.1 Copying properties between entities

1 Launch the MATCHPROP command.

2 **You are prompted**: Select entity to copy properties from:

3 Select the source entity.

4 **You are prompted**: Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text' 'Shadow Display' 'Hatch' 'Multileader' Select entities to copy properties to or [Settings/selection options (?)]:

5 (option) Choose the **Settings** option, then press Enter.

Adjust the settings, then click the **OK** button.

6 Select the target entities.

7 **You are prompted**: Select entities to copy properties to or [Settings/selection options (?)]:

8 Keep selecting entities or right-click to stop.

16.3.2 Copying properties between entities with BIM data

Whether the MATCHPROP command copies BIM data depends on the value of the BIMMATCHPROP system variable.

- **BimMatchProp = 0**: Match standard properties

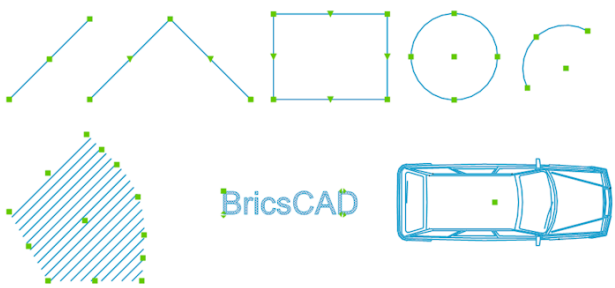
  Only standard properties will be copied, according to the currently selected options (see above).
- **BimMatchProp = 1**: Match standard and BIM properties

  BIM data, including IFC properties and user defined properties are copied, except:
  - BIM Type*
  - Composition/Profile
  - GUID
  - Quantities

* When using MATCHPROP between a BIM classified entity and an entity without any BIM data, the BIM type and the corresponding BIM properties will be copied along to the unclassified entity, except Composition/Profile, GUID and Quantities.

16.4 Grips editing

Grips are displayed as small colored squares at key-points on selected entities.



Whether grips are displayed or not and the appearance of grips are controlled through a series of grips settings in the **Settings** dialog box under **Drawing** > **Drafting** > **Display/Viewing** > **Grips**.

In complex drawings, using too many grips may clutter the display. The GRIPOBJECTLIMIT system variable allows you to suppress the display of grips if the number of selected entities exceeds a certain limit. If set to zero, all grips are displayed.

Grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

When the GEOMRELATIONS system variable is set, the geometric relation (perpendicular or tangent) between the entity being edited and other lines, polylines, arcs or circles is indicated (see Geometric Relations).

To select a grip, move the mouse over it, then click. Selected grips are displayed in a different color, which is defined by the HOTGRIP system variable. A selected grip is called a hot grip. By default, hot grips are displayed in red.

The **midpoint** grip on lines and infinite lines, the **startpoint** grip on rays, the **center** grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the **insertion point** grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

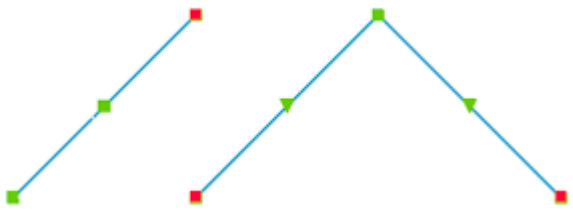
Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

16.4.1 Selecting multiple grips

1 Select all entities you want to edit.

2 Press and hold the Shift key, then click the first grip. The selected grip is displayed in red.

3 Keep the Shift key pressed, then click more grips. All selected grips are displayed in red.



4 Release the Shift key, then click the hot grip you want to use as the base point. The hot grips move simultaneously.

16.4.2 Grip Edit Command Options

When you start dragging a selected grip, BricsCAD displays a prompt in the command line:

Enter new point [Base point/Copy/Undo/eXit]:

You can choose an option using the keyboard, or in the **Prompt** menu.

Keyboard	Prompt Menu	Description
B	Base Point	You are prompted to specify a new base point, other than the selected grip.
C	Copy	Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies.
U	Undo	If the **Copy** option is chosen, deletes the latest copy.

Keyboard	Prompt Menu	Description
R	Reference	(Applies to Grip Rotate only) Allows you to specify a reference angle. Type the angle in the Command line or specify two points.
X	Exit	Aborts the grip edit command.

16.4.3 The grips editing context menu

The grips editing commands can be chosen by repeatedly hitting the space bar or the Enter key, or in a right-click context menu.

The command options are: stretch, move, mirror, rotate, scale.

The stretch command option is selected by default.

16.4.4 Stretching entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point.

4 All selected grips move simultaneously.

5 **You are prompted**: Enter new point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

16.4.5 Moving entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

16.4.6 Copying entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the move.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Move** in the context menu.
- Hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Move** is active.

You are prompted: Enter move point: Base point/Copy/Undo/eXit:

6 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

7 Do one of the following:
- Choose **Copy** in the context menu.
- Type **C**, then press **Enter**.

8 Do one of the following to define the displacement point:
- Specify a point.
- Type an angle and a distance in the dynamic entry fields.

9 Repeat the previous step to create more copies or press the Space bar or Enter to stop.

16.4.7 Mirroring entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the mirroring.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Mirror** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Mirror** is active.

You are prompted: Enter second point: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Specify a second point to define the mirror axis.

9 The selection is mirrored.

16.4.8 Rotating entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the rotation.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Rotate** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Rotate** is active.

You are prompted: Enter rotation angle: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

Specify a point.

8 Do one of the following to rotate the selection:
- Specify a point to define the rotation angle.
- Type the rotation angle in the Command line, then press Enter.

16.4.9 Scaling entities using grips

1 Select the entity or entities.

2 Select the grip(s) you want to manipulate.

3 Click the grip you want to use as the base point for the scaling.

4 The **Grip Stretch** command is launched.

5 Do one of the following:
- Right-click, then select **Scale** in the context menu.
- Repeatedly hit the **Space bar** or **Enter** key to cycle through the **Grips Editing** commands until **Scale** is active.

You are prompted: Enter scale factor: Base point/Copy/Undo/eXit.

6 (option) Choose the **Copy** option to keep the original(s).

7 (option) Choose the **Base point** option.

You are prompted: Enter base point.

8 Specify a point.

9 Type the scale factor in the Command line, then press Enter to scale the selection.

16.5 Selection methods

Before you start to use modification commands, you need to know how to compose a selection set. If the PICKFIRST variable is ON, you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command.

Using QUICK SELECT, you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

The SELECTIONPREVIEW and PREVIEWEFFECT system variables facilitate the selection procedure.

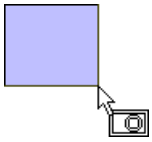
The SELECTIONPREVIEWFILTER system variable allows you to exclude entities from being selected: entities on locked layers, entities in Xrefs, tables, MTexts, hatches, entities in group.

16.5.1 Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click entities.
- Selected entity highlight and grips are shown.
- Area selection:
  - Click to define the first corner of a selection area.
  - Move the mouse to the left to define a **Crossing area**:

  - All entities which overlap or are completely inside the rectangle, are added to the selection set.
  - By default, the selection area is displayed in green with a dashed outline.
  - Click to define the first corner of a selection area.
  - Move the mouse to the right to define a **Window area**:

  - All entities which are completely inside, are added to the selection set.
  - By default the selection area is displayed in blue with a continuous outline.
- When working with 3D solids and surfaces, pressing the Ctrl key during window selection allows you to select faces or edges instead of the entire solid or surface. The icon next to the cursor indicates the selection mode:
  - Solids (default)

- Faces
- Edges

Depending on the value of the TIPS system variable, the Hotkey Assistant widget indicates the current selection mode:

- Press an hold the **Shift** key, then use one of the above methods to **remove** entities from the selection set.

Nota:

- The Selection Area system variables control the display of the area selection effects.
- When the PICKADD system variable is **Off**, you cannot add entities to a selection set.
- Press and hold the left mouse button to start window selection when the entity under the cursor highlights. This method is especially helpful for window selection in rendered 3D models.

16.5.2 Composing a selection set afterwards (post-pick)

If you launch a modification command when no selection set is active, **you are prompted** to select entities first. BricsCAD provides a range of selection methods to let you compose your selection set easily.

The various selection methods are:

- **Add to set**: Selected entities are added to the selection set.
- **Subtract from set**: Selected entities are removed from the selection set.
- **Picking**: Place the pickbox over a part of the object and click.
- **Select all entities**: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
- **Previous selection**: Re-use the previous selection set.
- **Last entity in drawing**: Select the most recent entity.
- **Window**: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a **rectangle**, **polygon** or a **circle**.
- **Fence**: All entities that cross a multi-segment line are selected.
- **Point**: Click a point to select all closed entities which enclose this point.
- **Select by Properties**: Displays the **Select by Properties** context menu to select entities by **Color**, **Layer**, **Linetype**, **Name**, **Thickness**, **Type**, **Value**, **Width** or **Handle**.
- **Undo**: Undoes the last selection action.
- **Location**: Displays the **Select Location** context menu.

16.5.3 Selecting overlapping entities

When entities lie on top of each or are obscured by other entities, it is difficult or sometimes impossible to select such entities..

Cycling through the overlapping entities, highlighting each entity one by one, helps you to select such entities.

Make sure SELECTIONPREVIEW is ON and/or the Quad cursor menu is active.

1 Place the cursor over the overlapping entities.

1 One of the entities highlights.

2 Repeatedly hit the Tab key.

3 The overlapping entities highlight one by one.

4 Click when the desired entity highlights.

5 Repeat the previous steps to select more entities.

Nota:

- Do not move the mouse while hitting the Tab key, because this will restart the highlight order.
- Hold down the Shift key to cycle the overlapping entities in reverse order.

16.5.4 Selecting subentities

By default the entire entity is selected. This method allows you to select subentities of compound entities, such as polylines, 3D polylines, regions, 3D solids, etc..

1 Place the cursor over the subentity.

2 When the entire entity highlights, hold down the Ctrl key.

3 Click when the subentity is highlighted. The properties of the selected subentity show in the **Properties** panel.

4 (option) Repeat the previous steps to select more subentities.

The following subentities can be selected: polyline segments, vertices and segments of 3D polylines, vertices, edges and faces of regions, surfaces, meshes and 3D solids.

16.6 Quick select

Using **Quick Select**, you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

After composing the filter criteria, you can choose to add the matching entities to a new selection set, add the entities to the current selection set or remove them from the current selection.

Quick Select applies to the entire drawing or to a previously created selection.

16.6.1 To create a selection set

1 Click the **Quick Select** switch button ( ) on the **Properties** panel.

The **Properties** panel switches to **Quick Select** mode.

If a selection set exists then the **Apply to current selection set** option is checked.

The **Selection list** button indicates the number of entities in the drawing, or the number of selected entities.

2 (option) Click the **Selection list** button, to see a list of all entity types in the current drawing or current selection set.

A list of all entities in the drawing and the number of entities of each type is displayed.

3 Select an entity type in the list, e.g. **Line**.

All Line properties show in the **Properties** panel.

4 Select a property you want to use as a filter, e.g. **Color**.

5 Specify a color.

6 By default all entities of this color will be selected.

7 Optionally, specify a different selection operator.

Click the = sign, then select an operator in the list.

8 Do one of the following:

- Click the **Add to new selection set** button ( ).

  The entities that match the selected criteria are selected.

- Click the **Add to the current selection set** button ( ).

  The entities that match the selected criteria are added to the current selection.

- Click the **Remove from current selection** set button ( ).

  The entities that match the selected criteria are removed from the current selection.

9 (option) Repeat the above procedure to further refine the selection set.

16.7 Draw order

When new entities overlap existing entities, they display and print on top of the previously drawn entities. The DRAWORDER command can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

1 Launch the DRAWORDER command.

2 **You are prompted**: Select entities to change the draw order:

3 Select the entities, then right-click or press enter.

4 **You are prompted**: Change draw order: Above/Under/Clear/Front/<Back>:

5 Choose an option.

6 Depending on your choice:

- If you have chosen **Above** or **Under**, the Command line reads: Select reference entity:

  Identify the reference entity. The display of the selection set is changed with respect to the

reference entity.

- If you have chosen **Front** or **Back**, the display of the selection set is changed accordingly.

16.7.1 Using Draw Order tools

Draw Order tools are available in the **Tools** menu and the **Draw Order** toolbar.

Icon	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)
	Send Under	Place the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back (HATCHTOBACK command).
	Text and Dimensions to Front	Brings all texts and/or dimension entities to front. Options: **Text / Dimensions / Both** can be chosen either in the Command line or in the prompt menu. The default option is **Both** (TEXTTOFRONT command).

16.8 The manipulator

The manipulator is a versatile grip tool to swiftly move, rotate, scale and mirror selected entities. Holding down the Ctrl key when starting the action keeps the original entities, thus creating copies. Upon selecting an entity, the manipulator is displayed at a user-controlled location and orientation. The location and orientation are controlled in a similar way to a dynamic UCS. When multiple entities are selected simultaneously, the manipulator is displayed at their geometric center. When a selection is built up step by step, the initial location of the manipulator is preserved. The manipulator can be moved to any desired location or orientation using its handles and the context menu options.

Depending on the operation type and if dynamic dimensions are active, a dynamic entry field is available to set the distance, angle or scale factor.

16.8.1 Adaptive snap step

The snap step size automatically adapts to the current screen zoom factor if SNAPTYPE= 2. The ADAPTIVEGRIDSTEPSIZE system variable controls the smallest available step size. The adaptive step size is used to dynamically adjust the step size in the Manipulator Ruler, which enables you to quickly modify entities using precise values, without having to key in numbers.

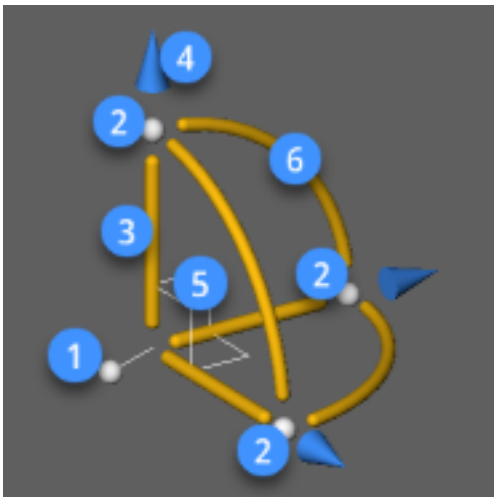
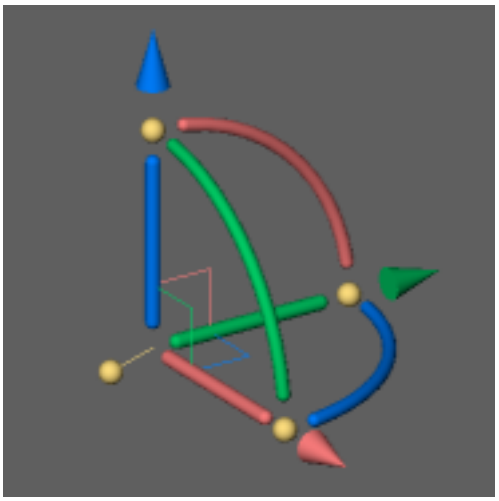
16.8.2 The manipulator settings

The manipulator settings can be found in the **Settings** dialog box under **Program Options** > **Manipulator**.

When the Manipulator is displayed, select **Manipulator Settings** in the right-click context menu.

16.8.3 The manipulator layout

Check the **Classic Color** option in the manipulator context menu for the classic layout. Uncheck the option for the default layout.

	
Default	Classic

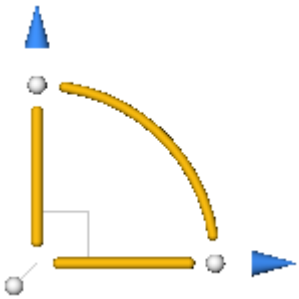
4 handles

- **1**: manipulator handle
- **2**: axis handle

12 command trigger zones

- **3**: axis
- **4**: arrowhead
- **5**: plane swatch
- **6**: rotation arc

When the current view is parallel to one of the main axes of the coordinate system, as in a plan view, the manipulator is simplified to a 2D version.



16.8.4 The location and orientation of the manipulator

The initial location and orientation of the 3D manipulator tripod depends on:

- The entity type.
- The point you click on the entity.

Entity Type	Selection Point	Manipulator - Anchor Point	Manipulator - Orientation
Line	Near endpoint	At endpoint	X-axis through the other endpoint
Line	Near midpoint	At midpoint	X-axis through the nearest endpoint
Polyline Line Segment	Near vertex	At vertex	X-axis through the other vertex of the selected line segment
Polyline Line Segment	Near segment midpoint	At the segment midpoint	X-axis through the nearest vertex
Polyline Arc Segment	On the arc segment	At the arc center	X-axis through the nearest vertex
Arc	Random	At the arc center	X-axis through the nearest endpoint
Circle	Random	At the circle center	X-axis parallel with the X axis of the current coordinate system
Spline	Random	At the nearest endpoint	Through the other endpoint
Region	On a line segment	At the nearest endpoint of the line segment	X-axis through the other endpoint

Entity Type	Selection Point	Manipulator - Anchor Point	Manipulator - Orientation
Region	On an arc segment	At the center of the arc segment	X-axis through the nearest endpoint of the arc segment
Text, MText, Block, Xref	Random	At the origin of the entity	X-axis at the rotation angle of the entity
3D Solid 3D Solid Face	On a face of the solid	At the nearest vertex	X-axis along the nearest edge XY-plane in the selected face
Multiple Entities	Random	At the geometric center of the selection set**	Parallel with the axes of the current coordinate system

* See Selection Modes to learn how to select a 3D solid face or the entire solid.

** The position of the manipulator does not change when more entities are added to the selection set.

16.8.5 Displaying the manipulator

The value of the MANIPULATOR system variable controls when the MANIPULATOR appears automatically:

- 0: Never
- 1: When entities are selected.
- 2: When the left mouse button is hold down longer when selecting an entity.

In order to display the Manipulator when multiple entities are selected:

- Select the entities one by one or use window selection, then long-click to select the last entity.
- Set the value of the MANIPULATOR system variable to 1.
- Compose the selection set, then launch the MANIPULATE command.

16.8.6 Moving the manipulator

Do one of the following:

- Long-click anywhere on the manipulator and move it to a new location.
- MANIPULATORHANDLE = 0: Click the manipulator handle (1) and move it to a new location*.
- The anchor handle stands out to avoid overlap with entity grips.
- MANIPULATORHANDLE = 1: Long-click the manipulator handle (1) and move it to a new location*.
- Place the cursor over the manipulator, then right-click and choose **Move** in the context menu and move it to a new location*.

* Specify the new location using entity snaps.

16.8.7 Rotating the manipulator

Select one of its axis handles (2) to rotate the manipulator. Pick a point or type a value in the dynamic entry field to specify the rotation angle.

Press the Shift key to rotate the Manipulator 90 degrees about its normal axis (Z-axis).

16.8.8 Reorienting the manipulator

1 Place the cursor over the manipulator, then right-click and choose **Reorient Manipulator** in the context menu.

   You are prompted: Specify origin of manipulator:

2 Pick a point.

3 The anchor point of the manipulator is placed at the specified point.

   You are prompted: Point on X-axis or <Accept>:

4 Pick a point.

5 The manipulator rotates about its X-axis.

   You are prompted: Point on the XY-plane with positive Y value or <Accept>:

6 Pick a point to define new position of the manipulator.

16.8.9 Aligning the manipulator with a coordinate system

Move the cursor over the manipulator, then right-click and select an option in the context menu:

- **Align with WCS**: the manipulator axes are forced to become parallel with the corresponding axes of the World Coordinate System.
- **Align with UCS**: if a UCS is currently the active coordinate system, the manipulator axes are forced to become parallel with the corresponding axes of the UCS.

16.8.10 Aligning the manipulator with the face of a solid

Dynamic UCS (DUCS) must be ON for this operation to succeed.

1 Place the cursor over the manipulator then right-click and choose **Move** in the context menu.

2 Hover over the face of a solid.

3 The XY-plane of the manipulator aligns with the solid face under the cursor.

4 Do one of the following:
   - Snap to a point on the selected face or on one of its edges.
   - Hit the Shift key to lock the XY-plane to the selected face, then snap to a point outside the face.

16.9 Moving entities

The MOVE command moves entities in the drawing.

The CUTCLIP command copies entities to the Clipboard, and then erases them from the drawing.

The PASTECLIP command (short for "paste clipboard") pastes entities from the Clipboard into the current drawing.

The PASTEORIG command pastes entities from the Clipboard into another drawing, using the coordinates of the source drawing.

The PASTEBLOCK command pastes entities from the Clipboard as a block into the current drawing.

Use CUTCLIP, then either PASTECLIP, PASTEORIG or PASTEBLOCK to move entities to another drawing.

The **Copy** option of the Grips Editing commands allows you to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

16.9.1 Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

1 Launch the MOVE command.

1 **You are prompted**: Select entities to move [selection options (?)]:

2 Select the entities, then right-click or press Enter.

3 The selection set is now attached to the cursor.

You are prompted: Enter base point [Displacement] <Displacement>:

4 Specify a point.

5 **You are prompted**: Enter second point or <Use base point as displacement>:

6 Specify a point.

7 The selection set is moved.

16.9.2 Moving entities between drawings

1 In the source drawing, select the entities you want to move.

2 Do one of the following:

- Right-click, then choose **Cut** in the context menu.

  Hold down the right-mouse button longer when the Quad is active.
- Choose **Cut** in the **Edit** menu.
- Press **Ctrl + X**.

The entities are deleted in the source drawing and copied to the Clipboard.

3 Switch to the target drawing.

4 Do one of the following:

- Choose **Paste** in the **Edit** menu.
- Press **Ctrl + V**.

The bottom left corner of the bounding rectangle of the selection is attached to the cursor.

5 Do one of the following:
   - Specify a point.
   - Press to paste the selection set using the coordinates of the source drawing.

Nota: Launch the UNDO command in the source drawing bring back the moved entities.

16.9.3 Moving a selection as a block

1 Select the entities you want to move.

2 In the source drawing, select the entities you want to move.

3 Do one of the following:
   - Right-click, then choose **Cut** in the context menu.

     Hold down the right mouse button longer when the Quad is active.
   - Choose **Cut** in the **Edit** menu.
   - Press **Ctrl + X**.

   The entities are deleted in the source drawing and copied to the Clipboard.

4 The entities are deleted in the drawing and copied to the Clipboard.

5 (option) Switch to the target drawing.

6 Choose **Paste as a Block** in the **Edit** menu or press **Ctrl + Shift + V**.

   The bottom left corner of the bounding rectangle of the selection is attached to the crosshairs in the target drawing.

7 Specify the insertion point.

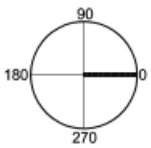
8 The selection is pasted as a block, using a default name.

9 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

16.10 Rotating entities

The ROTATE command rotates entities at a specified point along a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock (East) and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction, you can enter a negative value.



Nota:

- The **Angular Base** is controlled by the ANGBASE system variable, which in turn refers to the current UCS.

- The **Angle Direction** is controlled by the ANGDIR system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

16.10.1 General procedure

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Do one of the following:
- Specify a point.
- Type a value and press Enter.

The selection set is rotated.

16.10.2 Rotating a selection set relative to a base angle

1 Launch the ROTATE command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Rotation point:

6 Specify a point.

7 **You are prompted**: Rotation angle or [Base angle/Copy] <0>:

8 Choose the **Base angle** option.

You are prompted: Base angle <0>:

9 Do one of the following:
- Type a value and press Enter.
- Click two points to specify the base angle.

10 **You are prompted**: New angle <30.84>:

11 Do one of the following:
- Type a value and press Enter.

- Click a point.

The selection is rotated.

16.10.3 Rotating in 3D

1 Launch the ROTATE3D command.

1 **You are prompted**: Select entities to rotate [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to rotate [selection options (?)]:

4 Select more entities, or press Enter to stop selecting.

5 **You are prompted**: Select axis by [Entity/Last/View/Xaxis/Yaxis/Zaxis/2points] <2points>:

6 Press Enter to accept the 2points method to select the rotation axis.

7 **You are prompted**: Specify the first rotation axis point:

8 Specify a point.

9 **You are prompted**: Specify the second rotation axis point:

10 Specify a point.

11 **You are prompted**: Rotation angle or [Base angle/Copy]:45

12 Type a value and press Enter.

13 The selection is rotated.

16.11 Mirroring entities

The MIRROR command creates a mirrored copy of a selection set. The selection is mirrored along a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The MIRROR3D command creates a mirrored copy of selected entities in three-dimensional space. In this case the selection is mirrored along a mirror plane, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the XY, YZ, or XZ plane of the current UCS, or aligning the plane with the current view. You can choose to either delete or retain the original entities.

Nota: Whether text is mirrored or not by the MIRROR command is controlled by the MIRRTEXT system variable.

16.11.1 Mirroring entities along a line

1 Launch the MIRROR command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Start of mirror line:

6 Specify a point.

7 The selection set rotates dynamically.

You are prompted: End of mirror line:

8 Specify a point.

9 The selection is mirrored.

You are prompted: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

10 Decide whether or not to keep the original entities.

16.11.2 Mirroring entities along a plane

1 Launch the MIRROR3D command.

1 **You are prompted**: Select entities to mirror [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to mirror [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Define mirror plane by [Entity/Last/View/Z axis/X-Y plane/Y-Z plane/Z-X plane/3 points] <3 points>:

6 Press Enter to accept the 3 points method to define the mirror plane.

7 **You are prompted**: First point on plane:

8 Specify a point.

9 **You are prompted**: Second point on plane:

10 Specify a point.

11 **You are prompted**: Third point on plane:

12 Specify a point.

13 **You are prompted**: Delete the original entities? [Yes-delete entities/No-keep entities] <No-keep entities>:

14 Decide whether or not to keep the original entities.

16.12 Aligning entities

The ALIGN command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary, you can also scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.

16.12.1 Aligning in 2D

1 Launch the ALIGN command.

2 **You are prompted**: Select entities:

3 Select the entities, then right-click or press Enter.

4 **You are prompted**: Specify first source point:

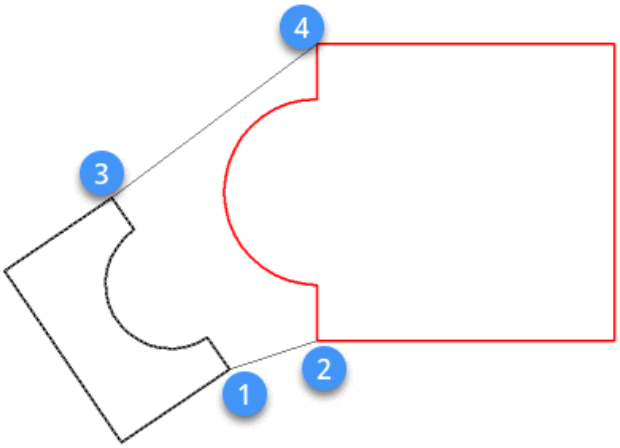
5 Snap to the first source point in the selection set (1).

6 **You are prompted**: Specify first destination point:

7 Snap to the first destination point on the reference entity (2).

8 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



9 Snap to the second source point in the selection set (3).

10 **You are prompted**: Specify second destination point:

11 Snap to the second destination point on the reference entity (4).

12 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

13 Right-click to use the ALIGN command in 2D.

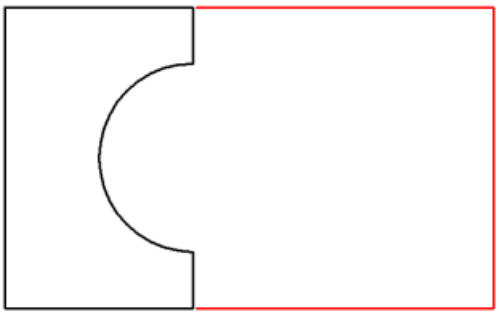
14 **You are prompted**: Scale objects based on alignment points [Yes/No] <No>:

15 Do one of the following:

- Press Enter if you don't want to scale the selection set.

- Type Y and press Enter to scale the selection set.
- Scaling adjusts the distance between the source points to the distance between the target points.

16



Result of the Align procedure with scaled selection set.

16.12.2 Aligning in 3D

1 Launch the ALIGN command.

1 **You are prompted**: Select entities:

2 Select the entities, then right-click or press Enter.

3 **You are prompted**: Specify first source point:

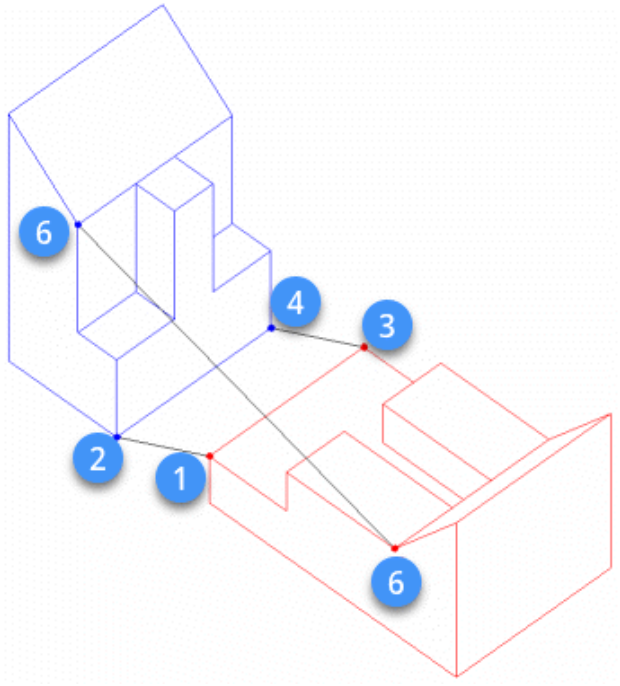
4 Snap to the first source point in the selection set (1).

5 **You are prompted**: Specify first destination point:

6 Snap to the first destination point on the reference entity (2).

7 A witness line is drawn between the source point and the target point.

You are prompted: Specify the second source point:



8 Snap to the second source point in the selection set (3).

9 **You are prompted**: Specify second destination point:

10 Snap to the second destination point on the reference entity (4).

11 A witness line is drawn between the source point and the target point.

You are prompted: Specify the third source point:

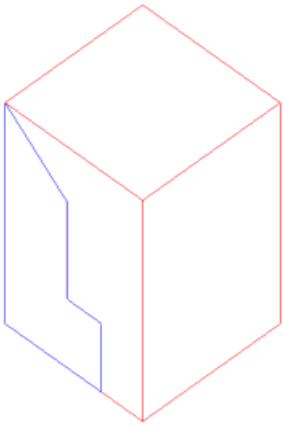
12 Snap to the third source point in the selection set (5).

13 **You are prompted**: Specify third destination point:

14 Snap to the third destination point on the reference entity (6).

15 A witness line is drawn between the source point and the target point.

The entity is aligned.



16.13 Changing entities

The CHANGE command lets you change the insertion point and orientation of texts and block inserts.

The **Properties** option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD Properties panel is much more versatile.

16.13.1 Changing text entities

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the text entities, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a new insertion point or press Enter to keep the original.

5 **You are prompted**: New text style Standard:

6 Type the name of a new text style or press Enter to keep the current style.

7 The text entity scales dynamically.

You are prompted: New height <current height>:

8 Do one of the following:
- Click to define the new text height.
- Type a value to specify the new text height.
- Press Enter to keep the current text height.

The text entity rotates dynamically.

You are prompted: New rotation angle <0>:

9 Do one of the following:
- Specify a point
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

You are prompted: New text <current text>:

10 Type a new text or press Enter to keep the current text.

11 When multiple text entities are selected, the procedure restarts for each entity.

16.13.2 Relocating blocks

1 Launch the CHANGE command.

1 **You are prompted**: Select entities to change [selection options (?)]:

2 Select the blocks, then right-click or press Enter.

3 **You are prompted**: Change point [special Entities/Properties...]:

4 Specify a point.

5 The block rotates dynamically.

You are prompted: New rotation angle <current angle>:

6 Do one of the following:
- Specify a point.
- Type a value and press Enter.
- Press Enter to keep the current rotation angle.

7 When multiple blocks are selected, the procedure restarts for each entity.

16.14 Copying entities

The COPY command duplicates entities within the current drawing using. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z).

Depending on the COPYMODE system variable, you can create multiple copies of the selection set or just one.

The COPYCLIP command copies entities to the clipboard, which you can then paste in the current drawing or in another drawing using the PASTECLIP or the PASTEBLOCK commands.

Nota: When DRAGSNAP is ON, dragged entities are displayed at the current entity snap location.

16.14.1 General procedure

1 Launch the COPY command.

1 **You are prompted**: Select entities to copy [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

4 **You are prompted**: Select entities to copy [selection options (?)]:

Current copy mode: Multiple

You are prompted: Enter base point [Displacement/mOde] <Displacement>:

5 Specify a point.

6 The copied selection moves dynamically.

You are prompted: Enter second point or [Array] <Use base point as displacement>:

7 Specify a point.

8 A first copy of the selection is created.

A second copy moves dynamically.

You are prompted: Enter second point [Undo/Array/Repeat/Exit]:

9 Do one of the following:
- Keep specifying points to create multiple copies.
- Press Enter to stop.

16.14.2 Using the Repeat option

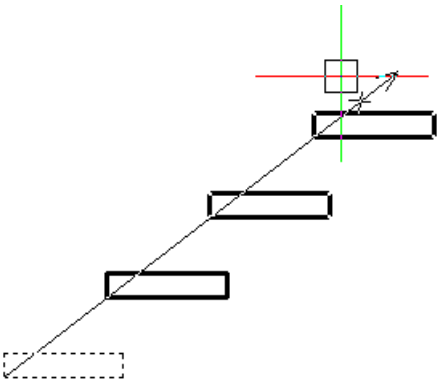
1 Execute steps 1-5 of the general procedure (see above).

1 **You are prompted**: Enter second point [Undo/Array/Repeat/Exit]:

2 Choose the **Repeat** option.

Multiple copies at regular intervals are created when you move the cursor.

You are prompted: Select end point (<ENTER> to key in number of repeats):



3 Do one of the following:

- Click when the desired number of copies is displayed.
- Press Enter.
- **You are prompted**: Enter number of repeats <1>:

  Type the desired number of copies, then press Enter.

16.14.3 Using the Array option

1 Execute steps 1-4 of the general procedure (see above).

1 **You are prompted**: Enter second point or [Array] <Use base point as displacement>:

2 Choose the **Array** option.

You are prompted: Enter number of items to array:

3 Specify the number of items.

4 The array is dynamically displayed.

You are prompted: Enter second point or [Fit]:

5 Do one of the following:

- Specify a point to define the distance between each item in the array.
- Choose the **Fit** option.

  You are prompted: Enter second point or [Array]:

  Specify a point to define the distance between the first and the last item in the array.

6 Do one of the following:

- Keep creating copies.
- Press Enter to stop.

16.14.4 Copying a selection as a block

1 Select the entities you want to copy as a block.

2 Launch the COPYBASE command.

3 **You are prompted**: Select base point:

4 Specify a point.

5 xx selected

6 (option) Switch to another drawing.

7 Launch the PASTEBLOCK command.

8 **You are prompted**: Select insertion point:

9 Specify a point.

10 The selection is pasted as a block.

11 (option) Launch the EXPBLOCKS command and rename the block in the **Drawing Explorer / Blocks** dialog box.

16.14.5 Copying entities between drawings

Entities can be copied between drawings using the following methods:

- COPYCLIP and PASTECLIP*: to copy the selection using a default base point.
- COPYCLIP and PASTEORIG: to copy the selection using the coordinates of the source drawing.
- COPYBASE and PASTECLIP*: to copy the selection using a user-defined base point.
- COPYBASE and PASTEBLOCK: to copy the selection as a block in the target drawing.

* Press Enter at the 'Specify insertion point' prompt to paste the selection using the coordinates of the source drawing (= equivalent of the PASTEORIG command).

16.15 Making parallel copies

The OFFSET command creates a copy of linear entities and aligns them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Depending on the selected options set by the SELECTIONMODES system variable, the OFFSET command accepts:

- an edge of a 3D solid if the **Select Edges** option is set.
- the outline of a 3D solid face if the **Select Faces** option is set.
- enclosed boundaries if the **Select Detected Boundaries** option is set.

Nota: The OFFSETGAPTYPE settings variable controls how potential gaps between segments are treated when closed polylines are offset.

16.15.1 Making a parallel copy at a specified distance

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:
   - Press Enter to accept the current distance.
   - Type a new distance in the Command line.
   - Define a new distance by specifying two points.

   You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 A preview of the parallel copy is displayed depending on the current cursor position with respect to the selected entity.

   You are prompted: Select side for parallel copy or [Both sides/Multiple]:

5 Click a point.

6 The parallel copy is created.

   If OFFSETERASE = ON, the original entity is deleted.

7 **You are prompted**: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

8 Do one of the following:
   - Select another entity.
   - Press Enter to stop.

16.15.2 Making a parallel copy through a point

1 Launch the OFFSET command.

1 **You are prompted**: Enter offset distance or [Through point/Erase/Layer] <Through point>:

2 Do one of the following:
   - Press Enter to accept the default **Through point** option (= when last used).
   - Choose the **Through point** option.

   You are prompted: Select entity/subentity or [Exit/selection options (?)] <Exit>:

3 Select a linear entity.

4 The parallel copy is dynamically displayed.

5 Click a point to create the parallel copy.

6 If OFFSETERASE = ON, the original entity is deleted.

   You are prompted: Select entity/subentity or [Undo/Exit/selection options (?)] <Exit>:

7 Do one of the following:
   - Select another entity.
   - Press Enter to stop.

16.16 Arraying entities

The **ARRAY** commands copy a selection set in a rectangular or polar (circular) pattern or along a linear entity. For a rectangular array, you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array, you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set around the center point or to keep its original orientation.

The ARRAYCLASSIC command creates a static polar or rectangular array of entities through a dialog box.

The -ARRAY command creates a static polar or rectangular array of entities.

The 3DARRAY command constructs static 3D rectangular arrays and rotated polar arrays.

The ARRAY command distributes entity copies in an evenly spaced associative rectangular array (ARRAYRECT command), a polar array (ARRAYPOLAR command) or a path array (ARRAYPATH command) array.

The ARRAYEDIT command allows you to edit associative arrays and their source entities.

16.16.1 Creating a static rectangular array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Rectangular**.

3 Click the **Select Entities** button.

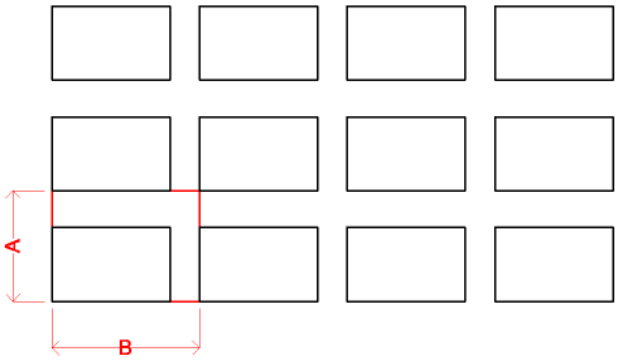
The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

5 The **Array** dialog box reopens.

6 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

The **Array** dialog box closes temporarily to let you pick a point in the drawing.



Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

7 Click the **OK** button to create the array.

Nota: Positive values in step 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

16.16.2 Creating a static polar array

1 Launch the ARRAYCLASSIC command.

1 The **Array** dialog box appears.

2 Under **Array Type**, choose **Polar**.

3 Click the **Select Entities** button.

The **Array** dialog box closes temporarily to let you select entities.

4 Select the entities, then right-click or press Enter.

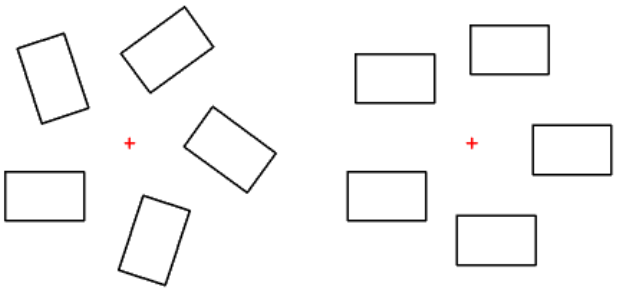
5 The **Array** dialog box reopens.

6 Choose a **Calculate** option: **Angle between**, **Angle to fill** or **Number of items**.

7 Fill out the **Settings** fields or press the **Pick Points** buttons to define the settings graphically.

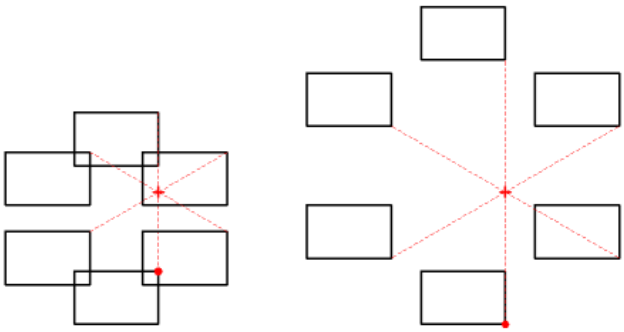
The **Array** dialog box closes temporarily to let you pick a point in the drawing.

8 (option) Check the **Rotate items as copied** option.



Entities rotated (left) or not (right) about the center point of the array.

9 (option) Define an **Entity base point**.



Polar array with base point (dot) and center point (cross).

10 Click the **OK** button to create the array.

16.16.3 Creating a static 3D rectangular array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Rectangular** option.

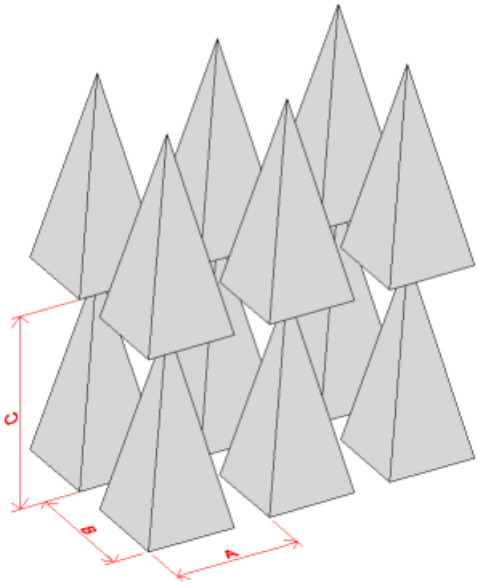
You are prompted: Number of rows in the array <1>:

7 Specify the number or rows.

8 **You are prompted**: Number of columns <1>:

9 Specify the number of columns.

10 **You are prompted**: Number of levels <1>:



Rectangular 3D Array:

row spacing A, column spacing B, level spacing C

11 Specify the number of levels.

12 **You are prompted**: Vertical distance between rows:

13 Type a value or click two points.

14 **You are prompted**: Horizontal distance between columns:

15 Type a value or click two points.

16 **You are prompted**: Depth between levels:

Type a value or click two points.

The array is created.

Nota: Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

16.16.4 Creating a static 3D polar array

1 Launch the 3DARRAY command.

1 **You are prompted**: Select entities to array [selection options (?)]:

2 Select an entity.

3 Entities in set: 1

You are prompted: Select entities to array [selection options (?)]:

4 Select more entities or press Enter to stop selecting.

5 **You are prompted**: Type of array [Polar/Rectangular] <Rectangular>:

6 Choose the **Polar** option.

You are prompted: Number of items to array or [enter to specify Angle between items] <enter to specify Angle between items>:

7 Specify the number of items.

8 **You are prompted**: Angle to fill (+ for ccw, - for cw <360>):

9 Type a value or press Enter to accept the default.

10 **You are prompted**: Rotate entities around the array? [Yes/No] <Yes>:

11 Choose **Yes** or **No**, or press Enter to accept the default.

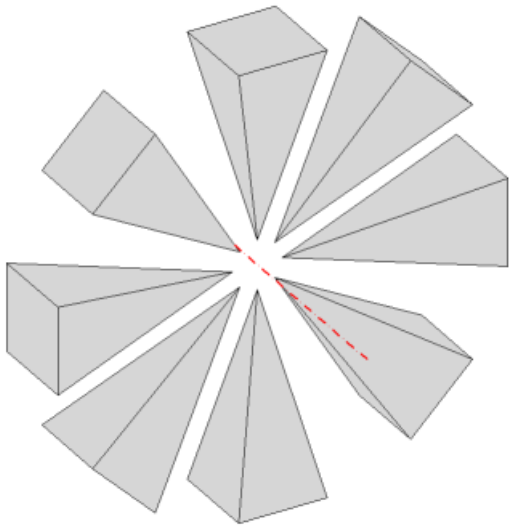
You are prompted: Center point of array:

12 Specify a point.

13 **You are prompted**: Specify second point along central axis of array:

14 Specify a point.

15 The array is created.



Polar 3D array around a horizontal axis (red).

16.17 Extending entities

The EXTEND command extends entities to a boundary, which is defined by one or more other entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).

When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

16.17.1 Extending entities

1 Launch the EXTEND command.

1 **You are prompted**: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

2 Select a boundary entity.

3 Entities in set: 1

You are prompted: Select boundary entities for extend [selection options (?)] <ENTER to select all>:

4 Select more boundary entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase]:

6 Select an entity:

7 The entity extends to the boundary entity.

You are prompted: Select entity to extend or shift-select to trim or [Fence/Crossing/Edge mode/Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Nota:

- Entities which are selected when you launch the EXTEND command will be used as boundary entities.
- Hold down the Shift key when you select an entity to trim the line by the boundary entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.
- If you select an entity near the end that cannot make an extension to one of the boundary entities, the EXTEND command is aborted.

16.18 Trimming entities

The TRIM command clips or trims entities by cutting entities.

If the EDGEMODE system variable is **On**, entities extend to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the PROJMODE system variable lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view pane
- True 3D mode (No projection).

When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

Entities in a block can be selected as cutting entities.

16.18.1 Trimming entities

1 Launch the TRIM command.

1 **You are prompted**: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

2 Select a cutting entity.

3 Entities in set: 1

You are prompted: Select cutting entities for trim [selection options (?)] <ENTER to select all>:

4 Select more cutting entities or press Enter to stop selecting.

5 **You are prompted**: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase]:

6 Select an entity.

7 The entity is trimmed by the cutting entity.

You are prompted: Select entity to trim or shift-select to extend or [Fence/Crossing/Edge mode/Projection/eRase/Undo]:

8 Select more entities or press Enter to stop.

Nota:

- Entities which are selected when you launch the TRIM command will be used as cutting entities.
- Hold down the Shift key when you select an entity to extend the line to the cutting entity.
- Use the Fence or Crossing selection method to select a series of entities simultaneously.

16.19 Lengthening entities

The LENGTHEN command changes the length of lines, open polylines and arcs dynamically, incrementally or relatively. You can also modify the included angle of arcs.

16.19.1 Changing the length dynamically

1 Launch the LENGTHEN command.

1 **You are prompted**: Edit length: Select entity to list length or [DYnamic/Increment/Percent/Total]:

2 Choose the **Dynamic** option.

You are prompted: Select entity to change or [edit Mode]:

3 Click the entity near the endpoint you want to lengthen.

4 The entity is lengthened (shortened) dynamically.

You are prompted: New end point for line:

5 Specify a point to define the new length.

6 **You are prompted**: Select entity to change or [edit Mode/Undo]:

7 Do one of the following:
- Edit another entity.
- Press Enter to stop.

Nota: The **Geometry** properties in the **Properties** panel allow you to edit the total length of lines and arcs and the start and end angle of arcs.

16.20 Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities, e.g. whether grips are displayed or not, is controlled by a number of settings.

16.20.1 Selection settings

Selection settings sit in the **Selection** settings group of the **Program Options** settings category in the **Settings** dialog box.

Name	System Variable	Description
Highlight	HIGHLIGHT	Determines whether or not entities highlight on screen when selected.
Image highlight	IMAGEHLT	Determines whether the entire raster image highlights or only the raster image frame when selected.

Name	System Variable	Description
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. **ON**: The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. **OFF**: You cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the **Shift** key, you can add entities. If you select an entity that was already selected while pressing the **Shift** key, this entity is removed from the selection set.
Pick automatic	PICKAUTO	Controls automatic window selection (inside or crossing) when selecting entities. The value is stored as a bitcode using the sum of the selected options. negative: Disables window selection, saving current value. 1 = draws a selection window if the mouse is over a blank area of the drawing when clicked. 2 = draws a selection window if the mouse is over an entity when dragged.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor (). If you select an entity by clicking the **Pick Box** must touch or overlap the entity. The default size of the **Pick Box** is 3.
Pick drag	PICKDRAG	Controls the method of drawing a selection window. **ON**: Allows you to define a selection window by dragging: press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. **OFF**: Define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. **ON**: Allows you to first compose a selection set, then launch a modification command. **OFF**: You must first start the command, then compose the select entities.

Name	System Variable	Description
Pick style	PICKSTYLE	Controls the selection of groups and associative hatches. 0 = No group selection, nor associative hatch selection. 1 = Group selection: if a member of a selectable group is selected, all members of the group are selected. 2 = Associative hatch selection: the hatch and its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet). 3 = Group selection and Associative hatch selection.
Match options for SELECTSIMILAR	SELECTSIMILARMODE	Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR command.
Selection area	SELECTIONAREA	Controls the display of selection area effects.
Selection area opacity	SELECTIONAREAOPACITY	Controls the transparency of the selection area: the higher the value, the more opaque the selection area.
Crossing area color	CROSSINGAREACOLOR	Specifies the color for crossing selection areas.
Window area color	WINDOWAREACOLOR	Specifies the color for window selection areas.
Selection preview display	SELECTIONPREVIEW	Controls when entities highlight when the cursor hovers over them: 0 = never; 1 = when no commands are active; 2 = when a command prompts for entity selection; 3 = both, when no commands are active and when a command prompts for entity selection. When the QUAD cursor menu is active, SELECTIONPREVIEW is neglected, unless a command prompts for entity selection.
Selection preview effect	PREVIEWEFFECT	Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported).

Name	System Variable	Description
Selection preview filter	PREVIEWFILTER	Controls which entity types are excluded from selection previewing. The options are: Entities on locked layers Entities in Xrefs Tables MText entities Hatches Groups
GL Selection Highlight Style	HIGHLIGHTEFFECT	Specifies the highlight method: 0 = dashed lines 1 = use different color 2 = use thickened line 3 = use different color and thickened line.
Selection Highlight color	HIGHLIGHTCOLOR	Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3.
Selection Modes	SELECTIONMODES	Defines which subentities should highlight in selection preview. The variable is stored as a bitcode using the sum of the values of all selected options. Click the buttons on the **Selection Modes** toolbar to set the value of SELECTIONMODES. A pressed button indicates the option is selected. 1 = Enable detection of 3D solid edges ( ) 2 = Enable detection of 3D solid faces ( )* 4 = Enable boundary detection ( ). Controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids highlight when no commands are active. The BOUNDARYCOLOR system variable specifies the color for highlighting detected boundaries. * Hold down the Ctrl key to invert the status of the **Select Faces** option: if ON it is possible to select 3D solids, if OFF it is possible to select solid faces. Repeatedly pressing the Tab key, while keeping the cursor at the same position, highlights all included subentity types one by one. See also Window Selection.
Constraint Bar Display	CONSTRAINTBARDISPLAY	Controls the visibility of a geometric constraint at its creation or selection.

Name	System Variable	Description
Dynamic Constraint Mode	DYNCONSTRAINTMODE	Controls the visibility of hidden dimensional constraints when the constrained entity is selected. When the cursor is over an entity with a dimensional constraint, the blue constraint glyph is displayed.
Preview selection delay	PREVIEWDELAY	Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor.

16.20.2 Grips settings

If the Grips system variable is different from zero, grips are displayed on selected entities.

Grips settings sit under **Display/Viewing** in the **Drafting** settings group of the **Drawing** settings category in the **Settings** dialog.

Grips can be used to manipulate entities (see Grips Editing).

Name	System Variable	Description
Grips	GRIPS	The value of GRIPS can be 0, 1 or 2. 0 : No grips. 1 : Display grips. Polyline endpoint and midpoint grips. 2 : Display grips. No midpoint grips on polyline segments.
Grips in blocks	GRIPBLOCK	Toggles the display of grips on entities in blocks on/off.
Grip color	GRIPCOLOR	Sets the color of unselected grips. Default value is green (index color no. 72).
Selected grip color	GRIPHOT	Sets the color of selected grips, so called 'hot grips'. Default value is red (index color no. 240).
Hover grip color	GRIPHOVER	Sets the color of an unselected grip when the cursor pauses over it. Default value is blue (index color no. 150).
Grip size	GRIPSIZE	Sets the size of the grip box, expressed in pixels. Default value = 4.
Grip object limit	GRIPOBJLIMIT	Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT. If set to 0 (zero), all grips are displayed.

Name	System Variable	Description
Grip tips	GRIPTIPS	(Not supported yet) Determines whether grip tips are displayed when the cursor hovers over grips on custom entities or dynamic blocks that support grip tips.
Grips attraction	ENABLEATTRACTION	Toggles the magnetism of grips on/off.
Grips attraction distance	ATTRACTIONDISTANCE	Sets the grips attraction distance: if the cursor is within this distance, the cursor is attracted by the grip and jumps to it.

16.21 Stretching entities

The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.

16.21.1 Stretching entities

1 Launch the STRETCH command.

1 **You are prompted**: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/selection options (?)]:

2 Use a crossing window (or crossing polygon) to specify the points that need to be moved.

3 Specify the bottom right corner of the crossing window.

A green rectangle is dynamically displayed.

You are prompted: Opposite Corner:

4 Specify the top left corner of the crossing window.

5 Entities in set: xx.

You are prompted: Select entities to stretch by [Crossing window/Crossing Polygon/Remove/Add/selection options (?)]:

6 Do one of the following:

- Use a crossing window (or crossing polygon) to specify the points that need to be added or removed from the selection.
- Right-click or press Enter to confirm the current selection.

You are prompted: Base point of displacement:

7 Specify a point.

8 The selected points move with the cursor.

You are prompted: Second point of displacement:

9 Specify a point to confirm the stretch distance.

16.22 Scaling entities

The SCALE command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the Command line. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is 3/2 of the original. The base scale and the new scale can also be defined graphically in the drawing.

16.22.1 Scaling a selection set

1 Launch the SCALE command.

1 **You are prompted**: Select entities to scale [selection options (?)]:

2 Select entities.

3 Entities in set: xx

You are prompted: Select entities to scale [selection options (?)]:

4 Select more entities or press Enter to continue.

5 **You are prompted**: Base point:

6 Specify a point.

7 The base point is not moved in the scaling.

You are prompted: Scale factor or [Reference/Copy] <current scale*>:

8 Type a scale factor or press Enter to accept the previously used value.

* The previously used scale factor in a drawing is remembered.

16.22.2 Scaling a selection set using a reference scale

1 Repeat steps 1 through 4 of the procedure above.

1 **You are prompted**: Scale factor or [Reference/Copy] <current scale*>:

2 Choose the **Reference** option.

You are prompted: Reference scale <1>:

3 Click the base point again (see step 3 in the previous procedure) (1).

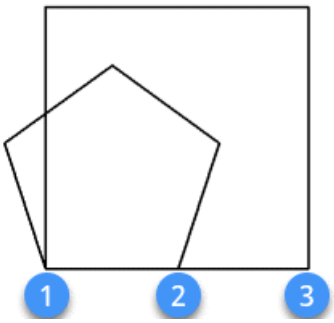
4 **You are prompted**: Second point:

5 Specify a point. (2)

6 The distance between the base point and the second point will be used as the reference scale.

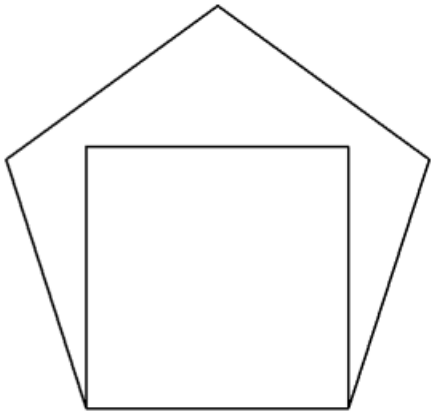
The selection scales dynamically.

You are prompted: New scale factor or [Points] <2>:



7 Do one of the following:
- Type a value.
- The distance between the reference points is set to the specified value.

  The size of the selection set is scaled accordingly.
- Specify a point to position the second reference point (see step 4) (3).



16.23 Breaking entities

The BREAK command removes a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point. However, you can use the **First** option to select a break point different from the one that selects the entity.

16.23.1 To break an entity

1 Launch the BREAK command.

1 **You are prompted**: Select entity to break [selection options (?)]:

2 Select the entity.

3 **You are prompted**: Select second breakpoint or [First break point/same as first point (@)]:

4 Select the **First** break point option.

You are prompted: First break point:

5 Specify a point.

6 **You are prompted**: Second break point:

7 Specify a point.

8 The part of the entity between the two break points is removed.

16.24 Joining entities

The JOIN command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

When you join colinear lines, the farthest endpoints remain at their existing locations. BricsCAD draws a new line between these points.

The resulting entity type depends on the types of the input entities and their coplanarity.

16.24.1 General procedure

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select source entities, then right-click to stop selecting.

3 BricsCAD reports the result:

4 37 entities joined into 10 polylines, 4 entities discarded from the operation

16.24.2 Joining two arcs

Coplanar arcs, sharing midpoint and radius (= lie on the same circle), are joined counterclockwise, therefore the result depends on the selecting order.

1 Launch the JOIN command.

1 **You are prompted**: Select source entity or multiple entities to join at once [selection options (?)]:

2 Select the first arc.

3 **You are prompted**: Select entities to join [selection options (?)]:

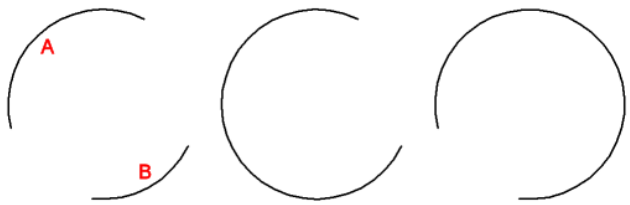
4 Select the second arc.

5 Entities in set: 1

6 **You are prompted**: Select entities to join [selection options (?)]:

7 Right-click.

8 BricsCAD reports: 2 arcs joined into 1 arc.



start situation (left) and result when selecting A first (middle) or B first (right).

16.25 Chamfering entities

The CHAMFER command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In BricsCAD you can choose between two chamfer methods:

- **distance-distance**: specify how far to trim the entities back from their intersection
- **distance-angle**: specify the length of the chamfer and the angle it forms along the first entity.

The current chamfer method, distances and angle are saved through a series of system variables:

- CHAMMODE: sets the chamfer method
- CHAMFERA: defines the first distance
- CHAMFERB: defines the second distance
- CHAMFERC: defines the chamfer length
- CHAMFERD: defines the chamfer angle
- TRIMMODE: defines whether or not to keep the original length of the chamfered entities or to trim or lengthen them to the endpoints of the chamfer line.

You can set the values in the **Settings** dialog box under **Drawing** > **Drafting** > **Entity Modification** > **Chamfer/Fillet**. The **Settings…** option of the CHAMFER command allows you to edit the variables in the CHAMFER command procedure.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer all vertices of the polyline. When chamfering a polyline, the chamfer line is joined with the polyline segments if TRIMMODE=ON.

16.25.1 Chamfering using the distance-distance method

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings…** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Directional/Select second entity (select with pressed SHIFT to make corner)

[Directional]:

5 Select the second entity.

6 The chamfer is created.

16.25.2 Chamfering using the length-angle method

1 Launch the CHAMFER command.

1 Chamfer (length=20, angle=45) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Select the first entity.

4 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

5 Select the second entity.

6 The chamfer is created.

16.25.3 Chamfering all vertices of a polyline

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 (option) Choose the **Settings...** option, edit the Chamfer variables in the **Settings** dialog box.

3 Choose the **Polyline** option.

You are prompted: Select 2D polyline to chamfer:

4 Select the polyline.

5 The polyline is chamfered.

Nota: When the chamfer method is **distance-angle**, the direction of the polyline defines which is the first entity of a vertex. Use the **Reverse direction** option of the PEDIT command if necessary to get the desired result.

16.25.4 Removing a chamfer

1 Launch the CHAMFER command.

1 **You are prompted**: Chamfer (dist1=15, dist2=15) Select first entity [chamfer Settings.../Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/selection options (?)]:

2 Select the first entity.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Hold down the Shift key, then select the second entity.

5 The chamfer is removed.

Nota: You can use this method to create a corner between two lines or an end segment of a polyline. When the two entities intersect, the part you click is kept.

16.26 Filetting entities

The FILLET command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

The TRIMMODE system variable defines whether or not to keep the original length of the filleted entities or to trim or lengthen them to the endpoints of the fillet arc.

16.26.1 Filleting two entities or polyline segments

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Select the first entity or polyline segment.

6 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

7 Select the second entity or polyline segment.

8 The fillet is created.

16.26.2 Filleting all vertices of a polyline

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Choose the **Radius** option.

You are prompted: Fillet radius <current radius>:

3 Type a value.

4 **You are prompted**: Fillet (radius=new radius): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

5 Choose the **Polyline** option.

You are prompted: Select 2D polyline to fillet [selection options (?)]:

6 Select the polyline.

7 The polyline is filleted.

16.26.3 Filleting two parallel lines

1 Launch the FILLET command.

1 **You are prompted**: Fillet (radius=10): Select first entity or [fillet Settings.../Polyline/Radius/Trim/Undo/Multiple/selection options (?)]:

2 Select the first line.

3 **You are prompted**: Select second entity (select with pressed SHIFT to make corner):

4 Select the second line.

5 The fillet is created.

16.27 Editing polylines

Use the PEDIT command to:

- Convert an entity into a polyline
- Fit a curve to a polyline
- Taper a polyline uniformly along its length
- Reverse the direction of a polyline

Edit a polyline in the **Properties** panel to:

- Open or close a polyline
- Apply a uniform width to a polyline
- Change the width of a polyline segment
- Set the linetype mode (PLINEGEN system variable)

Use the edit polyline segments tools in the Quad to:

- Insert a vertex
- Add vertices at the start or end
- Modify the bulge factor of a segment
- Stretch a vertex or a segment
- Delete a segment

Use grips editing to:

- Move a vertex
- Move a segment

16.27.1 Launching the PEDIT command

Do one of the following:

- Double-click a polyline.
- Click the Edit Polyline tool in the **Modify** menu.
- Click the Edit Polyline tool on the **Modify** toolbar.

16.27.2 Converting an entity into a polyline

1 Launch the PEDIT command.

2 Select the entity.

You are prompted: The selected entity is not a polyline. Turn it into one? [Yes-turn into polyline/No-leave as is] <Yes-turn into polyline>:

3 Press Enter or right-click.

4 **You are prompted**: Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/Spline/Taper/Width/Undo] <eXit>:

5 Choose an option or right-click to stop.

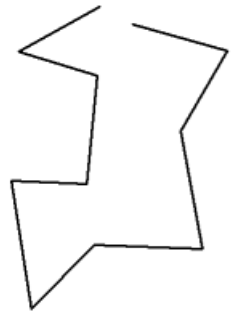
16.27.3 Fitting a curve to a polyline

1 Launch the PEDIT command.

2 Select the polyline.

3 Do one of the following:
- Choose the **Fit** option.
- Choose the **Spline** option.

4 Press Enter or right-click to conclude the PEDIT command.

		
Original polyline	After applying Fit	After applying Spline

Nota:

- Use the **Decurve** option to restore the original polyline.
- The SPLINESEGS (Spline Segments) system variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

16.27.4 Tapering a polyline uniformly along its length

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Taper** option.

You are prompted: Enter new starting polyline width <current width>:

4 Type a value.

5 **You are prompted**: Enter new ending polyline width <current width>:

6 Type a value.

7 The polyline is tapered.

8 Press Enter or right-click to conclude the PEDIT command.

16.27.5 Reversing the direction of a polyline

1 Launch the PEDIT command.

2 Select the polyline.

3 Choose the **Reverse direction** option.

The direction is reversed.

4 Press Enter or right-click to conclude the PEDIT command.

16.27.6 Opening or closing a polyline

1 Select the polyline.

2 Select the **Closed** property under **Misc** in the **Properties** panel.

3 Choose **No** or **Yes**.

Or

1 Double-click the polyline.

2 Type C to close, or O to open.

16.27.7 Applying a uniform width to a polyline

1 Select the polyline.

2 Select the **Global width** property under **Geometry** in the **Properties** panel.

3 Type the desired width.

16.27.8 Changing the width of a polyline segment

1 Select the polyline.

2 Select the **Vertex** property under **Geometry** in the **Properties** panel.

An X mark is displayed at the start vertex of the polyline.

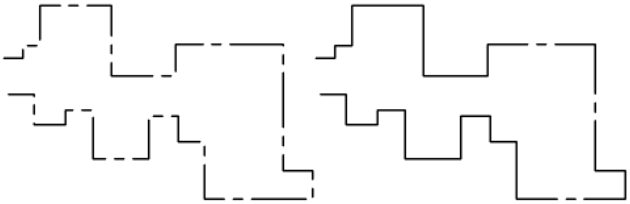
3 Click the arrow keys at the right hand side of the vertex property field to set the X marker at the start vertex of the segment you want to change the width of.

4 Select the **Start width** property and type the desired width.

5 Select the **End width** property and type the desired width.

16.27.9 Setting the linetype mode

The Linetype generation mode (PLINEGEN system variable) controls how a dashed linetype is applied to a multi-segment polyline. When disabled, dashed linetypes are applied per segment: start/end with a dash at each vertex.



PLINEGEN On (left) and Off (right)

1 Select the Polyline.

2 Select the **Linetype generation** property under **Misc** in the **Properties** panel.

3 Select **disabled** or **enabled**.

16.27.10 Using the edit polyline segments tools in the Quad

- Add vertex: adds a vertex to the segment under the cursor.
- Add vertex at end: adds as a new vertex at the end of the polyline.

  This option is only available if the cursor is on one of the endpoints of a polyline.
- Adjust bulge: modifies the bulge factor of the segment under the cursor.
- Delete segment: deletes the segment under the cursor.
- Stretch segment: moves the segment under the cursor.
- Stretch vertex: moves the vertex closest to the cursor.

1 Place the cursor over the segment you want to edit.

1 The entire polyline is highlighted.

2 Hold down the Ctrl key.

3 Only the segment under the cursor is highlighted.

The editing tools are available in the Quad.

4 Select a tool, then follow the instructions in the Command line.

16.28 Exploding entities

The EXPLODE command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

MText entities explode into single line texts.

In general, exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
- Colors and linetypes assigned BYBLOCK may appear different after exploding an entity, because they will adopt the default color and linetype.

The TXTEXP command explodes text entities into polylines. You can then use the JOIN command to create a single polyline for each letter. For some letters, such as a, e or p, a second polyline is needed for the 'hole').

16.28.1 Exploding entities

Do one of the following:

- Launch the EXPLODE command.
- Select entities, then press Enter.
- Select entities, then choose **Explode** in the **Modify** command group in the Quad.

16.29 Measuring entities

The MEASURE command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The markers are placed starting at the closest endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

16.29.1 Measuring an entity using points

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Do one of the following:
   - Type a value.
   - Specify two points.

   The points are placed along the entity at the specified interval.

Nota: The visibility of the points depends on the current value of the PDMODE system variable.

16.29.2 Placing blocks a specified interval along an entity

1 Launch the MEASURE command.

1 **You are prompted**: Select entity to measure:

2 Select an entity.

3 **You are prompted**: Segment length [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

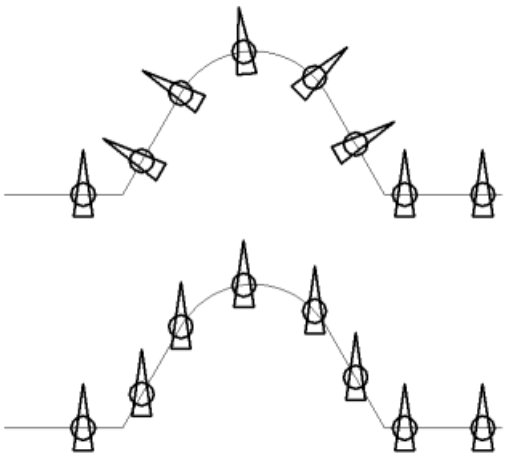
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

8 Choose whether or not to align the blocks.

9 **You are prompted**: Segment length:

10 Do one of the following:
- Type a value.
- Specify two points.

The blocks are placed along the entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

16.30 Dividing entities

The DIVIDE command places markers - points or blocks - along a selected entity. The markers divide the entity into a specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.

16.30.1 Dividing an entity using points

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Type a value, then press Enter.

5 The entity is divided by points.

Nota: The visibility of the points depends on the current value of the PDMODE system variable.

16.30.2 Dividing an entity using blocks

1 Launch the DIVIDE command.

1 **You are prompted**: Select entity to divide:

2 Select an entity.

3 **You are prompted**: Number of segments [insert Blocks]:

4 Choose the **Blocks** option.

You are prompted: Name of block to insert [? to list]:

5 (option) Press the ? key to see a list of available blocks in the drawing.

6 Type the name of a block.

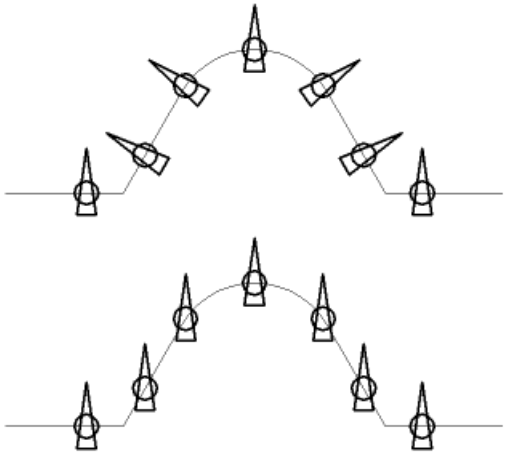
7 **You are prompted**: Align blocks with entity? [Yes-align blocks/No-do not align] <Yes-align blocks>:

8 Choose whether or not to align the blocks.

9 **You are prompted**: Number of segments:

10 Type a value, then press Enter.

11 The entity is divided by blocks.



Blocks aligned (top) or not aligned (bottom)

17. Stampa e plottaggio

17.1 General procedure for printing

The PLOT and PRINT commands plot drawings to printers and to files, through the **Print** dialog box.

The -PLOT command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

The QPRINT command prints the drawing with the default plot configuration, without displaying the **Print** dialog box.

The PAGESETUP command creates and edits page setups for plotting drawings in the **Drawing Explorer** dialog box.

A standard print procedure consists of the following steps:

- Choose either a **Layout** tab or the **Model** tab.
- Open the **Print** dialog.
- Select a **Page Setup name**. (1)

The following steps are optional and only necessary if you want to use overrides on the selected page setup definition.

- Choose a **Printer or Plotter configuration**. (2)
- Choose a **Plot style table**. (3)
- Choose a **Paper size**. (4)

  The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the paper size unit. (5)
- Choose the **Drawing orientation**.(6)
- Define the **Plot offset**. (7)
- Specify the **Plot area** (the area to plot). (8)
- Define the **Plot scale**. (9)
- Check the **Plot options**. (10)
- When printing model space, choose a **Shade plot** mode. (11)

  This option is disabled when printing a paper space layout.
- Check/uncheck the **Print to file** option. (12)
- Specify the **Number of copies**. (13)
- Click the **Preview** button to see a preview of the plot output.
- When in a layout, click the **Apply** button to update the layout.
- Click the **Print** button to start printing.

Nota: All print settings and options are saved in the **Model** and each of the **Layouts**.

Nota:

On Mac or Linux, printing will always print to PDF. So a PDF document will be generated that then still needs to be sent to the printer for physical printing.

On MAC and Linux there is a relevant PAGESETUP command.

17.1.1 Opening the Print dialog box

Do one of the following:

- Launch the PRINT or PLOT command.
- Press **Ctrl + P**.
- Choose **Print...** in the **File** menu.
- Click the **Print** tool in the **Output / Plot** ribbon panel (Drafting workspace).
- Select the **Model** tab or a **Layout** tab, then right-click and choose **Print** in the context menu.

17.1.2 Choosing the paper size

Press the down arrow, then select the paper size in the **Paper Size** list.

The available sizes depend on the selected printer.

17.1.3 Choosing the paper unit

Select either **Inches** or **Millimeters**.

The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

Nota: The **Scale** and **Plot offset** setting units are changed according to the chosen **Paper unit**.

17.1.4 Choosing the paper orientation

Select either **Portrait** or **Landscape**.

- **Portrait**: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- **Landscape**: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly.

When in a layout, click the **Apply** button to update the display.

17.1.5 Defining the plot offset

Select either **Center on page** or type the plot offset in the **X:** and **Y:** fields.

The offset is calculated from the bottom left corner of the paper.

When in a layout, click the **Apply** button to update the display.

To specify the plot area:

Option	Result
View	Prints the portion of the drawing that is displayed in the current viewport or a saved view.
Extents	Prints all entities in the drawing.
Limits	Prints to the limits defined in the drawing. This option is only available when printing in Model space.
Layout	Prints the current layout. This option is only available when printing in Paper space.
Window	Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing. Click the **Select print area** button (), then specify the two corners of the print area in the drawing. or Type the X- and Y-coordinates of the corners of the print area in the **X:** and **Y:** fields.

17.1.6 Defining the plot scale

Nota: The SCALELISTEDIT command allows you to edit the list of available scales in the **Print** dialog and the **Standard Scale** property of a paper space viewport.

... when printing a paper space layout:

If you print in paper space, the final scale of the plot output is the product of the viewport scale and the plot scale.

In order to keep control over the scale of the plot output:

- Specify the width and height of the viewports in drawing unit.
- Set the scale of each viewport to the scale you want in the plot output.
- The **plot scale** reflects the relationship between paper unit and drawing unit. (see table below)

Paper unit	Drawing unit	Plot scale	Printed mm or inches =	Drawing unit
mm	mm	1:1	1	1
mm	cm	10:1	10	1

Paper unit	Drawing unit	Plot scale	Printed mm or inches =	Drawing unit
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

... when printing model space:

If you print in model space, the scale of the plot output equals the plot scale, on the condition that your drawing unit matches the paper unit. Otherwise you need to multiply the plot scale with respect to the relationship between the paper unit and the drawing unit. E.g. if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the **plot scale** = 1/5, if 1 DU = 1 m the **plot scale** = 20/1 or 1/0.05

17.1.7 Setting the plot options

Click to toggle the **plot options** On / Off.

Option	Result
Plot object lineweights	If **Plot with plot styles** is off, object lineweights are plotted.
Plot with plot styles	Settings of the assigned STB or CTB file are used. ***Nota:*** This applies to viewports which have the 2D Wireframe or Wireframe visual style only.
Plot transparencies	Plots entities using the value of the **Transparency** property.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities are plotted last.
Hide paperspace entities	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.
Plot stamp	Adds a header and / or footer. Click the **Edit plot stamp button** ( ) to define the plot stamp.
Save changes to layout	All changes made in the **Print** dialog are saved in the layout. This option toggles the **Save changes to layout** variable.

17.1.8 Setting the shade plot mode

By default the Shade Plot mode is set to **As displayed**.

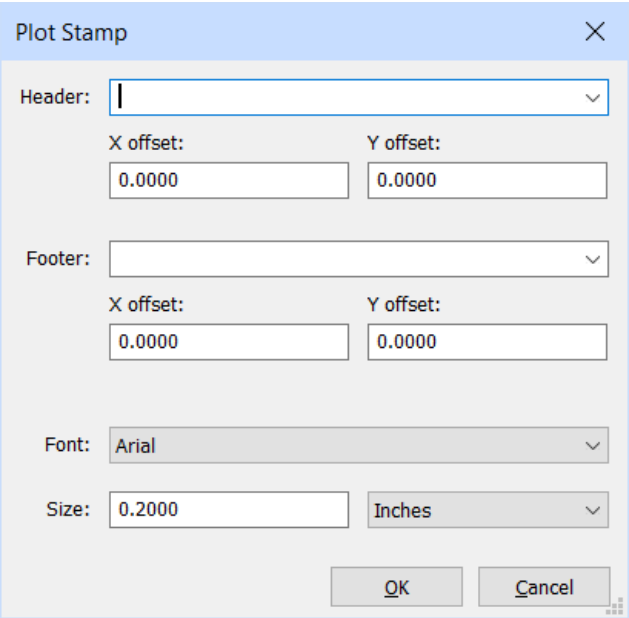
Click the list button, then select an option to plot using a different visual style.

Nota: This option applies to model space printing only. The Shade Plot mode of a paper space viewport is defined in the Shade Plot property of the viewport.

17.1.9 Defining the plot stamp

1 Click the **Edit plot stamp** button.

The **Plot stamp** dialog box appears.



2 In the **Header** field, do one of the following:
- Type the header text.
- Click the down arrow button, then choose a variable in the drop down list.
- Type one or more variable names (see table below).
- Leave the field empty.

3 Define the X and Y offset for the header.

4 In the **Footer** field, do one of the following:
- Type the footer text.
- Click the down arrow button, then choose a variable in the drop down list.
- Type one or more variable names (see table below).
- Leave the field empty.

5 Define the X and Y offset for the footer.

6 Select a font in the **Font** list button.

7 Enter a value in the **Font Size** field.

8 Set the unit type to **Millimeters** or **Inches**.

9 Click the **OK** button to save the plot stamp.

Variable Name	Description
$Name	Adds the name of the current user, as defined in the **User Name** field of the **License Information**.
$Company	Adds company name of the current user, as defined in the **Company** field of the **License Information**.
$Date	Adds the current date.
$DateTime	Adds the current date and time.
$DwgName	Adds the file name and path of the drawing.

Nota:

- The header and footer text consists of three parts (left, center and right), separated by commas.
- By default, the header and footer text is centered.
- The header and footer text can be outlined left center and/or right by separating the text with commas.
- e.g. **$Name,$Company,$Date** places your name left, company name centered and the current date at the right.
- Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.
- e.g. **$Company, ,** ($Company,<space>,<space>) outlines the company name left. , ,$Company (<space>,<space>,$Company) outlines the company name right.
- The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the **Settings** dialog box under **Program Options** > **Plot and Publish** > **Plot Stamp**.

17.1.10 Specifying the number of copies

Do one of the following:

- Type the desired number in the **Number of copies** field.
- Click the up-arrow and the down-arrow buttons to specify the number of copies.

17.1.11 Previewing the plot output

Click the **Preview** button on the **Print** or **Page Setup** dialog box.

Choose **Print Preview...** in the **File** menu.

Type **preview** in the Command line, then press Enter.

17.2 Plotter configuration

Configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD™ apply to Windows system printers only.

In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD for many plotting/printing devices and keep multiple configurations for each single device.

Each plotter configuration file contains information such as:

- The device driver and model
- The output port to which the device is connected
- Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the **Plotconfig** subfolder of your Roamable root folder.

After creating a PC3 file, it is available in the list of plotter configuration names in the **Printer / Plotter configuration** list on the **Print** or **Page Setup** dialog boxes.

17.2.1 Configuring the built-in PDF printer

1 Open the **PRINT** dialog box.

2 Click the **Printer/Plotter configuration** list button and choose **Print As PDF.pc3** in the list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter Configuration Editor** dialog is displayed with the **Print As PDF.pc3** configuration file loaded.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 (Option) Edit the **Graphics** settings.

6 (Option) Execute the **Scale Calibration**.

7 Click the **Custom Properties** button.

The **Print As PDF - Custom Properties** dialog is displayed.

8 (Option) Adjust the **Output quality** and **Layers support** settings.

9 Click the **Manage Custom sizes ...** button.

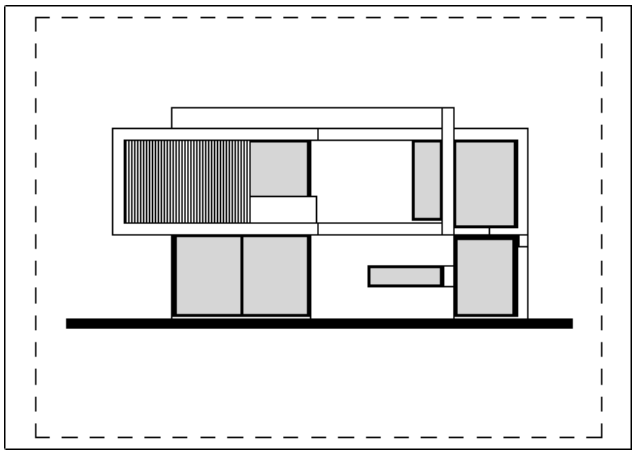
10 To create a custom paper size, do the following:

- Click the **Add** button.

  A new paper size is added.
- Double-click the newly created paper size.
- Type a name for the new paper size, then click the **OK** button.
- Define **Width** and **Height** for the new paper size.
- Define the **Paper Margins**.

Paper margins define the printable area of paper sheet.

If the DISPPAPERMARGINS system variable is ON, dashed lines indicate the paper margins on the paper sheet in a layout.



The paper sheet background is displayed if the DISPPAPERBKG system variable is ON.

- Click the **OK** button to create the custom paper size.

11 Click the **OK** button on each of the previously opened dialogs.

The custom paper size(s) can now be selected in the **Paper Size** list button.

17.2.2 Creating a plotter configuration

1 Launch the PLOTTERMANAGER command.

1 A **File** dialog box is displayed with the folder, defined by the PLOTCFGPATH system variable.

2 Double-click **Create a Plotter Configuration** shortcut.

The **Plotter Configuration Editor** dialog box is displayed.

3 In the **Printer name** list, select the printer you want to create a plotter configuration (PC3) for.

4 Click the **Settings** tab on the **Plotter Configuration Editor** dialog.

5 Click the **Custom Properties...** button.

The **<Selected Printer> Properties** dialog opens.

The layout of this dialog is different for each printer.

Custom Properties are not supported on the Linux platform.

6 On the **Properties** dialog of the selected printer, adjust the printer settings, then click the **OK** button to confirm.

The **<Selected Printer> Properties** dialog closes.

7 Click the **OK** button on the **Plotter Configuration Editor** dialog.

The **Save Plotter Configuration As** file dialog appears.

8 (option) Edit the PC3 file name.

9 The default name is **<Selected Printer>.pc3**.

10 Click the **OK** button on the **Changes to a Printer Configuration File** dialog.

The PC3 file is created.

Nota: You can create multiple plotter configurations for the same printer if you save them under different names.

17.2.3 Editing a plotter configuration

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Select a plotter configuration (PC3) in the **Printer / Plotter configuration** list.

3 Click the **Edit Plotter Configuration** button.

The **Plotter configuration editor** dialog opens.

4 Proceed in the **Creating a plotter configuration** procedure above starting from step 4.

Nota: If you select a system printer in step 2, a new plotter configuration is created for the selected printer. When you click the **Edit Plotter Configuration** button in step 3, the **Properties** dialog of the selected printer opens first. After clicking the **OK** button on the **Properties** dialog, the **Plotter configuration editor** dialog opens. You can then complete the **Creating a plotter configuration** procedure.

17.2.4 Assigning a plotter configuration

1 Choose **Model** or the **Layout** to which you want to assign a specific plotter configuration file.

2 Do one of the following:
- Choose **Page Setup** in the **File** menu.
- Type **pagesetup** in the Command line, then press Enter.

3 Choose one of the following from the **Printer / Plotter configuration** list:
- A system printer
- A configuration file (PC3 file)
- None (uses Default)

4 Click the **OK** button.

Nota:

- When BricsCAD cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to **None**.
- The **None** printer device has its own set of paper sizes that can be stored in the drawing, very much the same as for any 'real' printer. If a layout is opened and the stored printer is absent, the printer switches to **None** while the paper size is maintained. The missing printer and the previous paper size are indicated upon opening the **Print**dialog. The previous paper size can be saved in the drawing.
- PREVIOUS PAPER SIZE: drawings created by other parties are often set up for a printer that is not available in your office. In such case, BricsCAD resets the printer device to **None**, which results in using the default printer. Instead of resetting the paper size to a default, BricsCAD shows the Previous Paper size, in order to inform you about the intended paper size and allow you to choose a similar size on any of the available printers.

17.3 Plot style tables

The **Plot Style Table Editor** displays the plot styles contained in the specified plot style table.

17.3.1 Opening the Plot Style Table editor

Do one of the following:

- Click the **Edit Plot Style** button on the **Print** or **Pagesetup** dialog box.
- Choose **Plot Style Manager...** in the **File** menu, then double-click a **CTB** or **STB** file.
- In a **Windows Explorer** window, double-click a **CTB** or **STB** file.

The Plot Style Table Editor includes the following tabs:

- The **General** tab
- The **Form View** tab
- The **Lineweights** tab
- The **General** tab

The **General** tab lists general information about the plot style table.

Name	Description
Plot style table file name	Displays the name of the plot style table file you are editing.
Description	Displays a description you want to include about the plot style table.
File info	Displays information about path, number of plot styles and the version number of the Plot Style Table Editor. ***Nota:*** A color-dependent plot style table always contains 255 plot styles.
Apply global scale factor to non-ISO linetypes	Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.
Scale factor	Specifies the factor by which to scale non-ISO linetypes and fill patterns.

The **Form View** tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

The first plot style in a **Named Plot Style** table is **NORMAL** and represents an entity's default properties (no plot style applied). You cannot modify or delete the **NORMAL** style.

Property	Description
Plot styles	Displays the names of plot styles contained in the selected plot style table. • Always contains 255 styles (1 for each color). • They are tied to entity color. • You cannot add nor delete a plot style. • You cannot rename a plot style. Named plot style table (STB): • Contains one or more plot styles. • You can add or remove plot styles, except for the **Normal** style, which cannot be deleted. • Plot styles can be renamed, except for the **Normal** style, which cannot be renamed.
Description	Description for the selected plot style.
Color	Specifies the plotted color for an entity. The default setting for plot style color is **Use Object Color**. If you assign a plot style color, the color overrides the entity's color in the plot output.
Dither	Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are mapped to the nearest color, resulting in a smaller range of colors when plotting. Dithering is available only when you select the object's color or assign a plot style color. By default, this option is active. Setting it to Off has no effect in BricsCAD.
Grayscale	Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output.
Pen #	Specifies a pen to use when plotting entities that use this plot style. You can select a pen number from 1 to 32.

Property	Description
Virtual pen #	Specifies a virtual pen number between 1 and 255. Choose **Automatic** to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index. The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select **255 Virtual Pens** under **Color Depth** under **Vector Graphics** on the **Settings** tab in the **Plotter Configuration Editor**).
Screening	Specifies the color intensity of the plot on the paper. The valid range is 0 through 100. If you select 100 the drawing will plotted with its full color intensity. In order for screening to work, the **Dithering** option must be active.
Linetype	Displays a list with an example and a description of each linetype. The default setting for plot style linetype is **Use Object Linetype**. The selected linetype overrides the entity's linetype in the plot output.
Adaptive	Adjusts the scale of the linetype to complete the linetype pattern. You can activate this property if a complete linetype pattern is more important than a correct linetype scaling.
Lineweight	Displays a sample of the lineweight as well as its numeric value. You can modify an existing lineweight if the one you need is not available. The default setting for plot style lineweight is **Use Object Lineweight**. The selected lineweight overrides the entity's lineweight in the plot output.

Property	Description
Line end style	Displays a list of several end styles for the lines. By default, the option **Use Object End Style** is selected. If you select a line end style, the object's line end style will be overridden in the plot output.
Line join style	Displays a list of several join styles for the lines. By default, the option **Use Object Join Style** is selected. If you select a line join style, the object's line join style will be overridden in the plot output.
Fill style	Displays a list of several fill styles. By default, the option Use **Object Fill Style** is selected. If you select a fill style, the object's fill style will be overridden in the plot output.

Nota: Entities which are created in **true color** are always plotted using their own object properties (color, lineweight, ...).

The **Lineweights** tab lists the available lineweights.

Lineweights with a tick are used in the **Lineweight** field on the **Form View** tab.

To edit a lineweight, select the lineweight in the list, then click the **Edit** button.

17.3.2 Assigning a plot style table

In BricsCAD, you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose **None (use Default)** from the list of plot style tables.

In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or **By Layer**.

Plot style tables are used only if the **Plot with plot styles** plot option on the **Print** dialog box is ticked. When you turn off plot style tables, entities print according to their own properties.

Assigning a plot style table

1 Open the **Print** or **Page Setup** dialog box.

2 Select a **Plot style table** from the list box in the **Plot Style Table (pen assignments)** section.

3 Click the **OK** button.

17.4 The plot styles manager

The **Plot Styles Manager** is a window that lists plot style table files (CTB and STB files).

The Plot Styles Manager allows you to:

- Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the **Create a Plot Style Table** wizard.
- Edit existing Plot Style Table files.

17.4.1 Launching the plot styles manager

Do one of the following:

- Choose **Plotstyle manager...** in the **File** menu.
- Launch the STYLESMANAGER command.

A **File** dialog box is displayed, showing the content of the PlotStyles folder, which is defined by the PLOTSTYLEFOLDER system variable.

17.4.2 Creating a new plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the **Create a Plot Style Table** shortcut. The **Add Plot Style Table Editor** dialog box opens.

3 Choose **Create new from scratch**.

4 Do one of the following:

- Select either **CTB** or **STB** from the **Table type** selection button.
  - **CTB** (Color-dependent Plot Style Table) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file.
  - **STB** (Named Plot Style Table) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.
- Choose **Create from existing table**, then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.

5 Type a name in the **Table name** field, then click the **OK** button.

The **Plot Style Table Editor** dialog box is displayed.

17.4.3 Creating a new plot style table in the Print dialog box

1 Launch the PRINT command.

1 The **Print** dialog box appears.

2 Under **Plot style table**, click the **Create new plot style**.

The **Add Plot Style Table** dialog box appears.

3 Type a name in the **Table name** field, then click the **OK** button.

If the **Launch Editor** option is chosen, the **Plot Style Table Editor** dialog box appears.

If the **Use this plotstyle for the current drawing** option is chosen, the new plot style table is used in the current layout.

Nota: The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

17.4.4 Editing a plot style table

1 Launch the Plot Styles Manager.

2 In the PlotStyles folder, double-click the plot style table you want to edit.

3 The **Plot Style Table Editor** dialog appears.

17.4.5 Editing a plot style table in the Print dialog box

1 Open the **Print** dialog box.

2 Select a plot style table from the **Plot Style Table** list button.

3 Click the **Edit plot style** button.

17.5 Batch plot

The PRINT command prints a single layout at a time. The PUBLISH command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a DSD (Drawing Set Description) file.

The SHEETSET command manages sheet sets and allows you to print, publish or etransmit a sheet set as a whole or a selection of sheets. A sheet set is saved to a DST file (Sheet Set Data) file.

18. Rendering

18.1 Using TwinMotion

Nota: This feature is only available for BricsCAD on Windows.

Twinmotion offers real-time, immersive 3D architectural visualization using the power of Unreal Engine from Epic Games. All your BricsCAD model data are exported using the Unreal Engine file type (*.udatasmith) in the EXPORT command.

To use Twinmotion with BricsCAD V21 you need to:

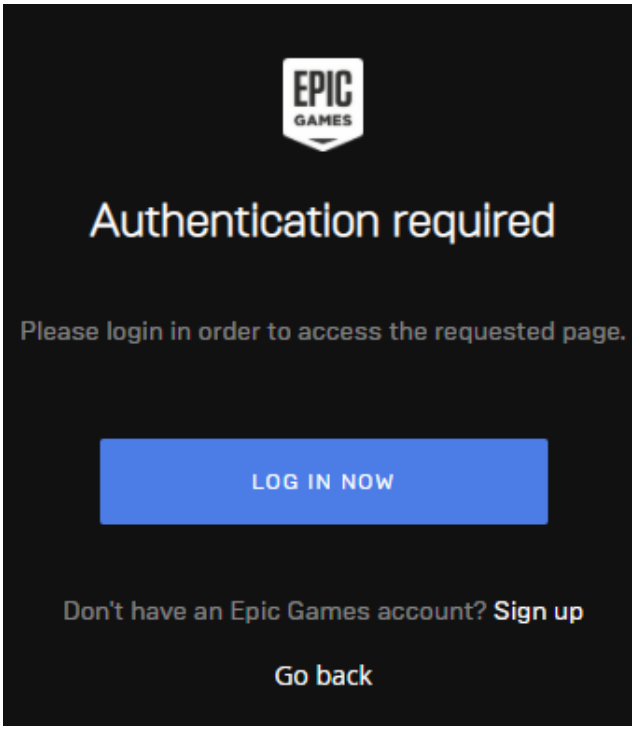
- Install the **Epics Games Launcher**.
- Download and install **Twinmotion**.
- Export your model to a ***.udatasmith** file.

18.1.1 Installing Twinmotion

From the Twinmotion Help pages:

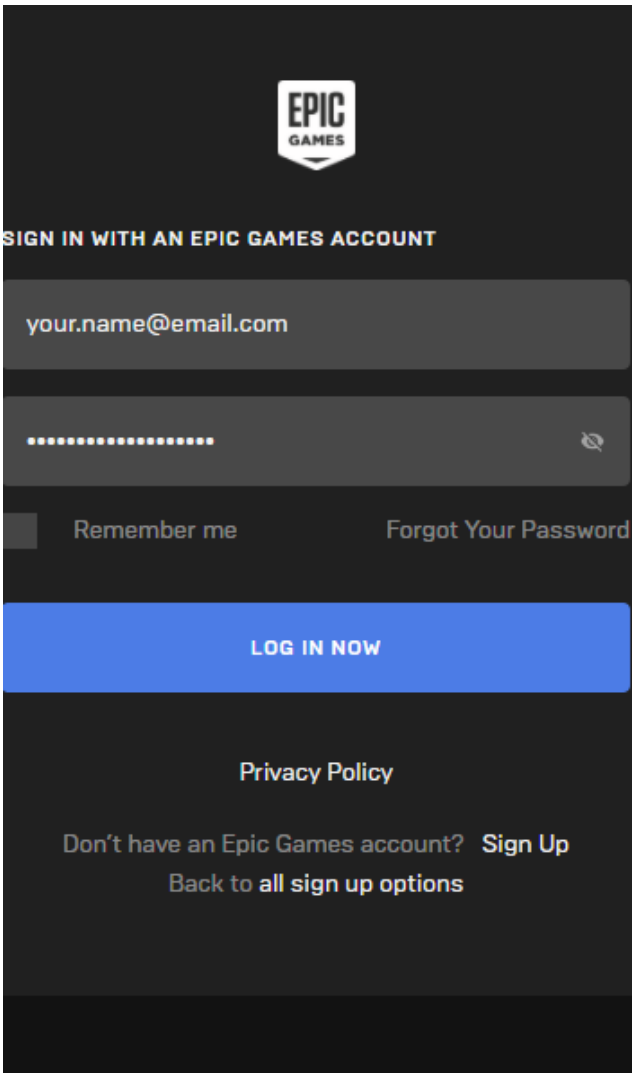
1 Go to https://www.unrealengine.com/en-US/twinmotion.

2 Click on **Get Started Now**.



If you do not have an Epic Games account yet, click **Sign up**, then fill in all required information and click **Create Account**.

3 Click **LOG IN NOW**.



4 Once you are logged in, click on **Download the Epic Games Launcher**.

5 Once the Epic Games Launcher is downloaded, make sure to click on the **Unreal Engine** tab on the left side.

You will now see the **Twinmotion** tab in the top right.

6 Click on **Install** to install Twinmotion.

7 Carefully choose your file location, it is strongly recommended to use the default location.

18.1.2 Exporting your model

1 Open your model in BricsCAD BIM.

2 Launch the DATASMITHEXPORT command.

3 The **Export drawing to** dialog box appears.

The **Save as type** is Unreal Engine (*.udatasmith).

4 Type a name or accept the default name, then click the **Save** button.

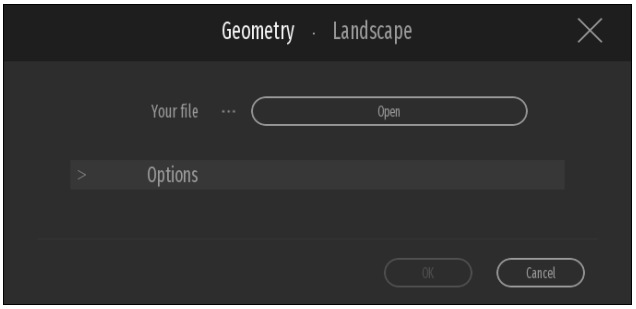
The exported model can now be imported in Twinmotion.

18.1.3 Importing your model in Twinmotion

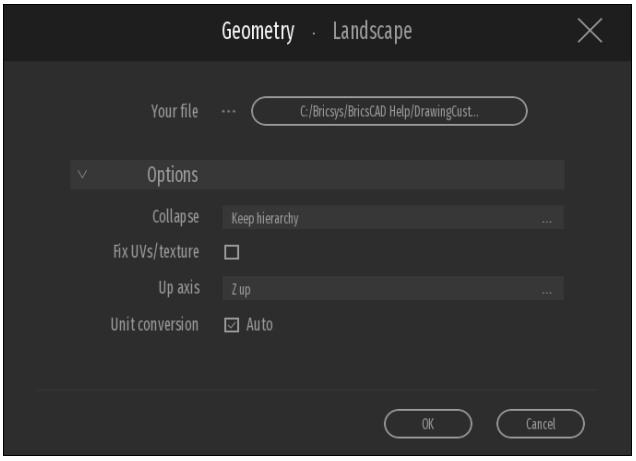
1 Launch Twinmotion.

2 Do one of the following:
   - Click the **Import Files** button.
   - Press **Ctrl + I**.

3 Make sure **Geometry** is selected, then click the **Open** button.



On the **Import** dialog, select a **Unreal Engine** (*.udatasmith) model file.



4 Click **Options**, and set the **Collapse** option to **Keep Hierarchy**.

   This structures the model the same way as the layers used in BricsCAD.

5 Click the **OK** button to import the model.

18.2 Rendering overview

Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png.

Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the

RenderUsingHardware system variable. It is recommended to install the most recent drivers for your graphics card.

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset via dialog box.

18.3 Rendering preferences

The RENDERPRESETS command allows you to create and edit render presets, and to set the current render preset through a dialog box.

18.3.1 Setting the render preferences

1 Open the **Settings** dialog box.

2 Under **Program Options**, expand the **Rendering** settings category.

Current material:

Specifies the material for new entities. By default, the **Global** material is used.

Lighting units:

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles
- 2: Enables International lighting units: Lux

Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 or 2.

Default lighting:

A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

Linear contrast:

Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

Linear brightness:

Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

Render using hardware:

Choose **User software only (slower)** if hardware rendering fails. It may be necessary to restart BricsCAD after changing the setting.

Real world scale:

Controls the rendering of materials with units set to real-world scale.

Sky status:

Specifies whether sky illumination is computed at render time. (Not implemented yet)

Texture map path:

Defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under **Textures**, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the **Diffuse map** setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the TextureMapPath user preference to folder 1, 2 or 3.

Tile mode light synch:

Controls the synchronization of lighting in all model space viewports. (Internal use only)

18.4 Rendering tools

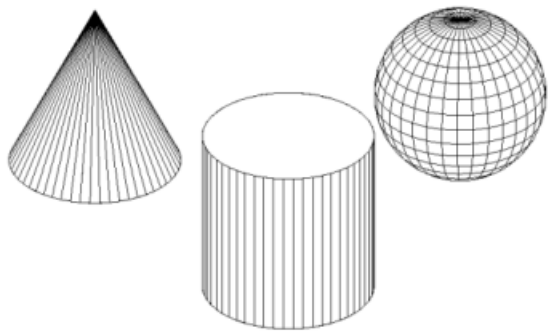
Rendering tools are available in:

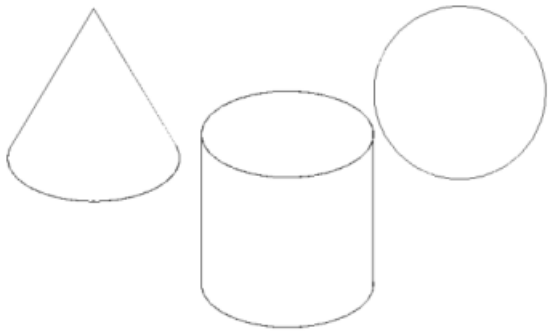
- The **Rendering** toolbar.
- In the **View** > **Rendering** menu.
- The **View** and **Visualize** ribbon tabs.
- In the **General** tab of the **No Selection** Quad.

Icon	Tool Name	Command	Description
	Render	RENDER	Photorealistic rendering using materials and lights
	Hide	HIDE	Hidden line view (*). If the previous shademode option was **2Dwireframe**, wireframe representation is restored if you zoom or pan. If the previous shademode option was one other than **2Dwireframe**, the hidden line representation is kept until a different shade mode is chosen.
	Shade	SHADE	Shades the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe	SHADEMODE + 2	Wireframe representation
	3D Wireframe	SHADEMODE + 3	Wireframe representation
	Hidden	SHADEMODE + H	Hidden line view (*)
	Flat	SHADEMODE + F	Shades the entities between the polygon faces. Curved surfaces are segmented

Icon	Tool Name	Command	Description
	Gouraud	SHADEMODE + G	Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges	SHADEMODE + L	Combines flat shading and wireframe lines and curves
	Gouraud + Edges	SHADEMODE + O	Combines gouraud shading and wireframe lines and curves
	Materials	MATERIALS	Opens the **Drawing Explorer - Materials**
	Material Map	MATERIALMAP	Allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.
	Light List	LIGHTLIST	Opens the Drawing Explorer - Lights
	Sun Properties	SUNPROPERTIES	Opens the Drawing Explorer - Lights
	Geographic Location	GEOGRAPHICLOCATION	Opens the **Geographic Location** dialog box.
	Animation Path	ANIPATH	Records the animation of a camera moving along a path or panning in a 3D model and saves it to a movie file.
	Animation Editor	ANIMATIONEDITOROPEN	Opens the **Animation Editor** panel.

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.



Hidden line of a cone, a cylinder and a sphere with DISPSILH Off (top) and On (bottom).


18.5 Render materials

The MATERIALS command creates materials and edits their properties through the Drawing Explorer.

The MATBROWSEROPEN command opens the **Render materials** panel, which shows an overview of available high-resolution render materials.

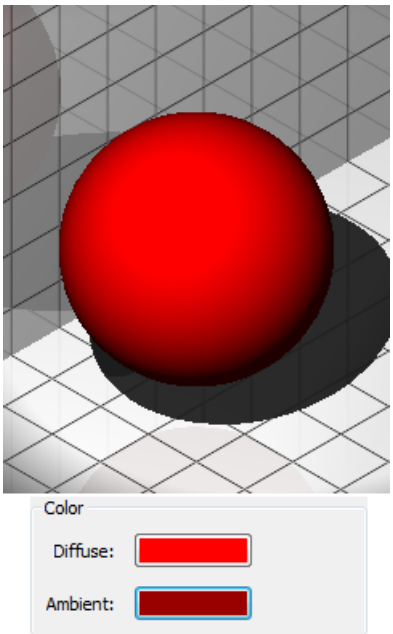
The MATBROWSERCLOSE command closes the **Render materials** panel.

The MATERIALMAP command allows you to adjust how a texture image is mapped to faces of a 3D solid and filled entities such as 3D faces, wide polylines and polygon meshes.

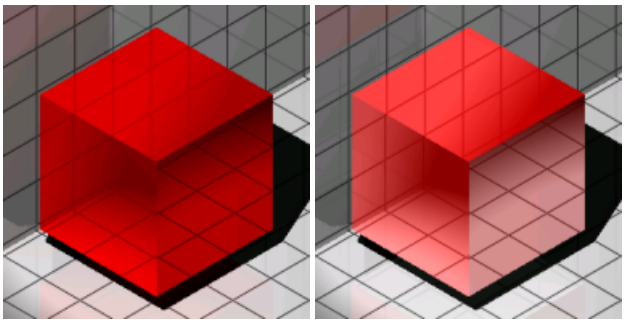
Material definitions are saved in the drawing. Each drawing contains the **Global** material, which you cannot to delete or rename. You can edit the properties of the **Global** material though.

18.5.1 Materials glossary

- **Diffuse color**: the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.
- **Ambient color**: the color of the object in the shadow, under ambient light rather than direct light.

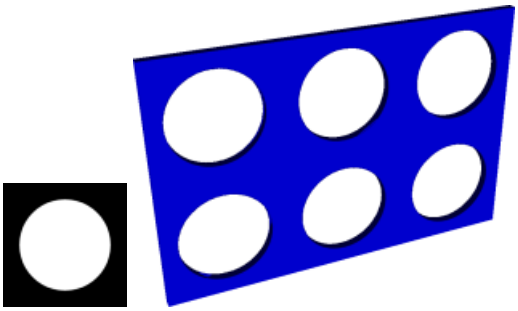


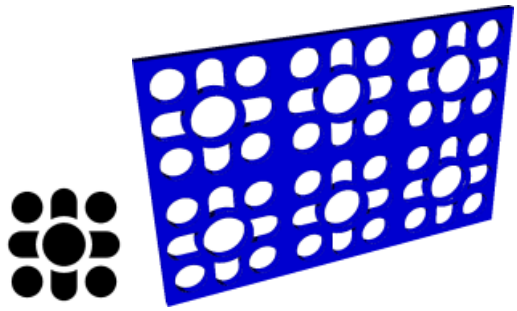
- **Highlights**: highly reflective highlights in the color of the material.



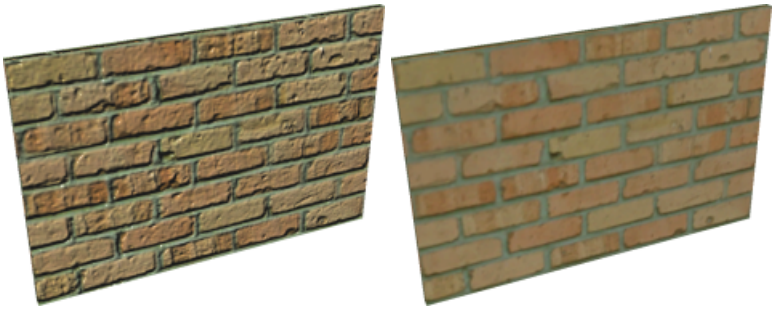
Metallic Highlights (left) and Non-metallic Highlights (right)

- **Shading**: how the color and brightness of a surface varies with lighting.
- **Texture-Mapping**: adds detail to a surface that is not included in the 3D model itself.
- **Diffuse map**: adds a structure or pattern to a surface to simulate a material.
- **Transparency map**: sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grayscales). (Black=Transparency map)





- **Bump map**: simulates a certain roughness or bumpiness on surfaces. (left = with bump map)



- **Reflectivity**: the property of a surface to reflect light.
- **Transparency**: sharp transmission of light through solid objects.
- **Translucency**: scattered transmission of light through solid objects.
- **Refraction**: bending of light when passing through transparent objects.
- **Self-illumination**: it appears as if a surface is emitting its own light. No light is cast on other objects.

18.5.2 Defining render materials

See the MATERIALS command.

18.5.3 Assigning render materials

Materials can be assigned to a layer and to entities or subentities. If the **Material** property of an entity is **BYLAYER**, the material assigned to the layer of the entity will be used.

By default, new entities get the **BYLAYER** material setting.

18.5.4 Adding a render material to the drawing

1 Select the material in the **Render Materials** panel.

2 Right-click and choose the **Add to drawing** option.

18.5.5 Assigning a render material to entities

When the current visual style is BIM, Modeling, Realistic or X-Ray, entities and subentities are rendered using their material.

Method 1: Using the Properties panel

1 Select the entities or sub-entities (e.g. faces or 3D solids).

2 Under **3D Visualization** > **Material**, click the down arrow and select the material in the **Material** list.

Nota: Materials in the drawing are available only.

Method 2: Using the Render Materials panel

1 Select the material, which launches the MATERIALASSIGN command.

1 The cursor turns into a paintbrush.

You are prompted: Select object to assign material to:

2 Click the entities or subentities to assign the material.

3 (option) Select a different material, then click more entities or subentities.

4 (option) Hold down the Alt key.

5 The cursor turns into a dropper.

You are prompted: Select material from object:

Click an entity or subentity to copy the material from, then click entities or subentities to assign the material to.

Nota: The Hotkey Assistant appears at the bottom of the screen, indicating the current selection mode: 3D solids or 3D solid faces. Hold down the Ctrl key to invert the selection mode.

To assign a material to a selection set:

1 Select the entities.

2 Search the material in the **Render Materials** panel.

3 Click the material.

4 The material is assigned to all entities or subentities in the selection.

18.5.6 Removing a render material

Assigning the Global material, removes a rendering material from an entity or subentity.

1 Select the entities and/or subentities.

2 Select the **3D Visualization / Material** field in the **Properties** panel and select the **Global** material.

Nota: Clicking the **Erase Material** tile in the **Render Materials** panel sets the Material property to BYLAYER.

18.5.7 Assigning a material to a layer

1 Select the **Material** field of the layer in the **Layers** panel or the **Drawing Explorer – Layers** dialog box.

2 Click the down arrow and select the material in the list.

3 Materials in the drawing are available only.

18.5.8 Adding the Redway material library

Materials in the folder linked under **RENDERMATERIALSPATH** appear in the **Render** materials panel. To add the **Redway Material Library**, follow this procedure:

1 Download the complete Redway Material Library here.

2 Unzip the downloaded file and copy the path of the folder. The path should end with ...

\all_redway_materials.

3 In the **Render Materials** panel, click the hamburger menu ( ).

4 Select **Manage libraries** in the menu.

The **Settings** dialog box is displayed with the RENDERMATERIALPATH system variable selected.

5 Click the Browse button ( ).

The **Path List** dialog box appears.

6 Click the **Add Path** button.

An empty path field is added.

7 Do one of the following:

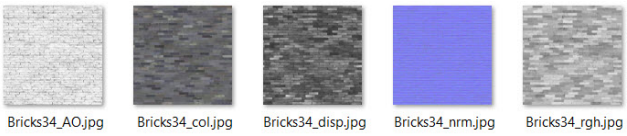
- Paste the path you have copied in the empty path field.
- Click the Browse button ( ), then select the folder in the **Choose a folder** dialog box.

The downloaded materials appear in the **Render Materials** panel.

18.5.9 Adding your own materials

1 Download the textures you want to use.

Nota: you can download a photo of your texture, but in order to get the most realistic view it is best to download a zip file that gives you different representations of the texture:



2 Launch the MATERIALS command.

3 The **Drawing Explorer – Materials** dialog box appears.

4 Click the **New** button.

5 Give your new material a name.

6 Check on the **Diffuse Map** and the **Bump Map**.

7 Edit the path of the diffuse map ( ) to the colored image of your texture.

8 Edit the path of the bump map ( ) to the normal image of your texture.

This will give your render material a more 3D and realistic look.

9 Edit the scale of your image until you get a realistic image in the preview.

10 The scale of the texture represents the number of drawing units covered by the full width of your image.

11 Right-click on your new material and choose **Add material(s) library** in the context menu.

12 Close the **Drawing Explorer** dialog box.

13 Open the **Render Materials** panel and click on the folder **User Defined**.

You find your new material in this folder of the library.

This folder is automatically generated in the following path: C:\ProgramData\Bricsys\RenderMaterials\UserMaterials\User defined.

18.6 Lights

Lighting is the key to make the viewer believe they are looking at a realistic scene.

18.6.1 Lighting glossary

- **Ambient light**: light from an unspecified source.
- **Back light**: light that adds depth and dimension by creating highlights.
- **Distant light**: light that is cast evenly throughout a scene with shadows all in the same direction.
- **Fall-off**: the attenuation or decrease in brightness the further from the light source.
- **Fill light**: the light that brightens dark areas and softens shadows from the main light.
- **Point light**: light that shines in all directions from a central spot.
- **Spot light**: focusable light that is aimed in one area.
- **Photometric web**: a 3D representation of the light intensity distribution of a light source.
- **Web light**: approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units).
- **Attenuation**: the fall-off in brightness the further from the light source.

18.6.2 System variables

LIGHTINGUNITS (Lighting units):

- 0: No lighting units are used. Enables generic lighting.
- 1: Enables American lighting units: Foot-Candles.
- 2: Enables International lighting units: Lux.

DEFAULTLIGHTING (Default lighting): A distant light that follows the view direction. This setting can be different per viewport.

- 0: Default lighting is used if no lights are switched ON.
- 1: Default lighting is used instead of the light sources defined in the drawing.

LINEARCONTRAST (Linear contrast): Specifies the ambient light intensity. Values between -10 and 10 are accepted. A value of -10 results in maximum ambient light. A value of 10 results in no ambient light.

LINEARBRIGHTNESS (Linear brightness): Specifies a scaling factor for the light intensity. Values between -10 and 10 are accepted. The default value is 0 (no scaling). Negative values decrease the light intensity, positive values increase the light intensity. This setting can be different per viewport.

LIGHTGLYPHDISPLAY (Light glyph display): Controls the display of light glyphs.

FBXEXPORTLIGHTS (Fbx Export Lights): Specifies whether to export lights from model space.

18.6.3 General properties

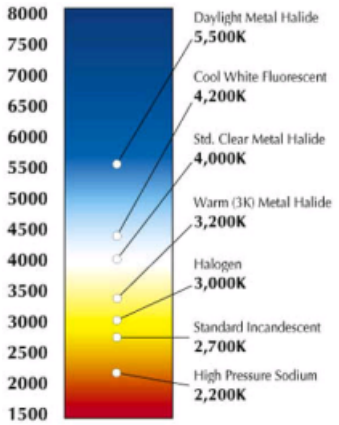
The general properties are common to all lights:

- **Name**: a user defined name for the light.
- **Type**: defines the type of light: spot, point, web or distant.
- **On/Off**: specifies whether the light is turned on or off.
- **Shadows**: specifies whether the light casts shadows or not. Turning shadows off to increases performance.
- **Intensity factor**: multiplies the **Lamp Intensity** property, the result is the **Resulting Intensity**.
- **Filter color**: defines the color of the light.
- **Plot glyph**: if on, the light glyph is plotted.
- **Glyph display**: controls the display of the light glyph.

18.6.4 Photometric light properties

LIGHTINGUNITS = 1 (American units) or 2 (International units).

- **Lamp intensity**: Defines the brightness of the light. The lamp intensity is expressed in candela (cd) which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function).
- **Resulting intensity**: the product of the lamp intensity and and intensity factor.
- **Lamp color**: defines the inherent color of the light or color temperature in Kelvin or standards.

	D65White Fluorescent Cool White White Fluorescent Daylight Fluorescent Incandescent Xenon Halogen Quartz Metal Halide Mercury Phosphor Mercury High Pressure Sodium Low Pressure Sodium
color temperatures	lamp color list

- **Resulting color**: this is the final color of the light: a combination of the lamp color and the filter color.
- **Photometric web**: If the light type is **Web**, **Photometric Web** and **Web offsets** are available (not implemented yet).

18.6.5 Geometry properties

- **Position**: controls the location of the light.
- **Target**: defines the target point for point lights, spotlights and weblights.
- **Targeted**: switches the **target** property on/off.

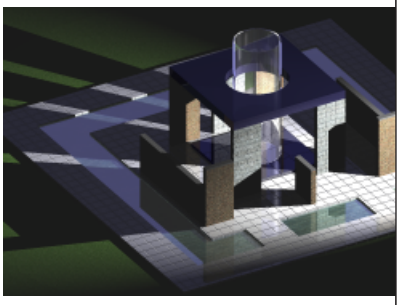
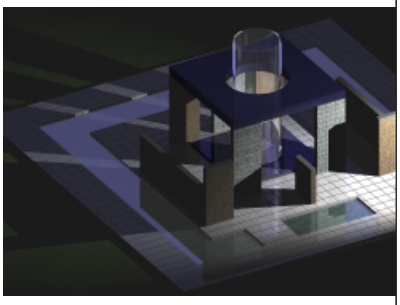
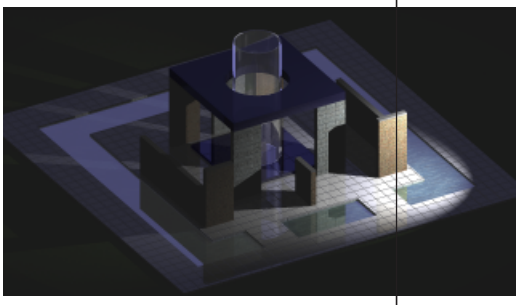
18.6.6 Attenuation properties

In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights.

An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene.

Attenuation properties apply to spot lights and point lights.

- **Type**: the options are **none**, **inverse linear** and **inverse square**
  - **None**: no attenuation. The distance to the light source has no influence.
  - **Inverse Linear** : the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.
  - **Inverse Square**: the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.
- **Use Limits**: if this property is set to Yes, the **Start limit offset** and **End limit offset** fields become active.
  - **Start limit offset**: defines the point where the light starts to shine, measured from the center of the light.
  - **End limit offset**: defines the point where the light stops to shine, measured from the center of the light.

		
No attentuation	Inverse linear attentuation	Inverse square attentuation

Nota:

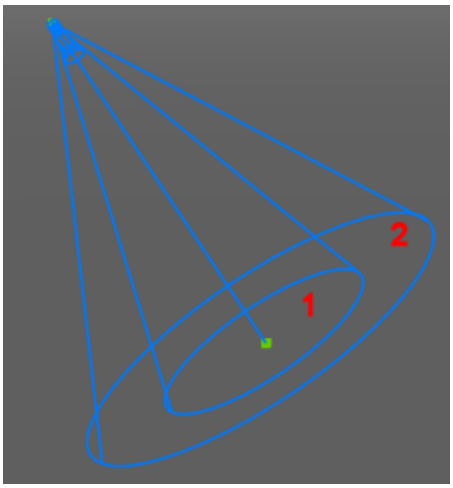
- The **Attenuation** properties apply to generic lights only (LIGHTINGUNITS = 0).
- Inverse square attenuation is applied automatically for photometric lights (LIGHTINGUNITS = 1 or 2).

18.6.7 Rendered shadow details

- Type: Sets the shadow type.
- Sharp (raytraced): generates sharp edged shadows.
- Soft (shadow map): generates realistic shadows with soft edges.
- Soft (sampled): Not supported yet
- Map size: specifies the amount of memory for shadow calculation. Click the field, then select a value in the drop-down list (64/128/256/512/1024/2048/4096).
- Softness: sets the softness of the shadow map. Enter a value between 1 and 10.

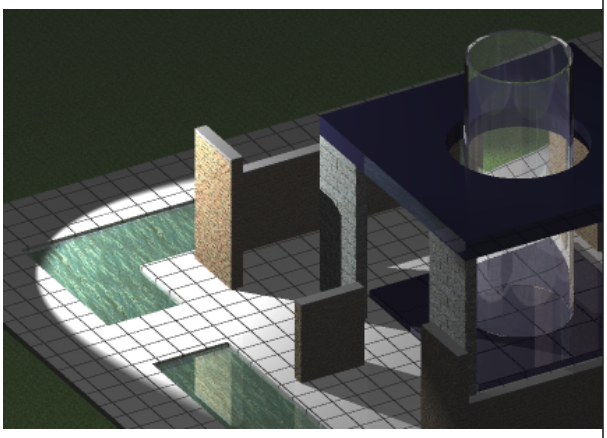
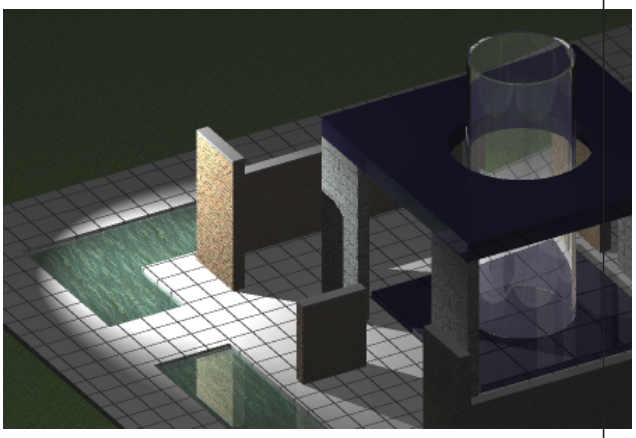
18.6.8 Spotlight hotspot and falloff properties

When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot = 1) that is surrounded by an area of lesser intensity (falloff = 2).



Hotspot angle: angle of the central light cone (defines the hotspot)

Falloff angle: angle of the full light cone

	
Hotspot angle = 55, Falloff angle = 60	Hotspot angle = 30, Falloff angle = 60

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity. If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left). The greater the difference between both angles, the softer the edge of the light cone (right).

18.6.9 Managing Lights

The LIGHTLIST command opens the **Drawing Explorer – Lights** dialog box.

Here you can:

- Create new lights.
- Delete lights.
- Select lights in the drawing.
- Switch lights On/Off.

18.6.10 Defining a light

1 Do one of the following:
   - Launch the LIGHT command.
   - Click the New ( ) button on the **Drawing Explorer – Lights** dialog box.

     You are prompted: Enter light type [Point/Spot/Web/Distant] <Web>:

     Specify the light type.
   - Click a toolbutton on the **Lights** toolbar or the **View** or **Visualize** ribbon tab.
   - : spot light
   - : point light
   - : distant light
   - : web light

   You are prompted: Specify source position <0,0,0>:

2 Specify a point in the drawing.

3 **You are prompted**: Specify target position <0,0,-10>:

4 Specify a point in the drawing.

5 **You are prompted**: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/Shadow/Attenuation/FilterColor/<eXit>:

6 Specify the properties of the light.

7 Choose **Exit** to conclude the command and create the light.

   A light glyph is displayed in the model, if LIGHTGLYPHDISPLAY = On.

Nota: It is not possible to define web lights if **LIGHTINGUNITS** =0 (No lighting units).

18.6.11 Editing lights

Lights can be edited in the **Properties** panel.

1 Do one of the following to select the light in the drawing:
   - Click the glyph of the light(s).
   - Select the light(s) in the **Drawing Explorer – Lights**, then right-click and choose **Select lights** in the drawing in the context menu.

   The properties of the selected light(s) appear in the Properties panel:

2 Edit the properties.

3 Press the Esc key to stop.

18.6.12 Defining the sun properties

1 Launch the SUNPROPERTIES command.

1 The **Drawing Explorer - Lights** dialog appears, showing the sun properties in the **Editor** pane.

2 Edit the properties.

Nota: When LIGHTINGUNITS = 1 or 2, the **Color** property of the sun cannot be edited.

18.6.13 Defining the geographic location

1 Launch the GEOGRAPHICLOCATION command.

1 The **Geographic Location** dialog box appears.

2 (option) Check the **Store geographic location information in drawing** option.

3 (option) Check the **Use Map Grid coordinate system** option.

4 Do one of the following:
   - Type the latitude and longitude of the location in the **Latitude** and **Longitude** fields and select the appropriate hemisphere for each setting.
   - Click the **Choose location...** button, then select a city and timezone and click the **OK** button.

5 (option) If you did not select a time zone in the previous step, choose a time zone in the **Time Zones** list.

6 (option) Specify the **Coordinates of the corresponding location in the drawing**.

7 Define the direction of the north.

8 Click the **OK** button.

18.7 Creating a rendered image

The rendered image appears in the current viewport in a separate window or can be exported to a file.

The background of a rendered view can be controlled by the Background property of the current named view.

The ANTIALIASRENDER system variable controls the amount of anti-aliasing (edge smoothing) applied during the rendering generation. The default value is 1, maximum value is 5.

Nota:

- High anti-alias values incur a high calculation cost.
- The values of the lighting system variables influence the rendering result.

18.7.1 Rendering a viewport

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Viewport** option.

3 Click the **OK** button.

The view is rendered.

18.7.2 Rendering in a window

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **Window** option.

3 Click the **OK** button.

The view is rendered in a popup window.

18.7.3 Saving a rendering to a file

1 Launch the RENDER command ( ).

The **Render** dialog box appears.

2 Click the **File** option.

3 Click the **Browse** button ( ), at the right hand side of the **File** field. The **Render output file** dialog box appears.

4 Select a folder and type a name for the output file in the **File name** field.

5 Click the **Save as type** button to select the file type.

The options are: **bmp**, **jpg** and **png**.

6 Click the **OK** button.

The **Render Output File** dialog closes.

7 Do one of the following:

- Type the desired resolution for the output size in the **Width** and **Height** fields.
- Check the **Same as viewport** size option to set the outsize to the size of the current viewport.

8 Click the **OK** button to create the file.

19. Feedback utente

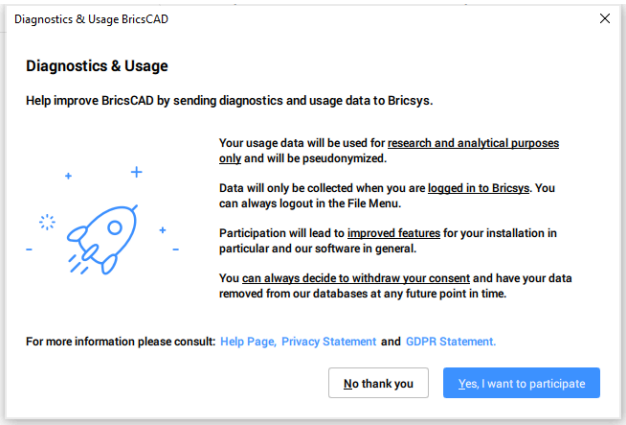
19.1 Diagnostics and usage data collection

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.

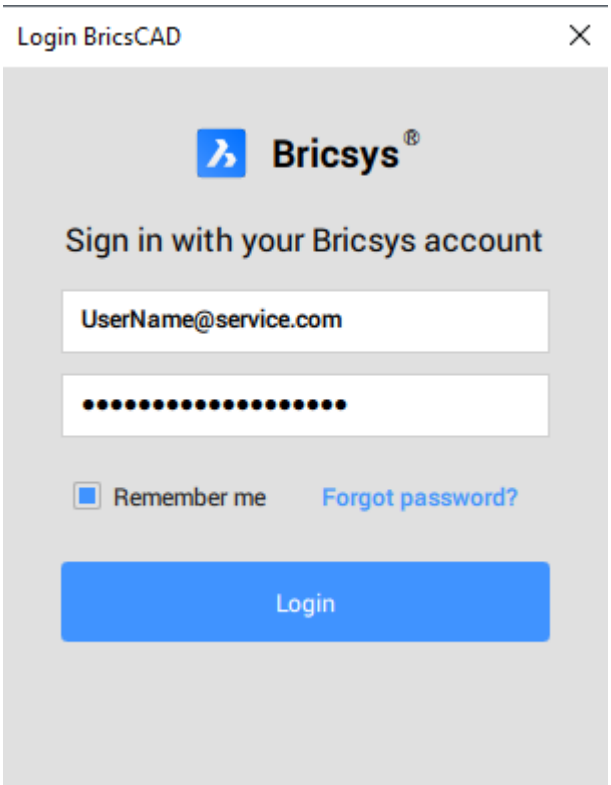
When transferring data:

- 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, set DATACOLLECTION to [0] Off.

2 Close and restart BricsCAD.

19.1.2 Restarting the process

1 In the Settings dialog box, under User Preferences, set DATACOLLECTION to [-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 Cosa c'è di nuovo

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 alla pagina 968

Building data alla pagina 975

Project collaboration alla pagina 984

Design documentation alla pagina 985

Point clouds alla pagina 994

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 alla pagina 31

User interface alla pagina 32

Productivity alla pagina 35

Drafting alla pagina 49

Modeling alla pagina 51

Civil tools alla pagina 60

Performance and compatibility alla pagina 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:

Panoramica alla pagina 1538

Gestore BOM alla pagina 1539

Librerie di parti alla pagina 1542

Parametri e Vincoli alla pagina 1543

Parti standard alla pagina 1543

Lamiera alla pagina 1543

20.1.1.1 Release notes

Click here for an overview of the release notes.

20.1.1.2 Diagnostics and Usage Data Collection

Many of the improvements were inspired by requests and suggestions from you, BricsCAD users. At Bricsys, we continue to expand our methods of gathering user feedback with new data collection tools. When you launch BricsCAD for the second time, a dialog box displays, inviting you to participate in the

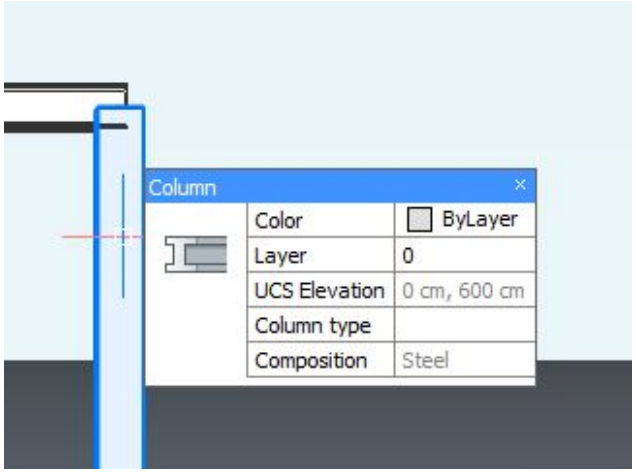
Diagnostics and Usage Data Collection program. The data collection process enables you to share anonymous diagnostic and usage data with Bricsys to help improve BricsCAD. You can withdraw your consent and have your data removed from our databases at any time.

20.1.2 Modeling techniques

20.1.2.1 BIM Profiles

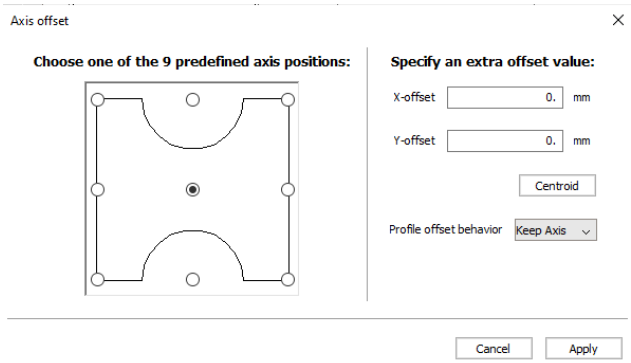
BIMAPPLYPROFILE / BIMLINEARSOLID

The composition material of a profile is now attached to the created solid in the BIMAPPLYPROFILE command and BIMLINEARSOLID commands.



BIMPROFILES

When changing the Offset property of a CustomShape profile via the Profiles dialog, you can choose whether to keep the position of the solid or its axis, for linear solids using said profile.



BIMPROFILESTANDARDS

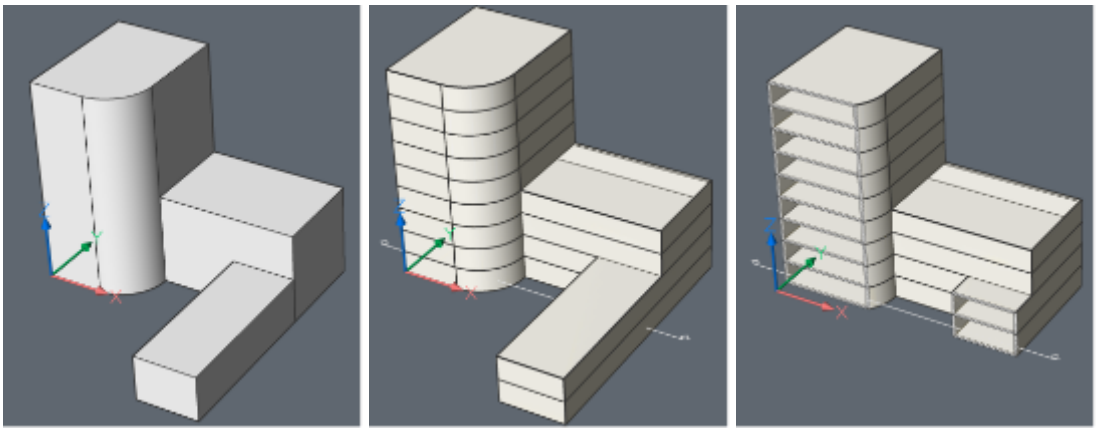
The BIMPROFILESTANDARDS setting is no longer case sensitive. In case there are no profiles matching the value of the BIMPROFILESTANDARDS setting, a warning message displays.

20.1.2.2 Mass Modeling tools

The first prototype of the new BIMQUICKBUILDING command has been implemented. The QuickBuilding tool allows you to generate a BIM building just from one massing solid. It automatically generates different levels, walls, spatial locations, etc. It can be very useful in early design stages, volume studies and conceptual design. The tool will assign BIM properties to each part of the new building.

Spatial Locations can now be used as input for BIMQUICKBUILDING. This workflow allows you to process different buildings simultaneously.

The command also works on solids without edges, for example, a sphere or a torus.

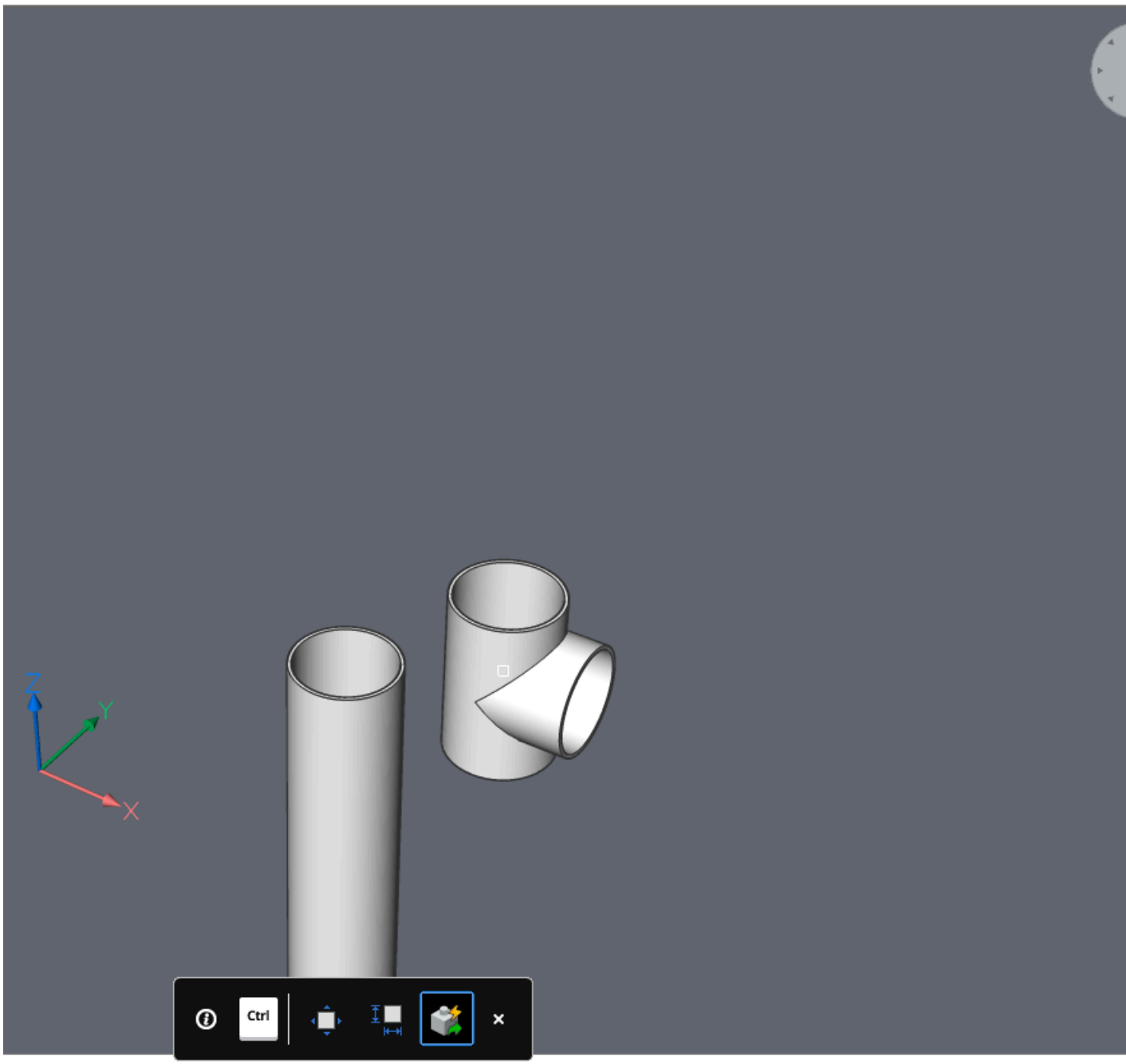


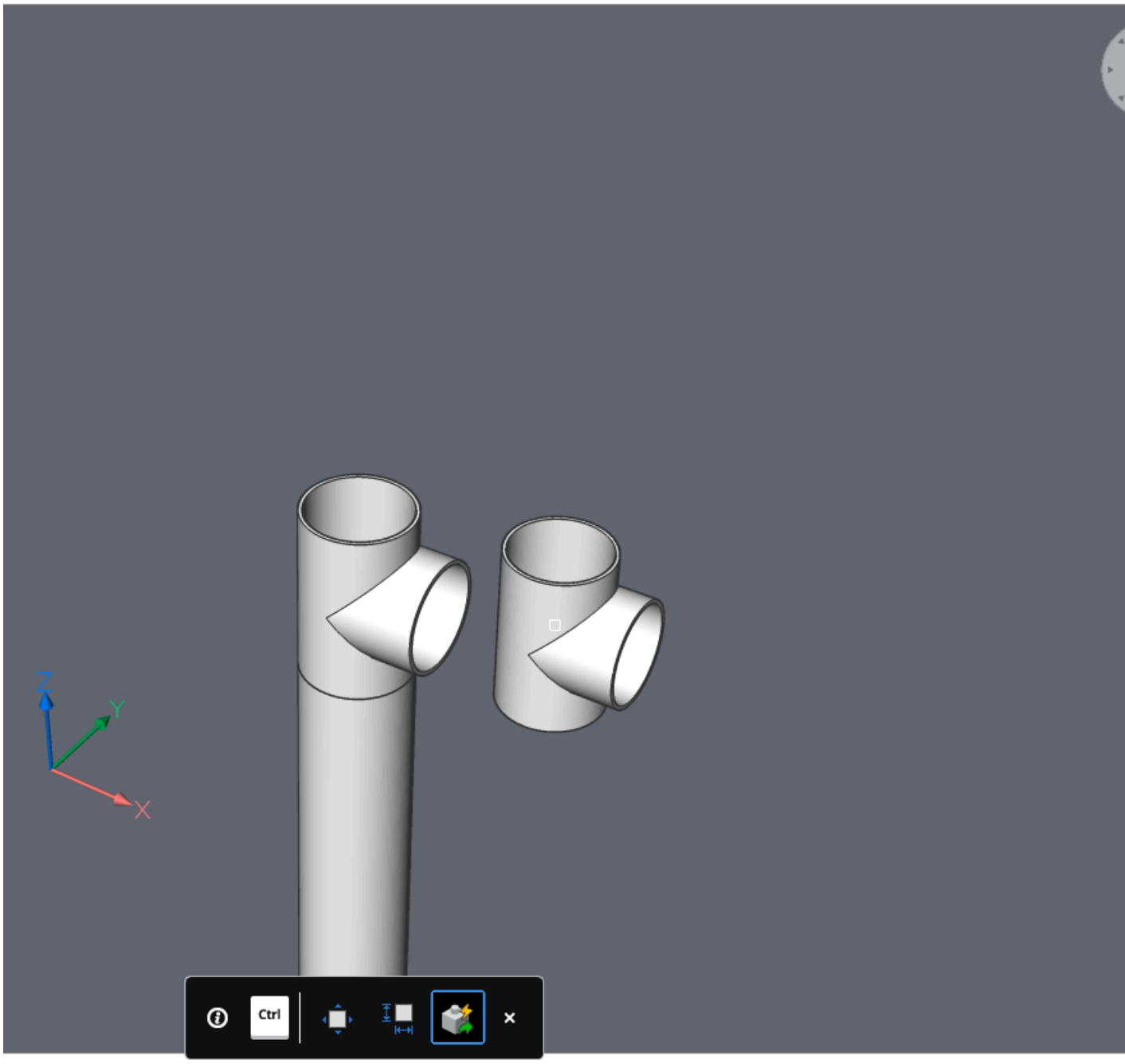
20.1.2.3 Insert BIM Component

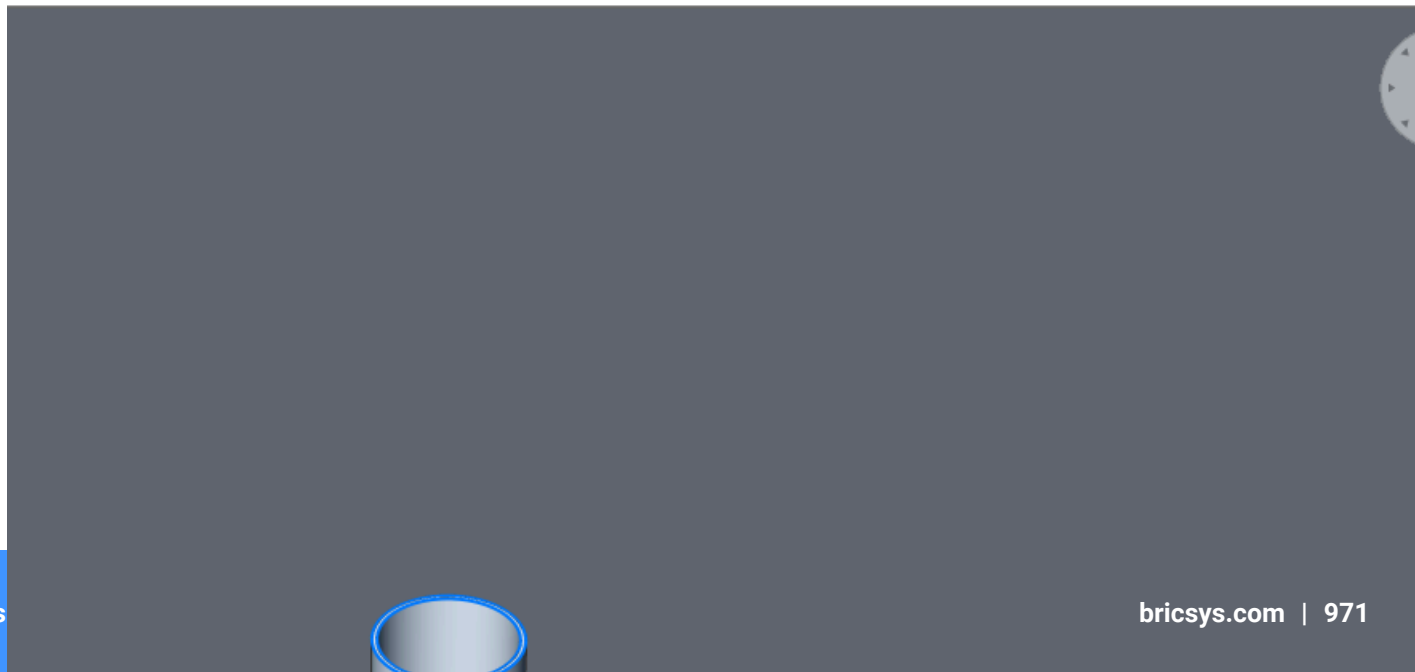
The BMINSERT command now includes a SMART insert option, which automatically adjusts parts to match current and relevant parts in the model file when inserted.

You can select SMART insert by clicking the option in the Command line, or toggle CTRL to the rightmost icon when prompted in the HotKey Assistant.

By detecting and creating 3D constraints between the relevant connection points, SMART insert resizes and attaches your desired component onto the current component/s with assembly constraints and common parameters to form a bigger part of a parametric assembly. This is particularly useful in modeling parametric MEP piping systems.

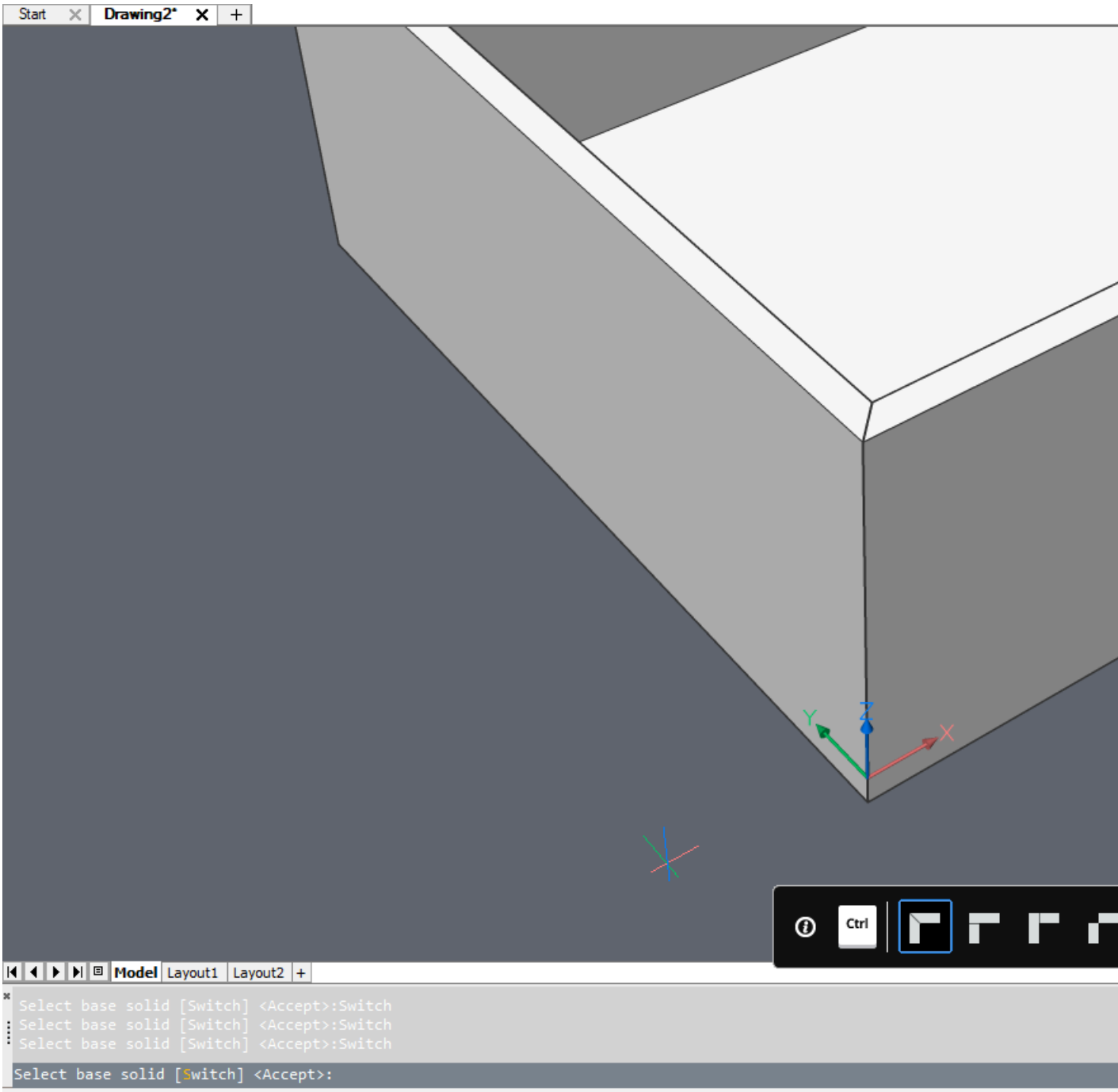

Z
Y
X
Ctrl

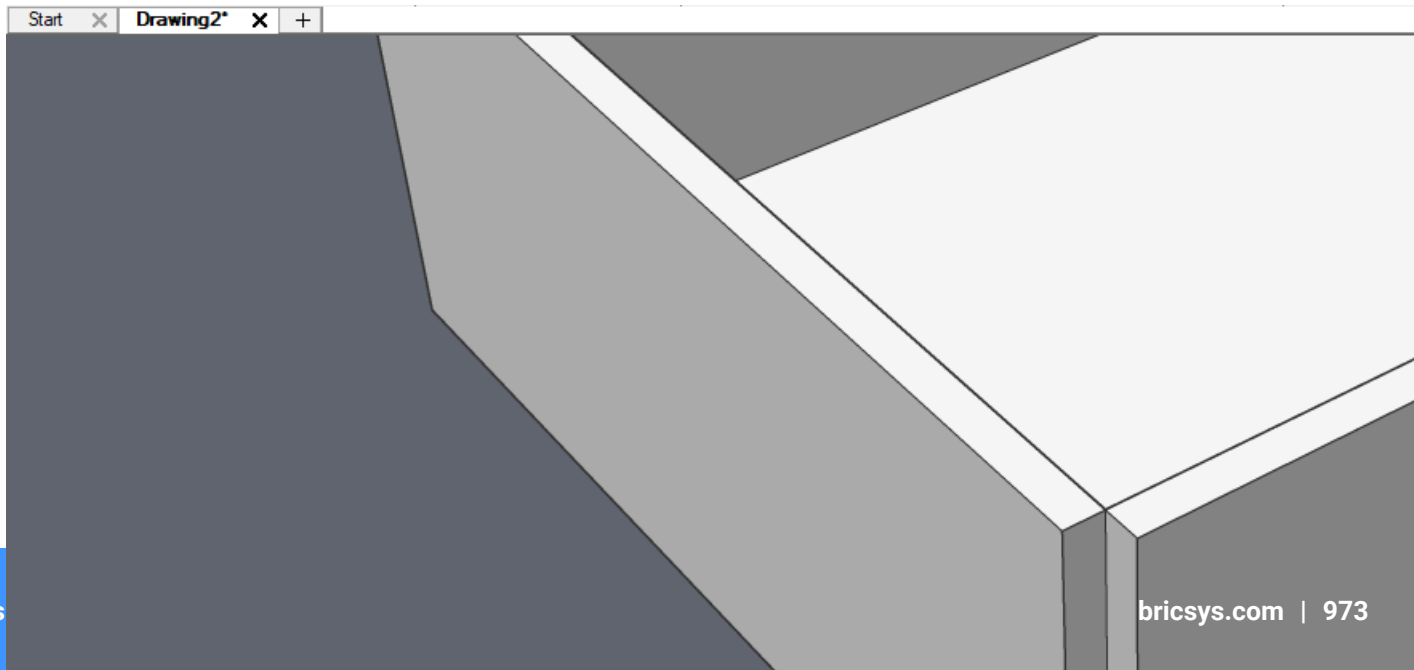

Z
Y
X
Ctrl

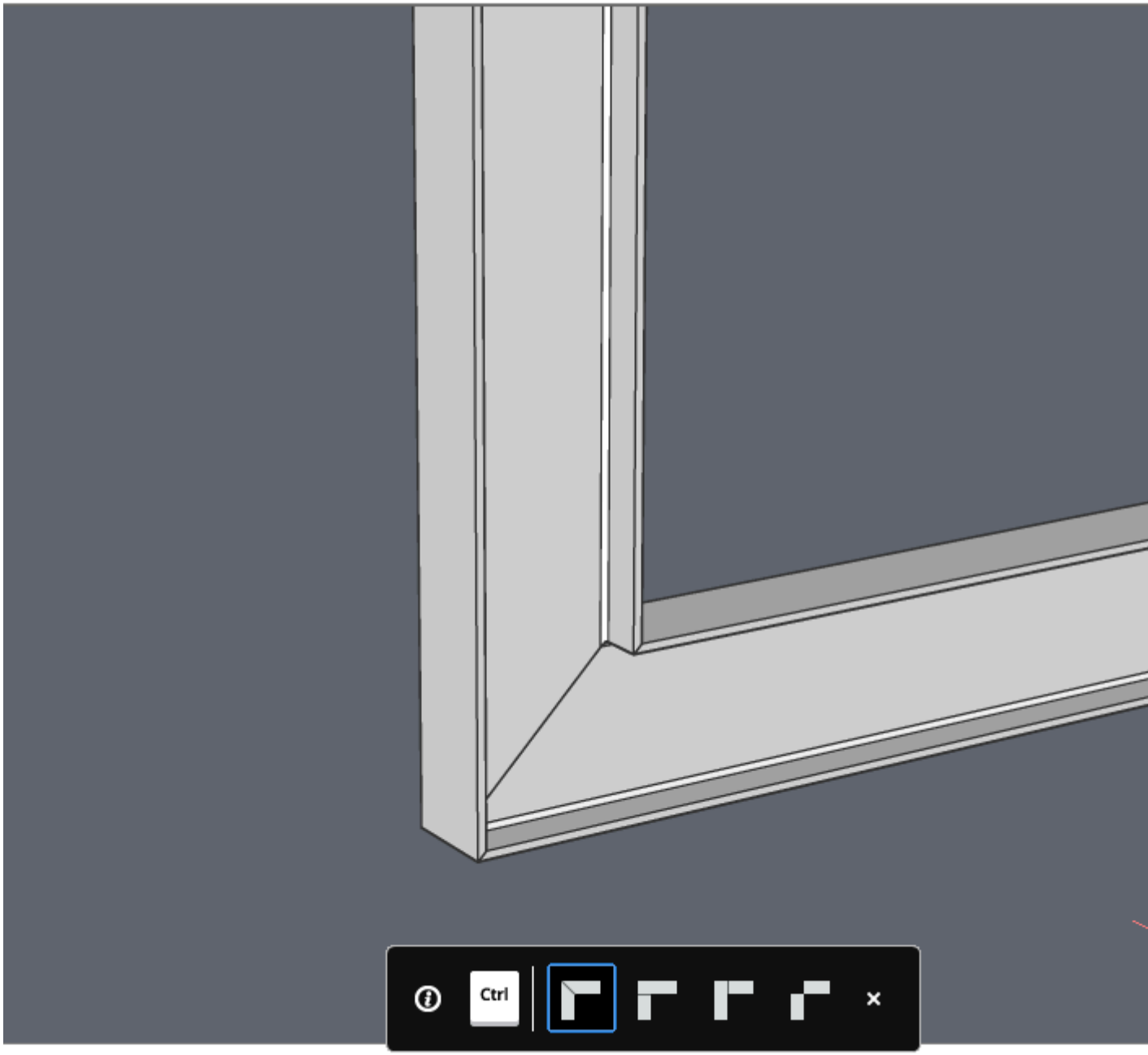


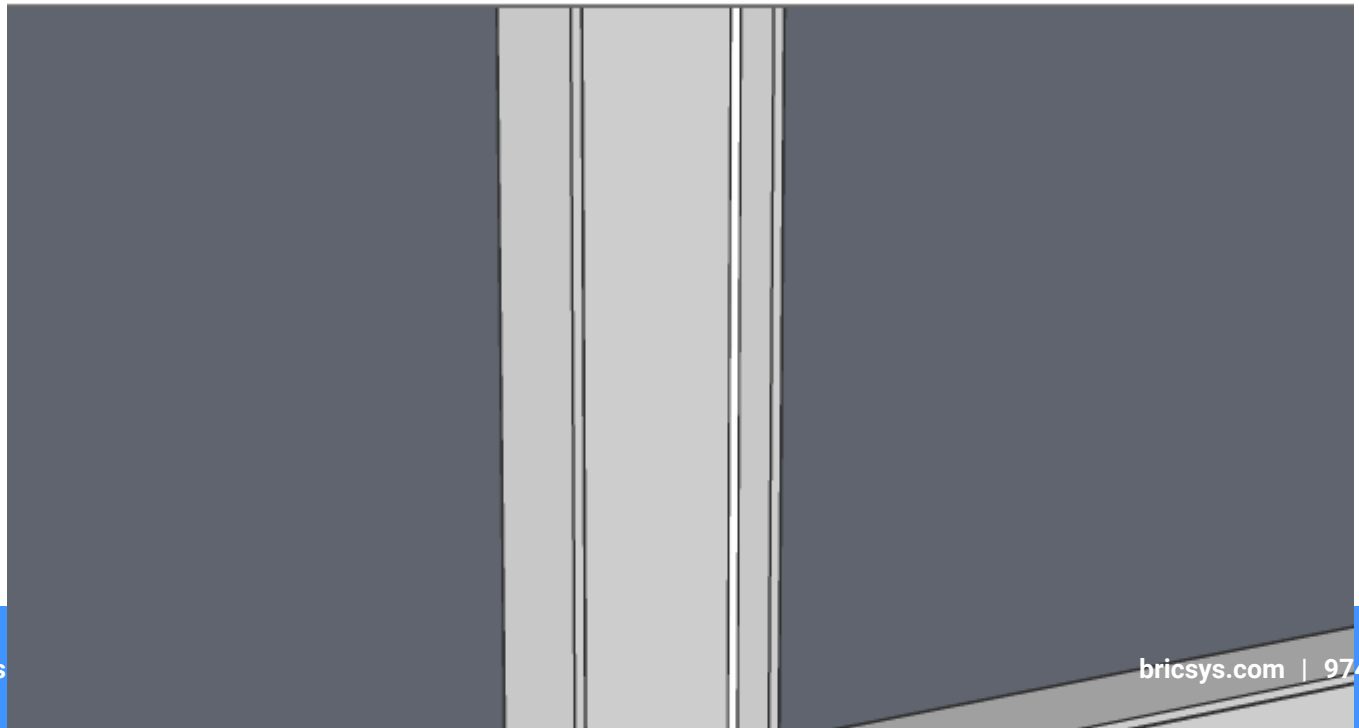
20.1.2.4 Disconnect

The LCONNECT and BIMSTRUCTURALCONNECT commands now include a fourth option: DISCONNECTED. You can cycle through the options with the CTRL button in the HotKey Assistant.


Start
Drawing2*
Y
Z
X
Ctrl
Model
Layout1
Layout2
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:Switch
Select base solid [Switch] <Accept>:


Start
Drawing2*


Ctrl

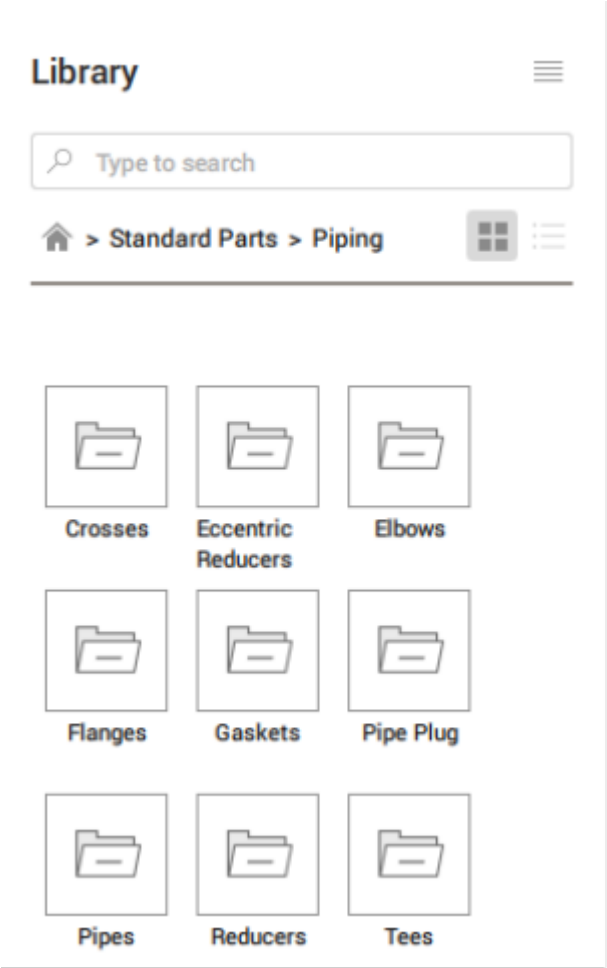


20.1.2.5 Quickdraw

The Quickdraw cursor now supports snapping to (double) lines inside XREFs. Walls now inherit the spatial location of the slab on which they are created.

20.1.2.6 Standards Parts

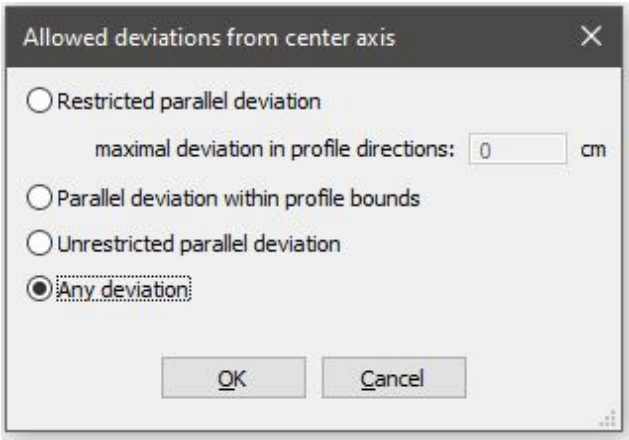
Piping parts were added to the standard parts category of the Library panel.



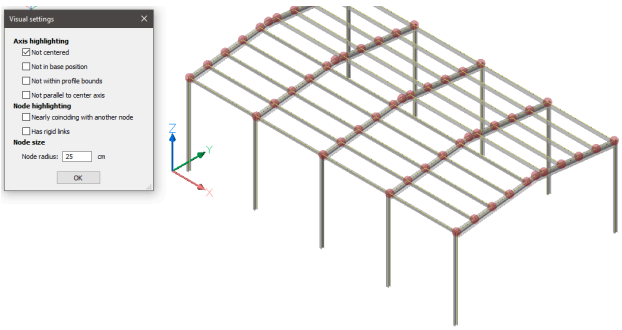
20.1.3 Building data

20.1.3.1 Analytical Model

This command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal, but allows you to further adapt the model to your desire.



The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. You can control the amount of such changes by the deviation settings.



After editing the model, you can export it to an IFC.

20.1.3.2 BIM API

Configurable quantities are now available in the BIM API, as well as the BIM API for Spaces.

20.1.3.3 BIM Compositions

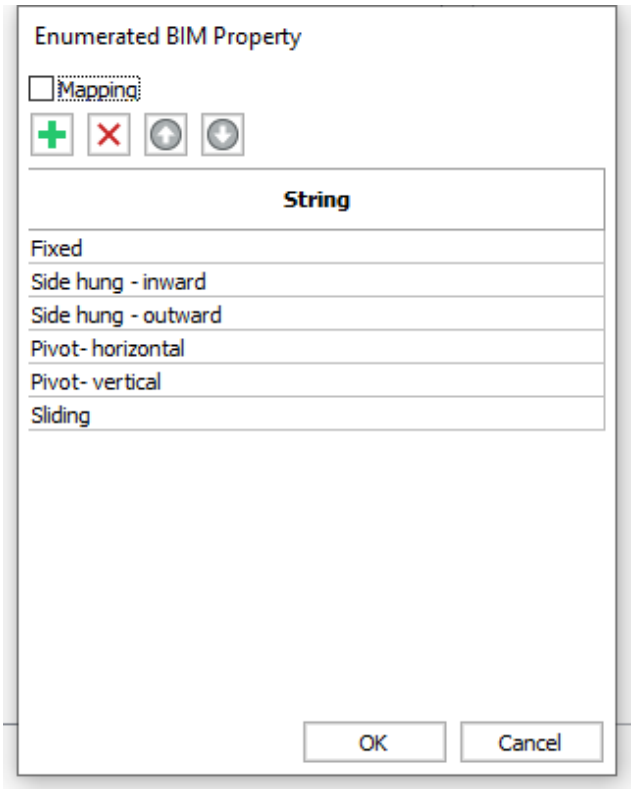
In V21, the render material display and LOD setting have been separated. A new button has been added in the Home/View ribbon panel, next to the LOD button to toggle the rendering composition materials.

LOD settings control whether ply solids are shown or not. If ply solids are not visible and render composition materials is on, only major faces of the multi ply solids will get render materials.

20.1.3.4 Custom BIM Properties

The Per Instance option of a custom property set allows you to enable/disable user-defined properties, on a per-entity basis.

The Enum option allows you to define a fixed number of possible values for custom BIM properties.



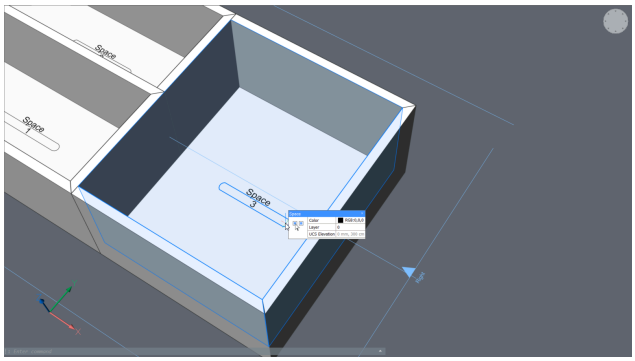
20.1.3.5 Spatial Elements

BimRoom

The BIMROOM command has been replaced by the BIMSPACE command. You can still open BIMROOMs created in V20, but it is not possible to create new BIMROOMs or update the geometry of existing BIMROOMs in BricsCAD BIM V21. Use the Space option of the BIMCLASSIFY command to convert a BIM room to a BIM space. Room properties and interior elevations, if any, are transferred to the corresponding Space.

BimSpace

You can create BIM Spaces entities from enclosed boundaries. When the wall elements enclose a space in your model, you can use the BIMSPACE tool to define a BIM Space. The space footprint preview now displays when executing the BIMSPACE command. These spaces are represented as regular solids so they can be easily manipulated. The Edit option of the BIMSPACE command allows you to add or remove the bounding elements of a space.



BimUpdateSpace

Recalculates a BIM space which is out-of-date, for example after adding or removing bounding entities. In the Properties panel, the Update Method property is added with the options Automatic and Manual.

The new Representation property allows toggling between Solid or Footprint representation of a space.

The Space tag has been added. The Space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. You can change the values of each attribute using the Properties panel.

The new Status property indicates whether a space is up-to-date or out-of-date.

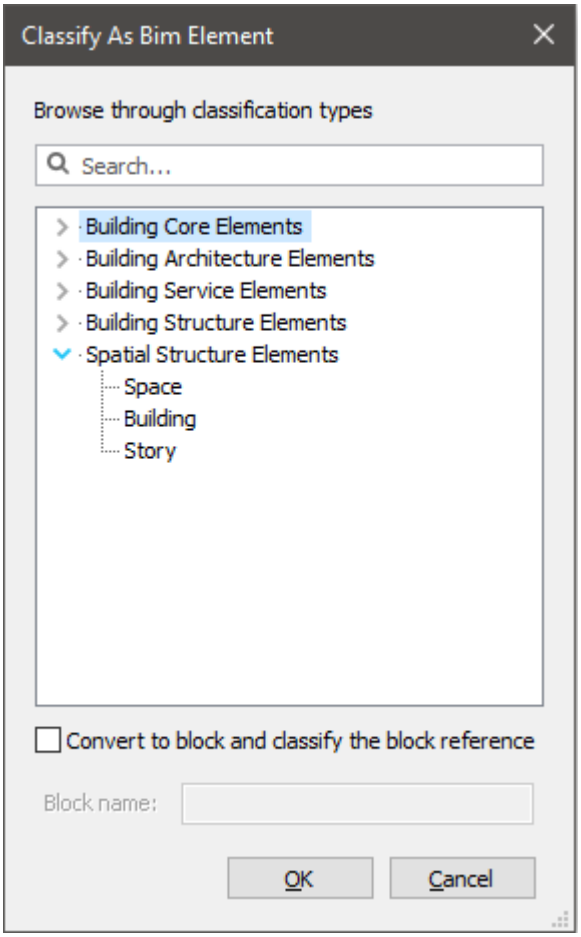
BIM	
Type	Space
Spatial Element Type	Space
Representation	Footprint
Update method	Manual
Status	Up to date
Name	Office
Description	Standard office
Building	Building
Story	Floor 0
Composition	
GUID	1zcNYneH56hRdMV23U7Iw$
Number	2
Interior Elevations	Off

Space icons

BIM Space tool icons for Edit Space, Update Space and Define Interior Elevations are available in the Quad, the Ribbon and the BIM Spaces toolbar.

Spatial Structure Elements

When using the BIMCLASSIFY command, a new category of classification has been added to the dialog box. The name of the new category is Spatial Structure Elements. Under this category you will find Space, Building and Story.

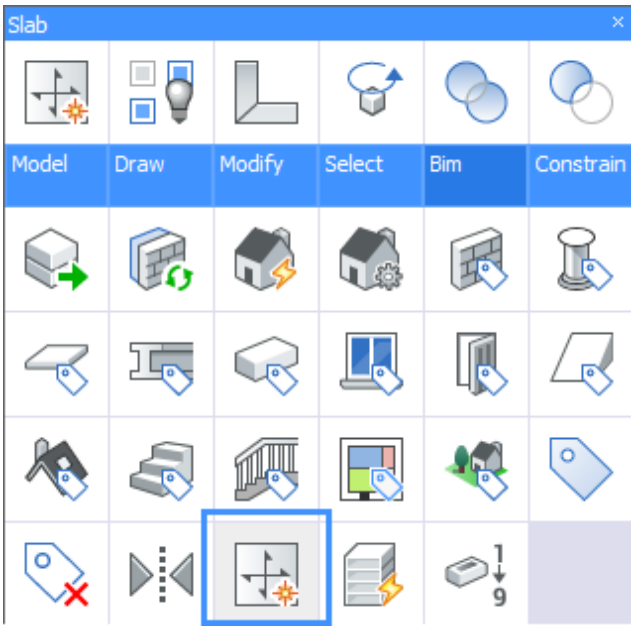


DefaultSpaceHeight

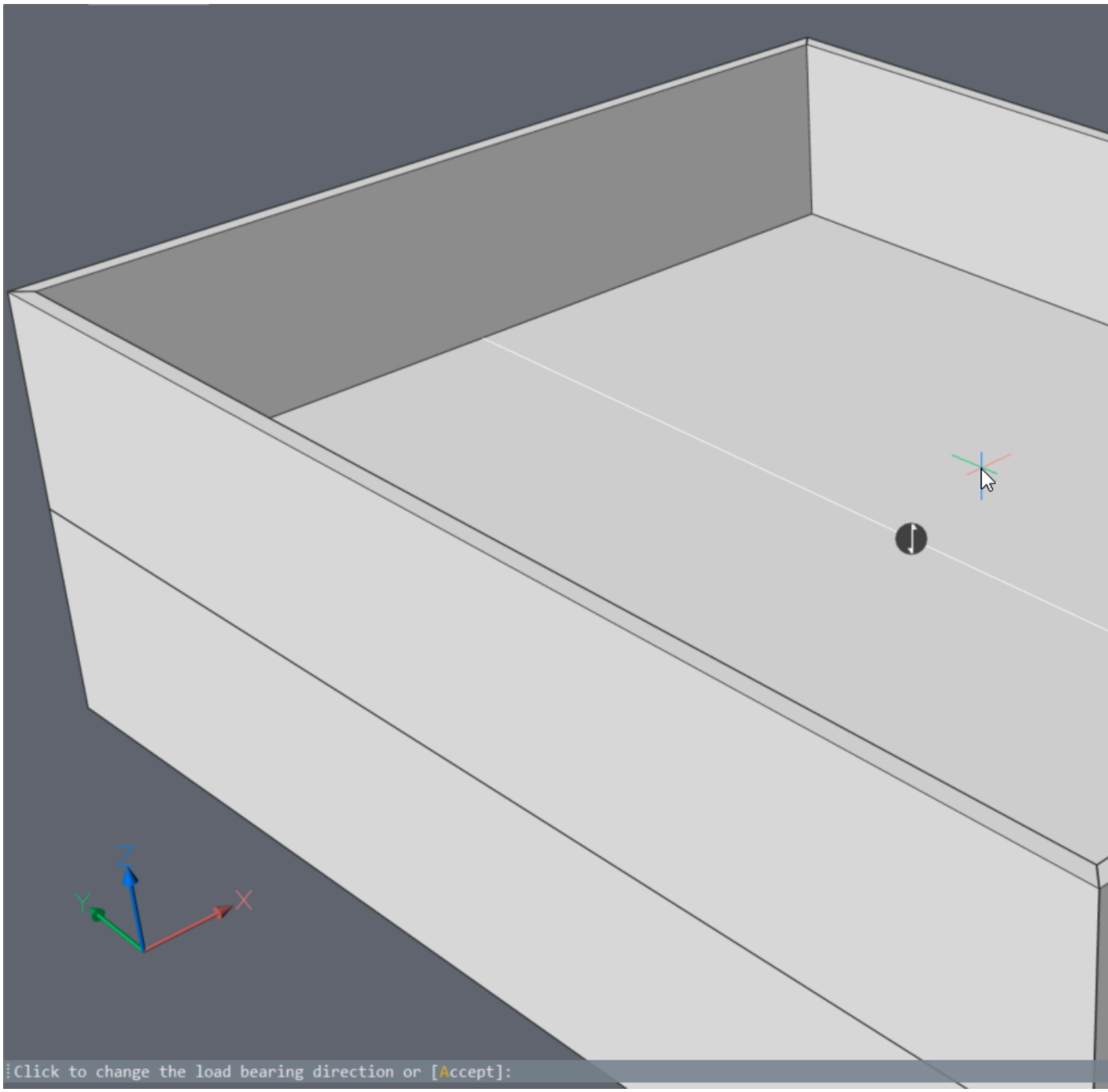
The ROOMHEIGHT system variable is replaced by the DefaultSpaceHeight user preference, the setting applies to Spaces.

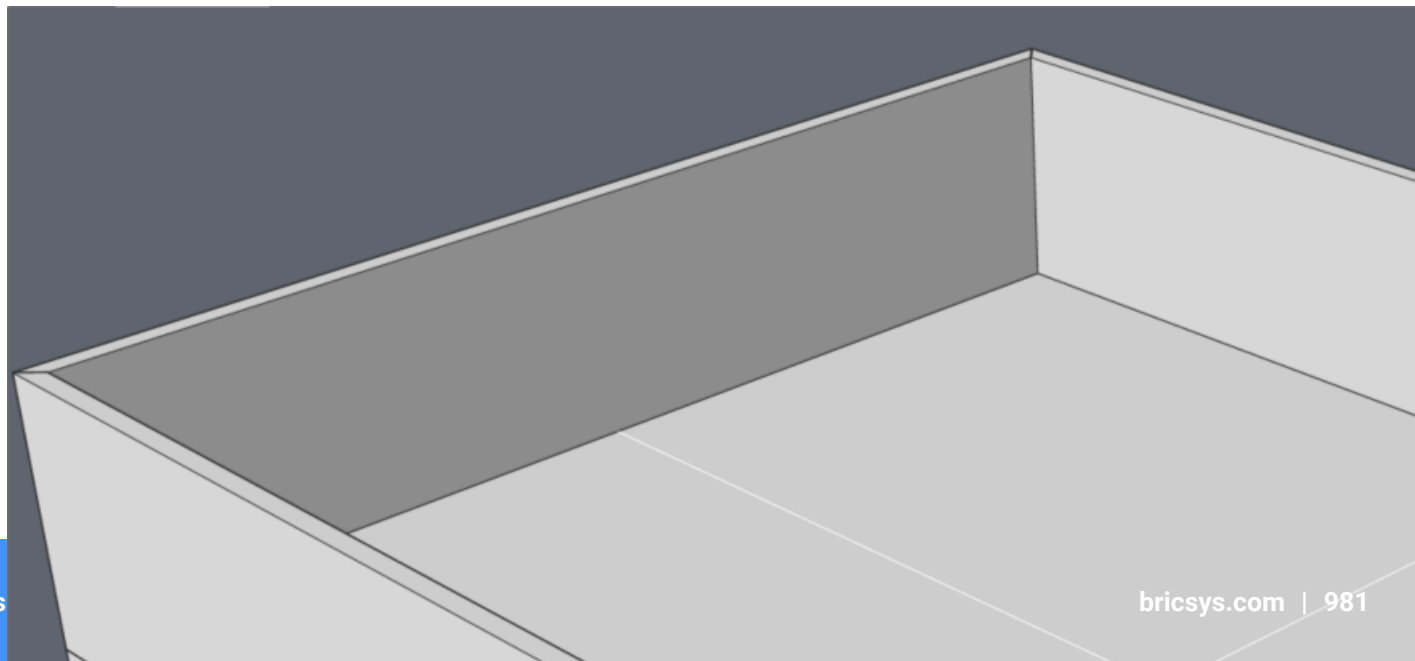
20.1.3.6 BIMSETLOADBEARINGDIRECTION

With the new tool button that launches the BIMSETLOADBEARINGDIRECTION command in V21, it is easy to set your loadbearing direction of your slab through the Quad.



As a result, you will get a black dot on your slab. Once you click on it, you can change the load-bearing direction of the slab in one of the two main directions of the slab or in both.


Z
Y
X
Click to change the load bearing direction or [Accept]:



20.1.3.7 Python scripts in BricsCAD BIM

Nota: This is a V21 Beta feature.

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a .py python script.

The python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

Executing BIMPYTHON opens a Windows dialog box where you can navigate and run your python script in .py format.

The new BIMACTIVATEPYTHON command allows you to activate and deactivate the virtual python environment in BricsCAD BIM, by toggling the values 0 and 1 for **OFF** and **ON** respectively. You need to ensure it is turned **ON** before using BIMPYTHON.

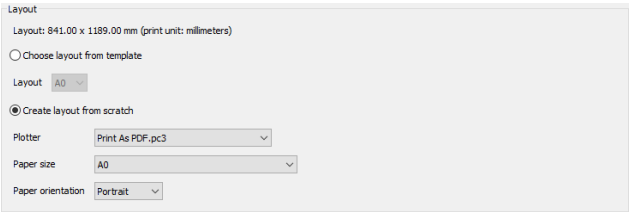
The Python Programming Language, together with its standard libraries, is embedded in BricsCAD BIM so there is no need for you to install them separately, unless you have custom packages and libraries which you wish to use in your scripts.

20.1.3.8 Project Browser

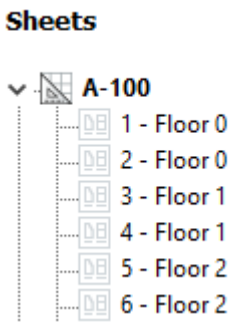
BIM project data are now stored in an external project **.bsyslib**, instead of the previously used **.projectroot**. When opening a pre-V21 BIM project, you can convert the **.projectroot** to a V21 project **.bsyslib**. To simplify the opening of a BIM project at a new location, paths to **.bsyslib** files are now stored with a relative path.

Sheets

It is now possible to create a layout from scratch in the Sheetset setup dialog box.



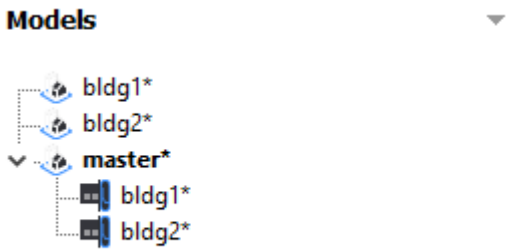
The Project Browser displays the Sheet Number (if it exists). The Sheet Number is assigned by the **Create Sheets** procedure. Sheet View Numbers are assigned starting from 1 for all section results created by the BIMSECTIONUPDATE command.



Models

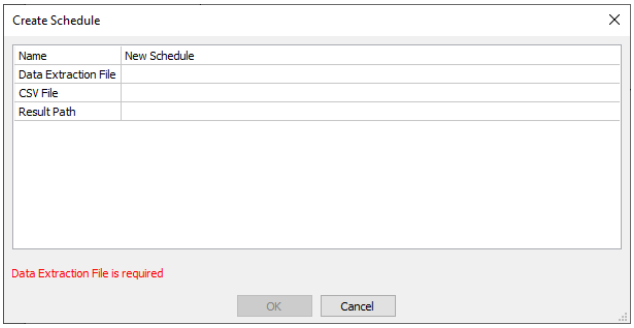
Master drawings and external references are listed in a tree-structured manner in the Models tab of the Project Browser:

A balloon is available to fix a project libraries' connection to a drawing sheet.

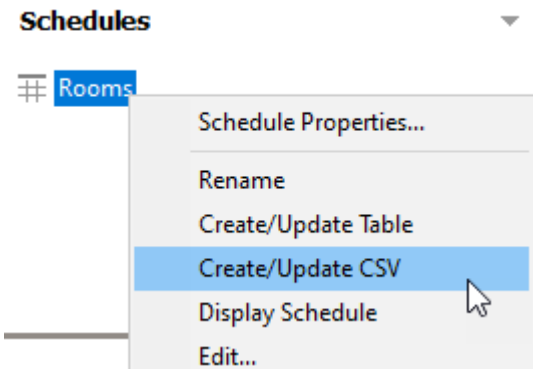


Schedules

The **Add Schedule** wizard, available in a context menu, allows you to quickly add a BIM schedule without the creation of a new .dxd file. The use of relative paths is now supported for Schedules.



The **Create/Update Table** and **Create/Update CSV** options are available in a context menu for creating or updating a schedule or a CSV output file.



20.1.4 Project collaboration

20.1.4.1 IFC IMPORT/EXPORT

A mechanism has been implemented to map IFC/BIM entities to BIM/IFC entities, to control which entities are imported/exported. BricsCAD BIM also added IFC Export of Annotation entities.

You can do this mapping by using mapping files. The IfcImportMappingMath and IfcExportMappingPath user preferences set the paths to the necessary mapping files. These files are simple text files containing some rules.

This mapping can be used to export BIM entities as IFC entities of another type, or to skip certain types altogether. An example of such an export file can be:

- BimDbColumn IfcMember
- BimDbBeam IfcMember
- BimDbFurnishingElement skip

In this example, columns and beams in the BricsCAD BIM model will be exported as Members in the IFC file, and furnishing elements will be skipped in the export process.

A similar process can be used when importing IFC files. An example of such an import file can be:

- IfcWall BimDbSlab
- IfcSlab BimDbWall

* skip

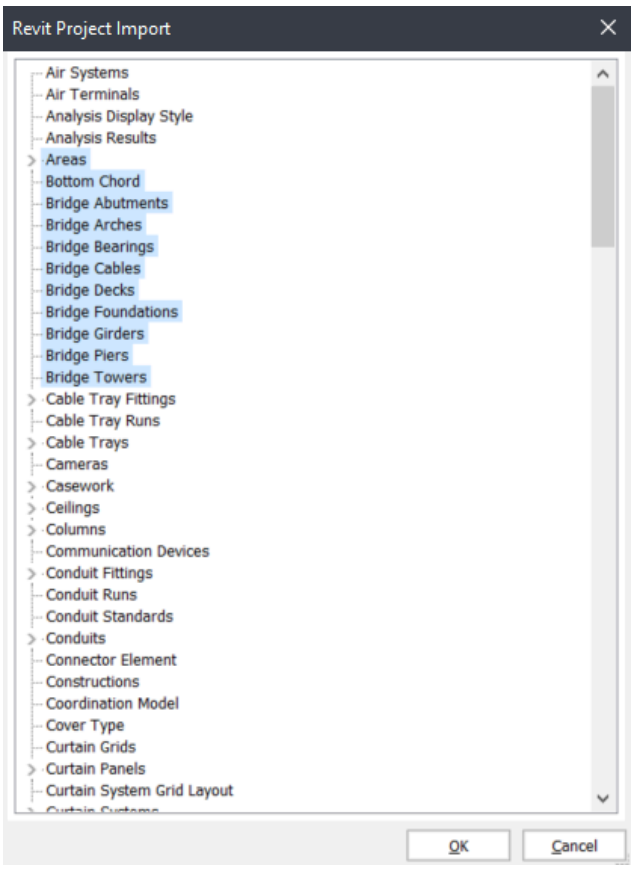
In this case, walls will be imported as slabs and slabs will be imported as walls. No other entities will be imported, as the asterisk means that only listed entity types will be taken into account, all unlisted entity types will be skipped.

The IfcCreateUniqueGuid user preference has been added to enable/disable the creation of unique GUIDs for nested elements inside classified blocks and Xrefs.

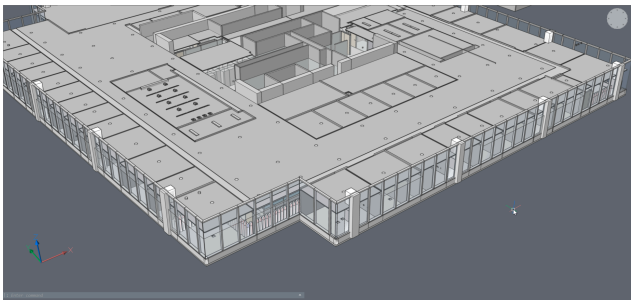
The flow connection point is exported to IFC as IfcBuildingElementProxy. Only 2D geometry or a block reference with 2D geometry can be classified as a flow connection point.

20.1.4.2 Importing RVT files

When importing Revit Objects using IMPORT, you can choose for Import all the solids or only import as an Overlay. You can filter the import based on Revit Views, Categories, Levels or WorkSets.



This is an example for the option Categories, where you can map specific categories for your RVT import. Colors and transparencies are now imported when importing a RVT file.



20.1.5 Design documentation

20.1.5.1 Sections

Activate SectionView

This new command allows you to look at the model from a section plane's perspective, without generating the section. This view allows you to manipulate the 3D model from a 2D point of view. To exit the active view, just orbit around the model.

Open BimSections

When switching to the model from a sheet view using the BIMSECTIONOPEN command, the model is rotated and zoomed in to the same zoom level as the sheet view.

Available properties and property values display in the filter parameters form.

The BIMSECTIONOPEN command now allows you to transition between the sheet viewport and drawing model seamlessly at the same zoom level, without the need to re-orientate yourself and the model to where you have been working earlier on the sheet.

When you switch between a sheet viewport to the drawing model using the updated BIMSECTIONOPEN command, the drawing model displays and orientates itself to the same zoom level as in the sheet viewport, regardless of the scale of the Standard Scale or Annotation Scale of the drawing or viewport.

Update BimSections

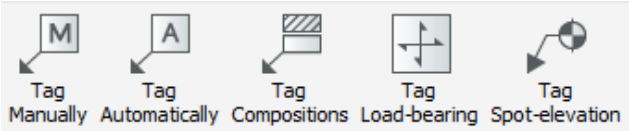
The associative data for 3D solids, needed for section generation and tagging of the section results by the BIMTAG command, are generated automatically in BricsCAD BIM V21 as you model, unless the new GENERATEASSOCATTRS (Generate Associative Attributes) system variable is set to Off.

Interior Elevations

Since Rooms in V20 and older are now deprecated and replaced by Spaces, the Interior option of the BIMSECTION command now supports Spaces.

20.1.5.2 Tags

The BIM Tags tool now includes options of tagging load-bearing directions and spot elevations on viewports on Paper Space. Both tag types work on plan and section drawings. Spot elevation tags also work on elevation drawings.



Automatically generated tags can have an offset, which is defined through the Offset attribute in TagTypeToStyle.xml found in the **SUPPORTFOLDER** > **Bim** > **Sections** folder. If the offset is not specified, the center of the tag block will be used.

Similarly, automatic and manually placed tags can be rotated if the autoRotation attribute is set to **true** in TagTypeToStyle.xml.

Tag Load-bearing

Load-bearing tags is only applicable if the load-bearing direction of the slab has already been set. Refer to the BIMSETLOADBEARINGDIRECTION command for more information.

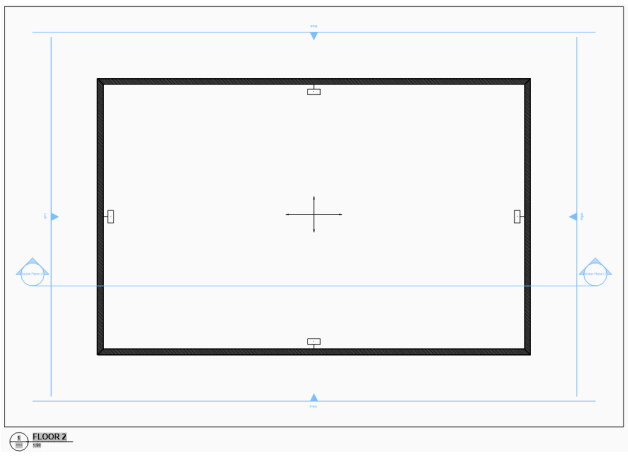
Tag Spot-elevation

The spot elevation level tag displays the actual elevation of a selected point of a BIM object in a section view. The height is measured relative to level 0.

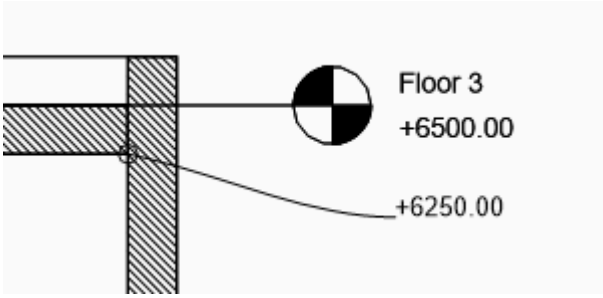
Tags in general

On Paper Space, click **Tag Load-bearing** from the ribbon and pick point on a sectioned BIM element as prompted. You will be prompted to pick a point on your desired sectioned BIM element or to change the current mleader style settings, like how you set BIMTAG in V20.

For example, you can have a two-way load-bearing span graphic tag on structural drawing plans.



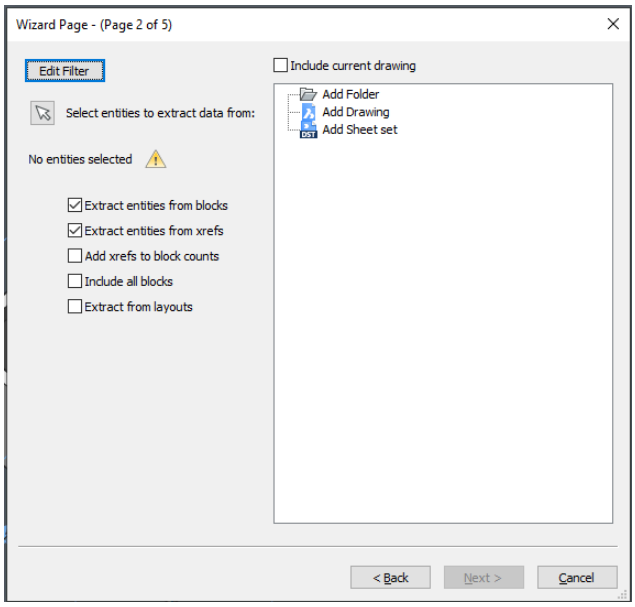
Or you can have spot elevations displaying elevation levels relative to the WCS.



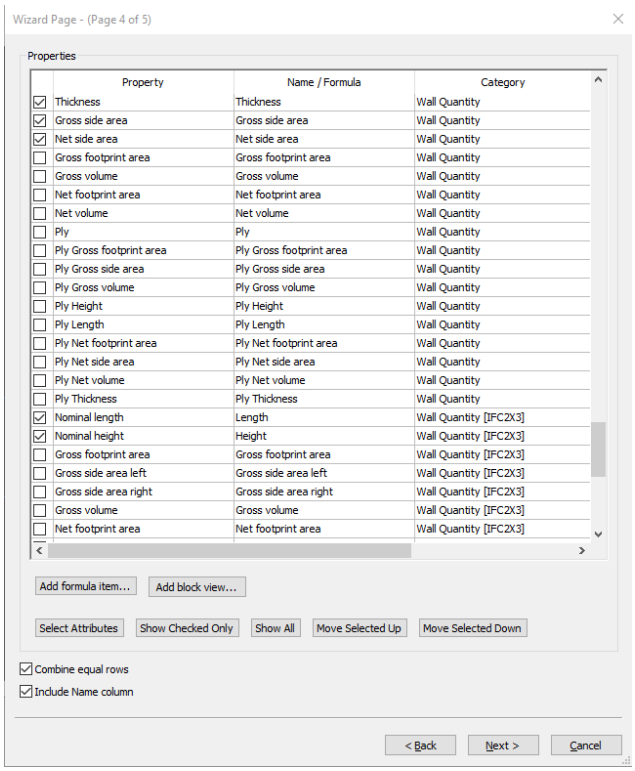
Like all other BIM tags, they are customizable multileader blocks. Therefore, for example, Spot Elevation tags can be further adjusted to suit your graphic and information preferences.

20.1.5.3 Data Extraction

The DATAEXTRACTION command now allows you to include all blocks and also extract from layouts in the new filter rule settings. You can now extract composition and physical material data.



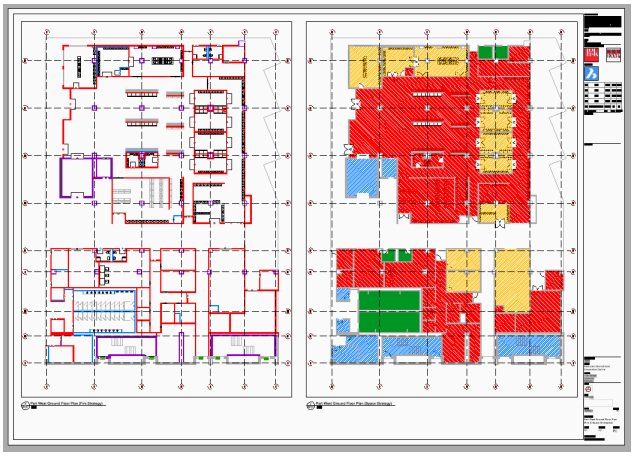
You can now extract and schedule quantity data based on IFC2x3 Element Types.



20.1.5.4 Drawing Customizations

Drawing Customization is a new tool that is available and only accessible from the panel. All the controls of Drawing Customizations are represented as graphic icons within the User Interface panel, such as the **+** symbol in a blue circle to create a new template. You can duplicate, rename or delete your customization templates.

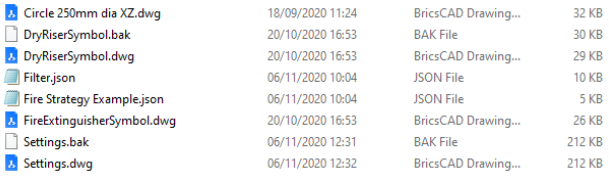
It allows you to create view templates to control and customize the visibility settings of your drawings on layouts. The Drawing Customizations tool in BricsCAD is operated by 3 principles – Entity Customizations, Styles and Filter Rules.



Drawing Customization templates are stored within the Customization folder located in the SUPPORTFOLDER, under **Bim folder** > **Sections subfolder**.

Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the Create New buttons.

Settings.dwg contains the values created in the Style tab in the form of layers and if you have created any custom properties that you wish to use as a Filter parameter, you have to ensure they are also reflected in the Settings.dwg.

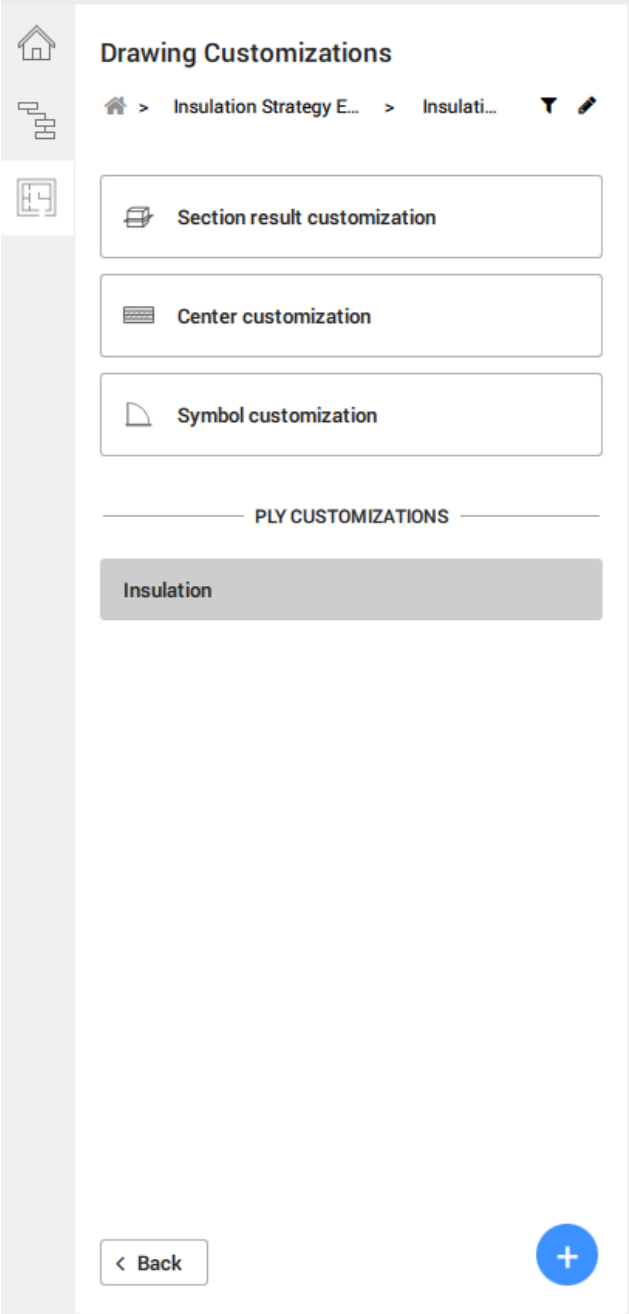


Circle 250mm dia XZ.dwg	18/09/2020 11:24	BricsCAD Drawing...	32 KB
DryRiserSymbol.bak	20/10/2020 16:53	BAK File	30 KB
DryRiserSymbol.dwg	20/10/2020 16:53	BricsCAD Drawing...	29 KB
Filter.json	06/11/2020 10:04	JSON File	10 KB
Fire Strategy Example.json	06/11/2020 10:04	JSON File	5 KB
FireExtinguisherSymbol.dwg	20/10/2020 16:53	BricsCAD Drawing...	26 KB
Settings.bak	06/11/2020 12:31	BAK File	212 KB
Settings.dwg	06/11/2020 12:32	BricsCAD Drawing...	212 KB

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.

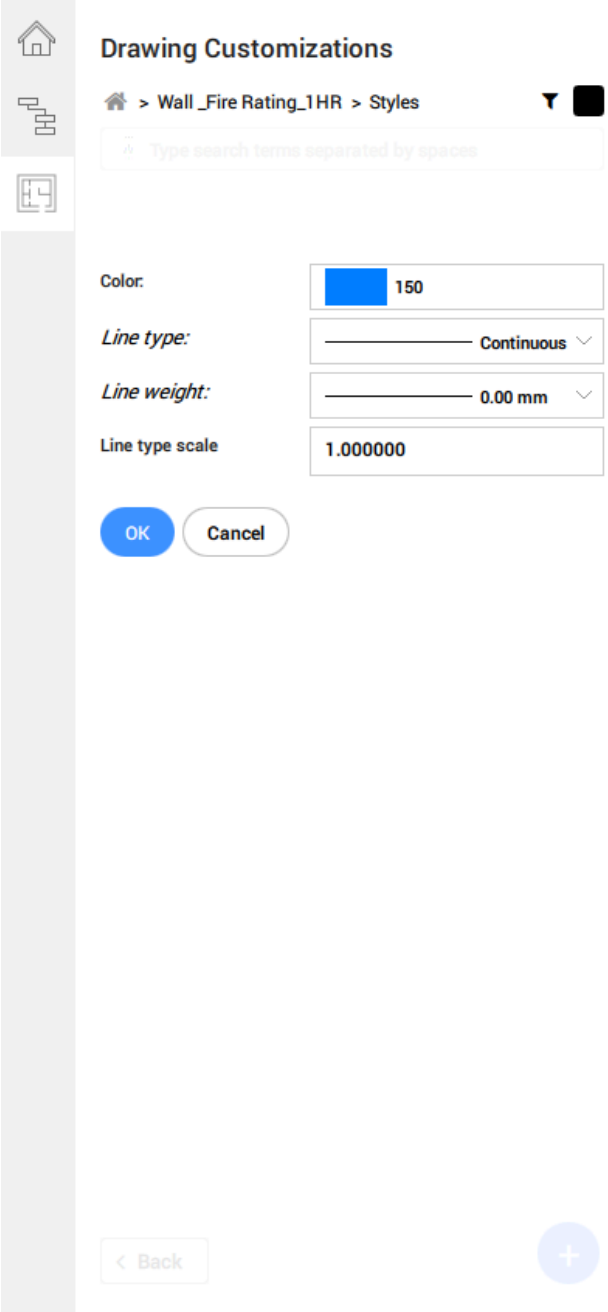
You can further apply your saved Styles to specific layers within your external symbol source, thereby allowing you to retain a general symbol drawing across several customization templates with different output results.

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.

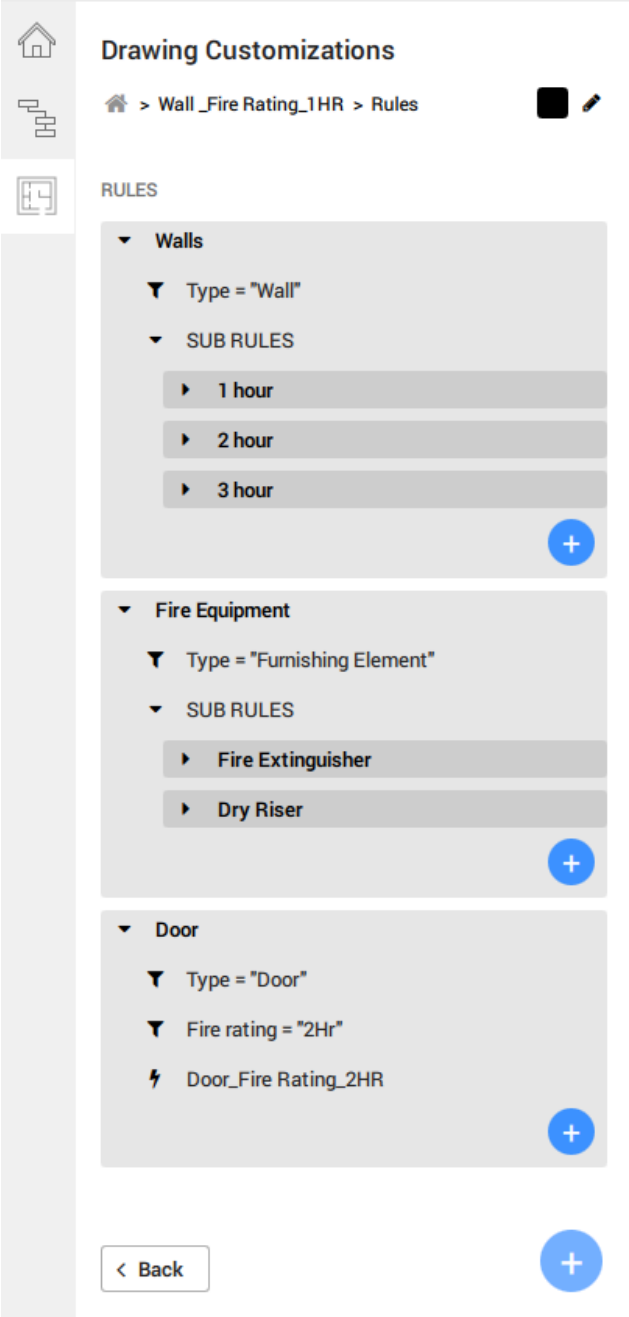
The operation of the Styles tabs is like how Entity Customization tabs can be controlled.

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.

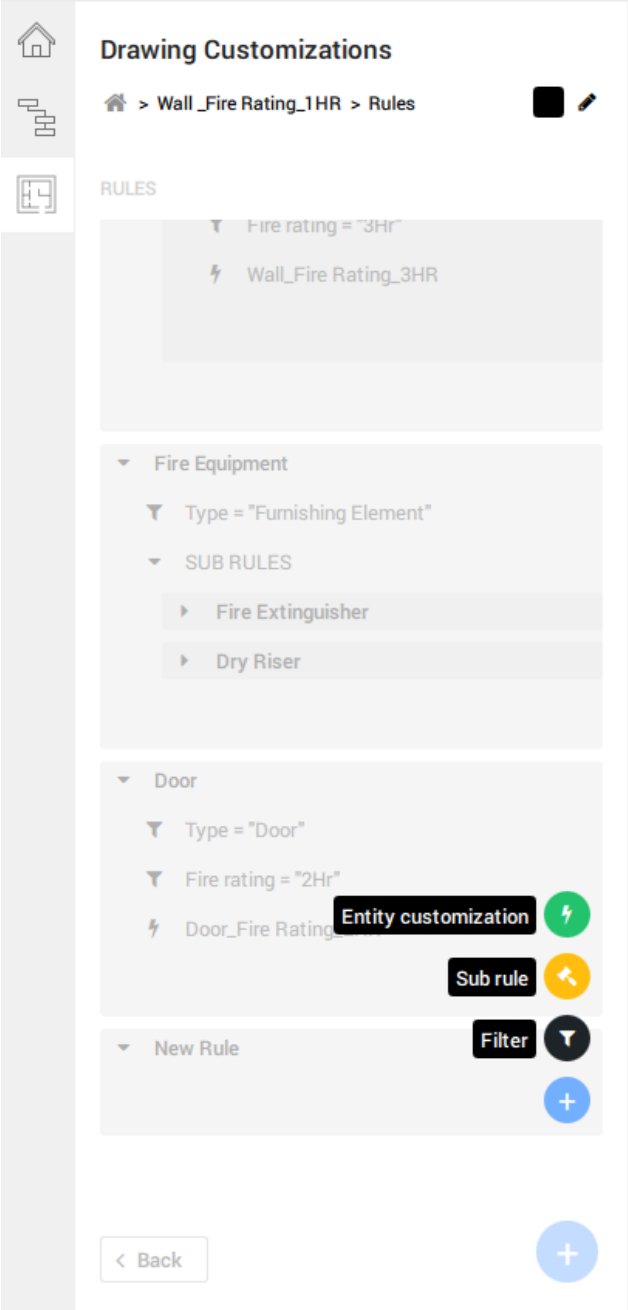


Filter Rules

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.

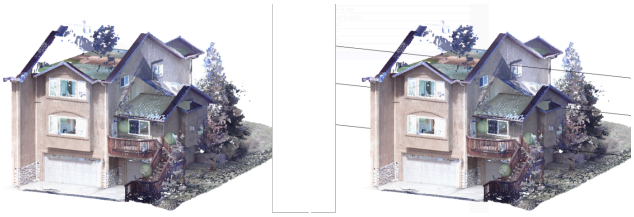
20.1.6 Point clouds

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.

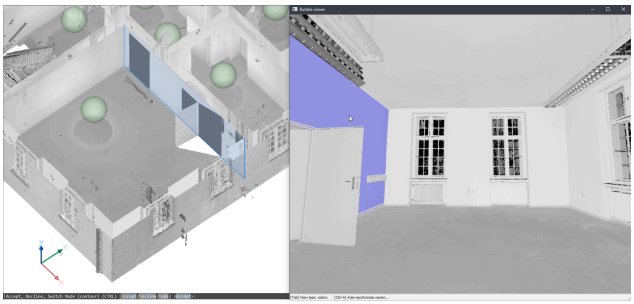


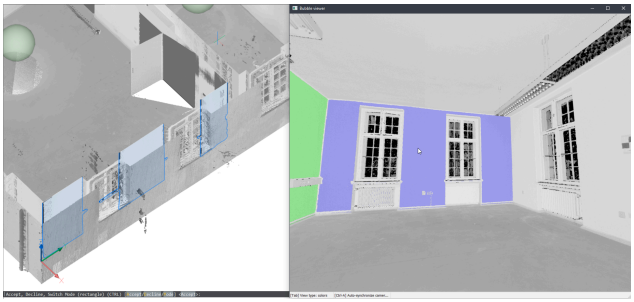
20.1.6.2 Planar Fit

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.

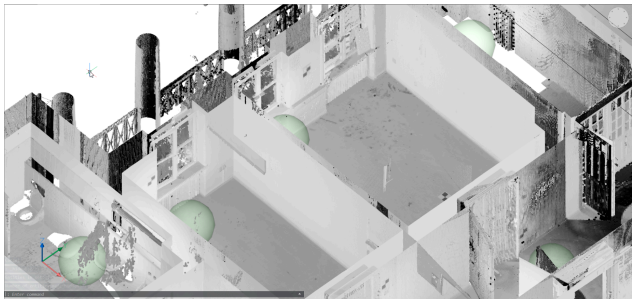


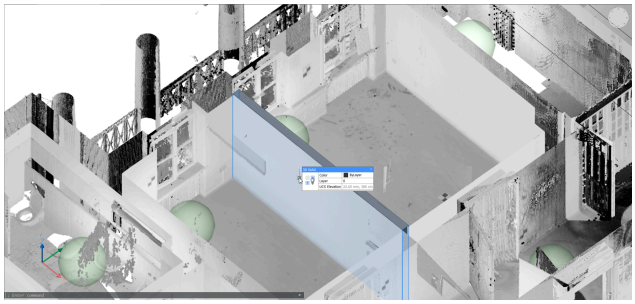


In model space

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:

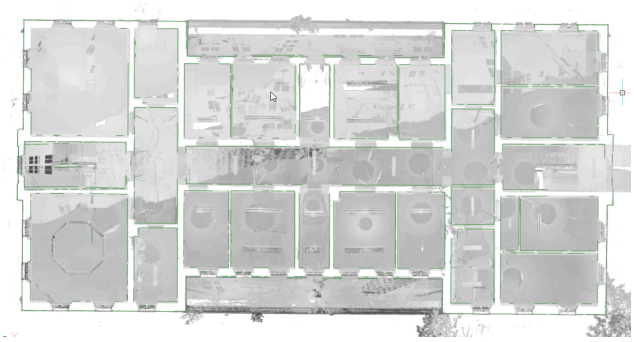
- It can create larger surfaces where only parts are visible in each scan position
- It can detect wall and slab thickness since it can take the opposite surface into account.





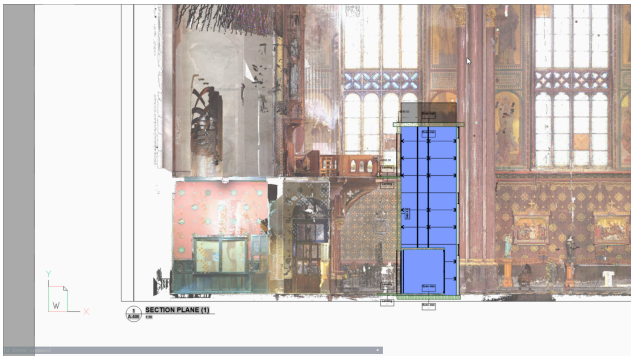
20.1.6.3 Point cloud projection

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 Guida introduttiva

20.2.1 Creating a BIM model

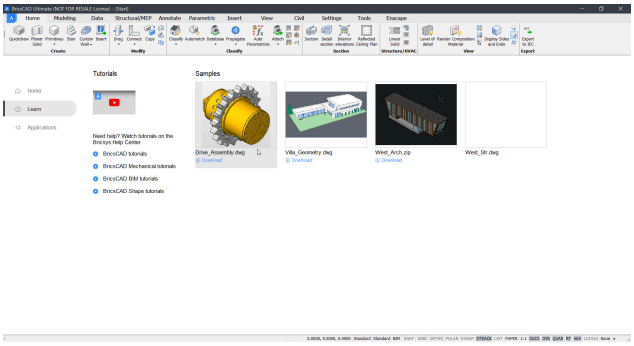
Do the following:

1 Start BricsCAD and choose the BIM workspace on the **BricsCAD Launcher**. In this workspace, all BIM tools and resources are available in the Ribbon, the Quad and various dockable panels.

2 On the Start tab, select a **BIM** template.

3 Click the **New Drawing** button to create the model.

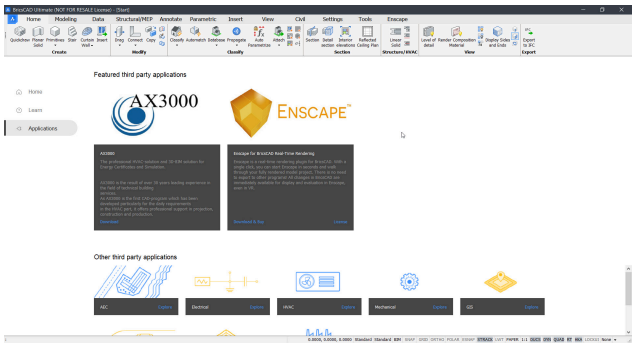
20.2.1.1 Learn



Here you will find links to tutorials for Shape, Core, BIM and Mechanical. You can also find some sample files that you can download and try out yourself.

20.2.1.2 Applications

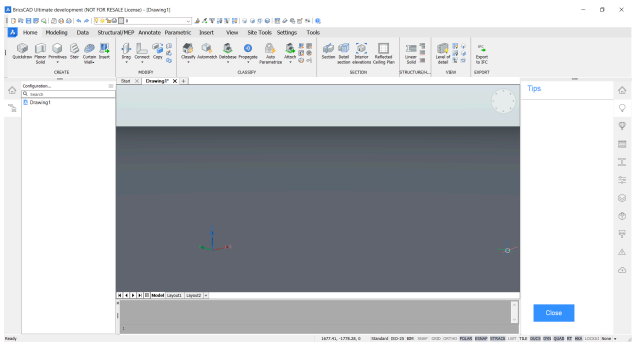
Here you can find some of the third party applications that are available for BricsCAD. You can find more applications on our Applications Store website.



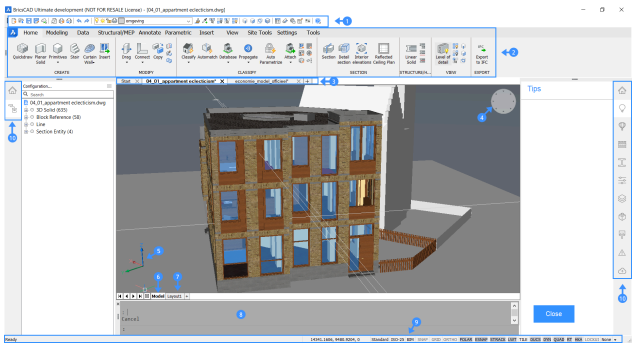
20.2.2 Exploring the interface

20.2.2.1 User interface

The BricsCAD BIM interface includes all tools, commands, and settings required to create and edit building elements. Start by creating building elements, opening an existing drawing or importing a building model.



20.2.2.2 Interface elements

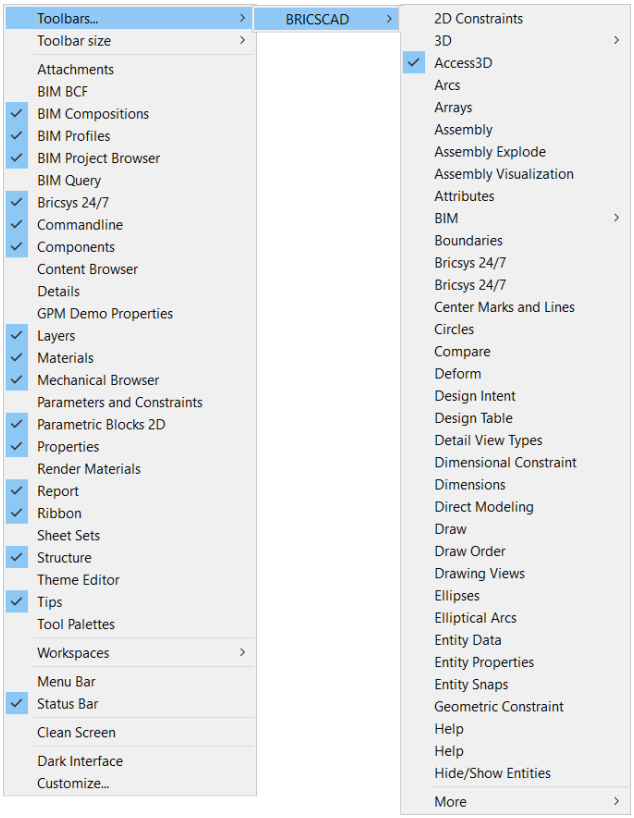


1. Toolbar

In BricsCAD, toolbars are arranged in collections of similar functions within a bar of tools. Each function is displayed with an icon and is categorized by topic. Toolbars are available for all command categories. For example, a BIM toolbar is one of the tool categories available in BricsCAD BIM. This toolbar includes the most used BIM tools for modeling. It is possible to customize existing toolbars or create your own custom toolbars.

By default, the Access3D Toolbar is shown on top of the screen.

Nota: To open more toolbars you can right-click, choose toolbars and check the topics you want.

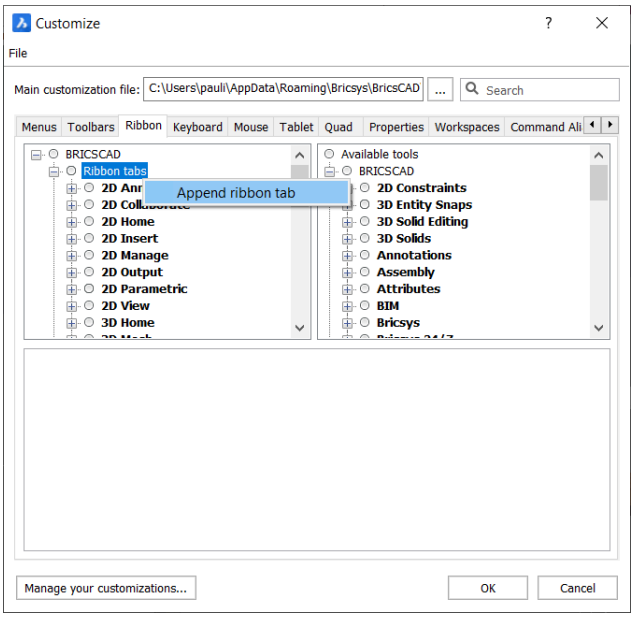
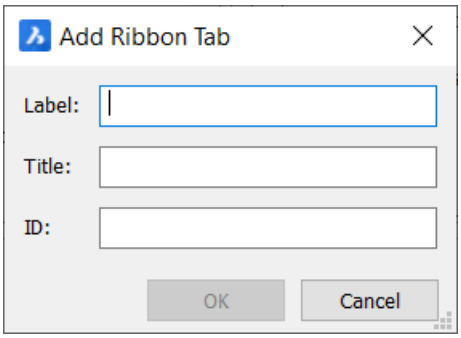


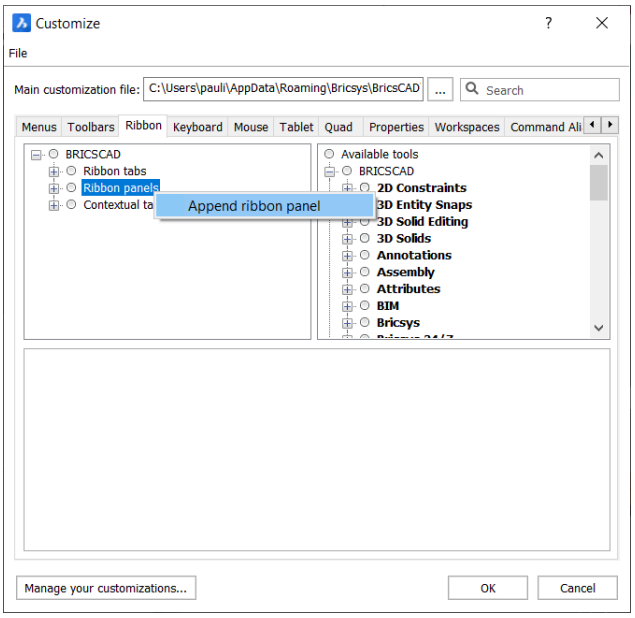
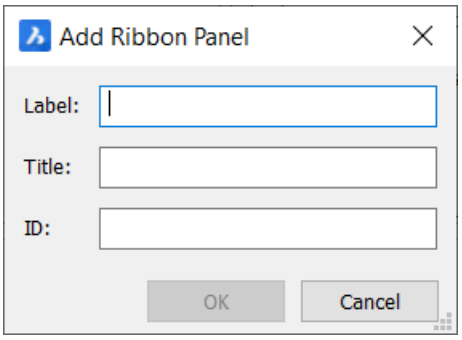
2. BIM Ribbon

A ribbon organizes tools in a series of panels (11) which are grouped in different tabs (12). Each tab contains panels and each panel contains a group of buttons and flyouts.

Nota: The content of the ribbon depends on the current workspace (see Editing In Workspace).

Nota: Use the RIBBON and RIBBONCLOSE commands to show or close the ribbon.

Creating a ribbon tab	Open the **Customize** Dialog box and click on the **Ribbon** tab. Right-click the **Ribbon tabs** group and choose **Append ribbon tab** in the context menu.  Fill out the fields in the **Add Ribbon Tab** dialog box:  **Label**: Caption on the ribbon tab. **Caption**: Used in the Select Ribbon Tab dialog when selecting ribbon tabs for a workspace. **ID**: A unique name to identify the ribbon tab.
Editing a ribbon tab	Open the **Customize** Dialog box and click on the **Ribbon** tab. Expand the **Ribbon tabs** group and you can edit each tab.

Creating a ribbon panel	Open the **Customize** dialog box and click on the **Ribbon panels**. Right-click the **Ribbon panels** group and choose **Append ribbon panel** in the context menu.  Fill out the fields in the **Add Ribbon Panel** dialog box:  **Label**: Caption of the panel in the ribbon. **Title**: Used in the Select Ribbon Panel dialog when selecting the panels for a ribbon tab. **ID**: A unique name to identify the ribbon panel.
Editing a ribbon panel	Open the **Customize** dialog box and click on the **Ribbon** panel. Expand the **Ribbon panels** group and you can edit each panel.

3. Working with Document Tabs

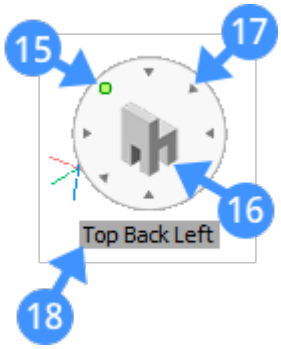
Document tabs allow you to:

- Easily switch between all open drawings.
- Drag and drop the tabs to change their order.
- Click the **X** icon (13) at the right-hand side of the tab to close it.

- Right-click on a document tab to display a context menu.
- Click the **+** icon (14) at the end of the document tab to open a new drawing.

4. Look from Widget

The Look From Widget appears in the upper right corner of the drawing area. When you hover the cursor over the widget, a small green rectangle appears (15), as does the preview of a chair (16). Clicking the cursor over one of the triangle shapes (17) shows what the 3D view will look like (18). The default location of the widget is the top right corner of the drawing area.



5. UCS Icon

UCS stands for a User-defined Coordinate System. The UCS icon shows the nature of the current coordinate system.

The display of the icon is controlled through the UCSICON system variable and the position of the UCS icon is defined by the UCSICONPOSITION system variable.

6. Model Tab

When you start a drawing session, your initial working area is called **Model Space**. **Model Space** is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or User Coordinate System (UCS). You view and work in model space while using the **Model** tab.

7. Layout Tab

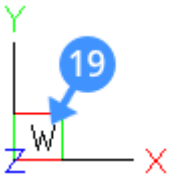
Each drawing has at least one layout and each layout is composed of one or more viewports. Each viewport can show a different part of the drawing at a different scale.

Paper space is a work environment that provides the model space view at a given scale, depending on the size of the paper.

In each layout, you can add the entities needed to complete a printed copy of the drawing; title blocks, legends, frames, etc. These entities are only visible in the layout when you add. They are not visible in the other layouts or in the model space.

UCS in Layout Mode

In paper space of layout modes, a W (19) letter appears in the UCS icon, which means the WCS is active.

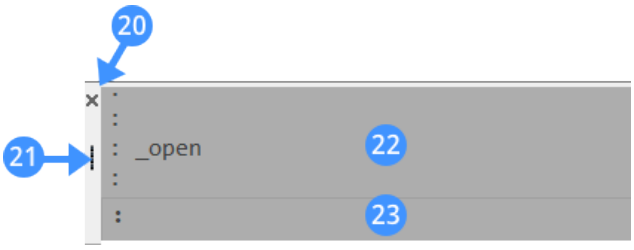


Nota: WCS stands for World Coordinate System.

8. Command line

The Command line consists of two fields.

- In the lower field, you can type the commands and BricsCAD shows prompt, options and other information regarding the execution of commands here. If the Command line is closed, this information shows in the Status Bar.
- The command history displays in the upper field of the Command line.



Close (20)	Click to close the Command line.
Grip (21)	Press and hold the left mouse button to drag the command bar.
Command history (22)	Shows the previously typed commands.
Command edit (23)	Type the commands and command options here.

9. Status Bar

The Status Bar sits along the bottom edge of the BricsCAD application window. It contains a lot of information about the settings in the current drawing. The status bar consists of 16 fields. All of these fields are optional, except for the Status field (24). If you click the small down arrow button at the right end of the status bar (25) a list of all field displays. Click a field to toggle its display.

Right-click a field to display an options menu for this field.

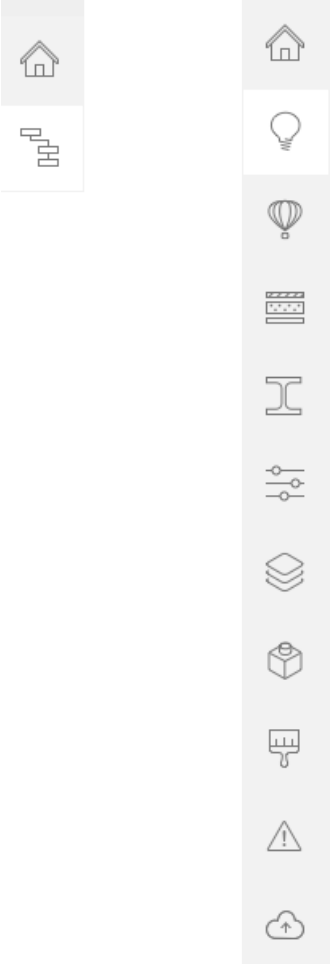
Status field (24)	Displays the status of the software.

List button (25)	Click the arrow to display a list of available Status Bar fields.
Fields (26)	Click a field to toggle it ON/OFF. Right-click to display a context menu.

10. Tool Panel

In BricsCAD BIM, the Tool Panel is displayed on both sides of the workspace. The Tool panel consists of BIM panels. The BIM panels are dockable and thus they are movable. The panels can be moved to another docked panel using the drag and drop method.

By default the Tool Panel looks like the image below: on the left-hand side it displays the **Structure Browser** and the **Project Browser** and on the right-hand side it displays **Tips**, **Components**, **BIM Compositions**, **BIM Profiles**, **Properties**, **Layers**, **Mechanical Browser**, **Materials**, **Report** and **Bricsys 24/7**.



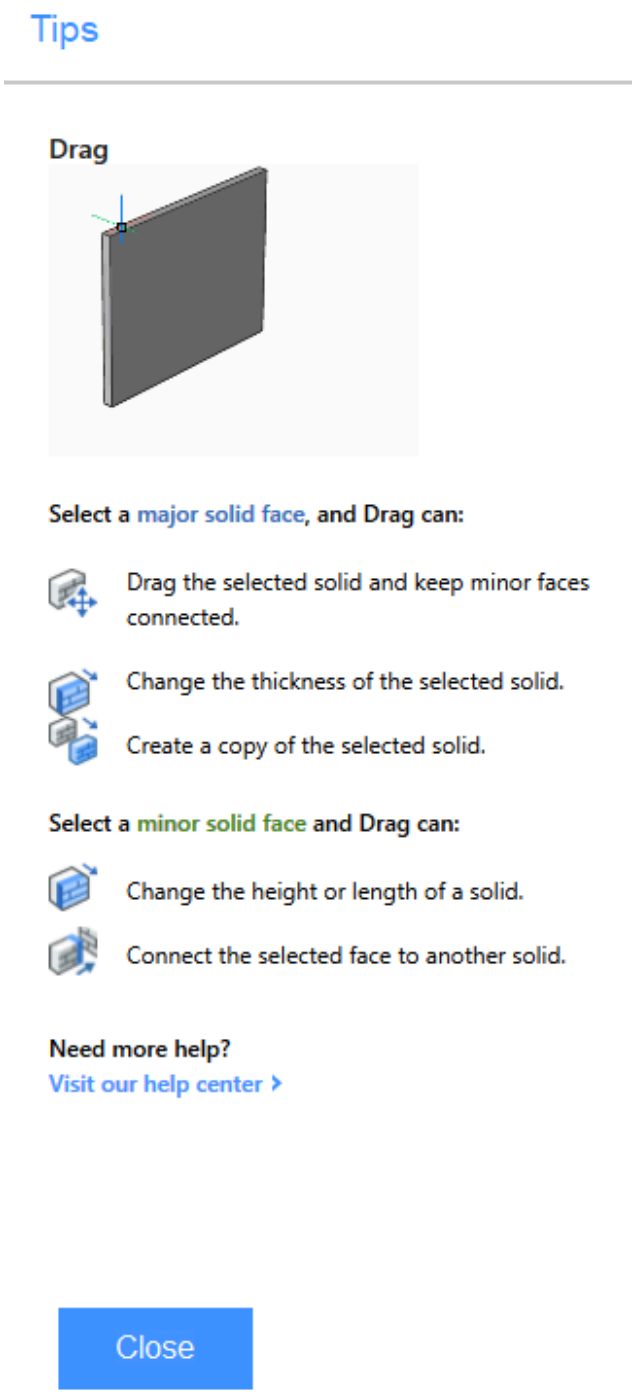
Nota: You can always add panels in the Tool Panel by right-clicking and checking those you want.

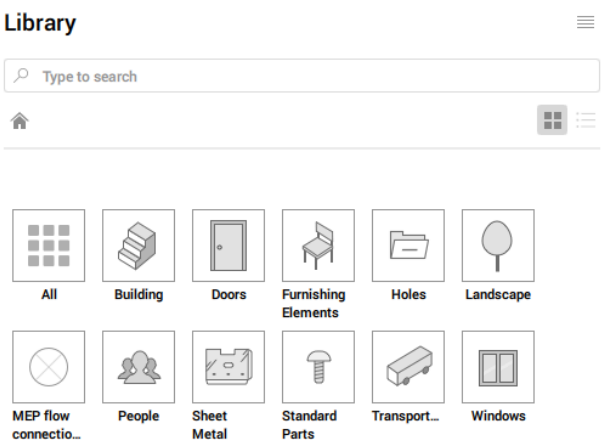
We will now have a look at the functions of the different dockable Panels from the Tool Panel:

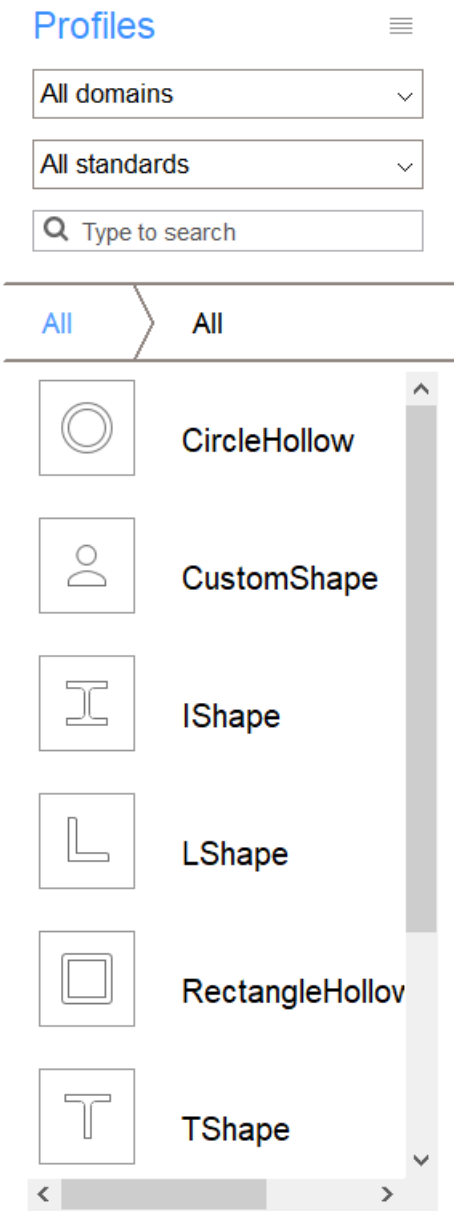
Project Browser	The BIM Project Browser allows you to easily navigate through the entire content of a project: models, sections, drawings, and schedules. See The **BIM Project Browser** article on how to use this panel.
Structure Browser	The **Structure Browser** is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the **Structure Browser** the BIM model can be organized in a way that you want to view the elements. Once you configured the tree, you can easily save this structured tree as a .cst file. By default, the .cst files are stored in the **Support Folder**. See The **Structure Browser** article on how to use this panel.

Tips

Tips panel gives general information about the tools and commands. The tips can change depending on the active command. The tips can only be seen when the panel is open.
The following illustration shows the **Tips** panel when the DRAG command is active:

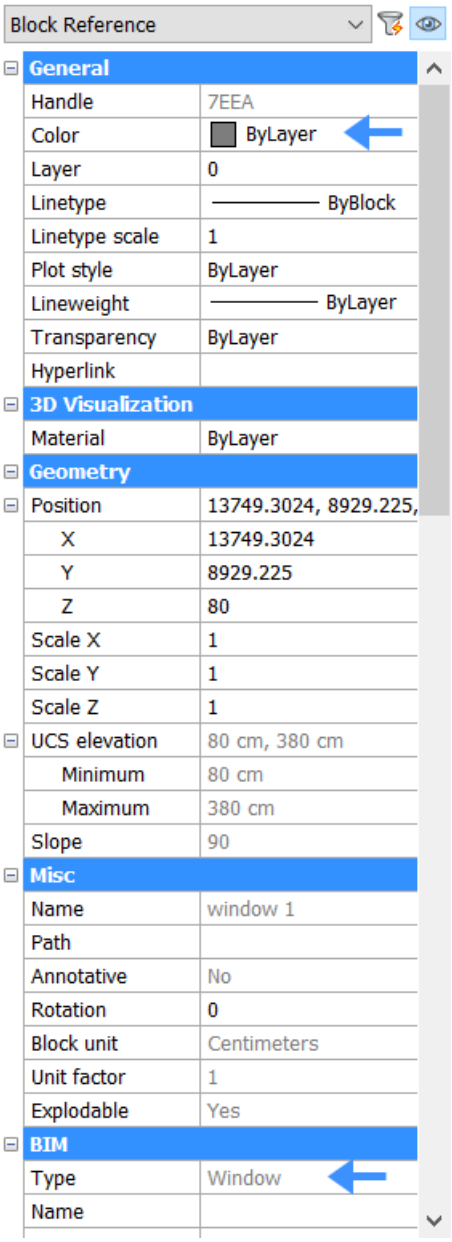


Library	The **Library** panel gives a list of components that you have by default. You can simply click and drag these components into your drawing.  At the bottom of the panel, you have the button . When you have made for example a drawing of a window, you can click on this button and add your own drawing to the component library. This way your window can be used as a component in all of your other drawings.
BIM Compositions	In the **BIM Compositions** panel, you find a library of predefined composition that you can assign to various solids of the BIM model by dragging them on. Assigning compositions to the elements in your model is a core step to increase the BIM model accuracy. Compositions can contain either multiple materials or a single ply. See The **BIM Compositions** article on how to use this panel in more detail

BIM Profiles	The **BIM Profiles** panel shows a library of predefined profiles that you can assign to linear solids by dragging them on.  See The **BIM Profiles** article to see how to work with this panel in more detail.

Properties

The **Properties** panel shows all the properties of a selected element. You always have some general info first, the other different tabs within this panel depend on the type of element that is selected.

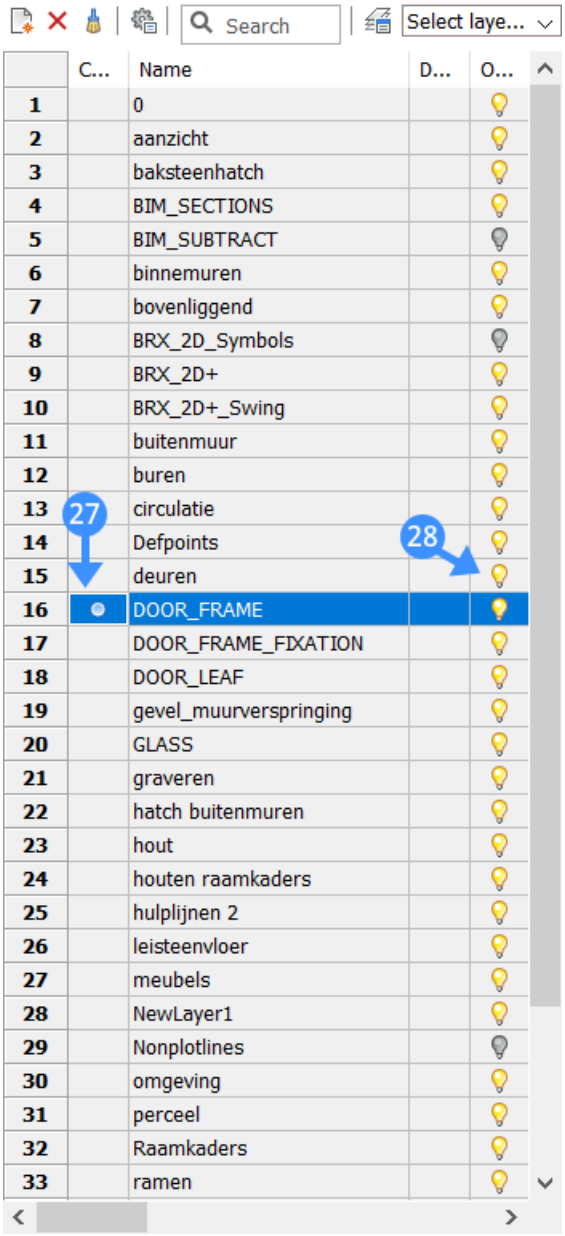


For example, here we can see that the selected item is a window and is colored by layer. You can adjust the properties as you like.

See the **Properties** article to see how to work with this panel in more detail.

Layers

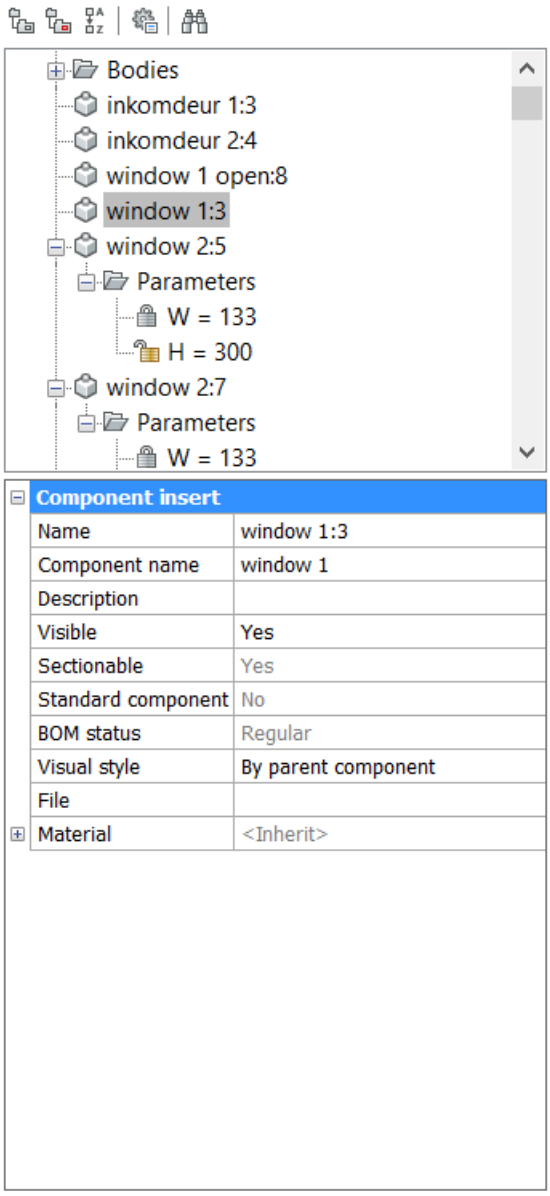
The **Layers** panel shows all the layers of your drawing.
It shows you the current layer (27), you can change it by clicking left of another layer. It also shows if a layer is ON or OFF by the state its lightbulb icon (28). Other layer properties such as color, line type,... can be found and adjusted by scrolling to the right.



In this example the current layer is **DOOR_FRAME** and it is turned ON.

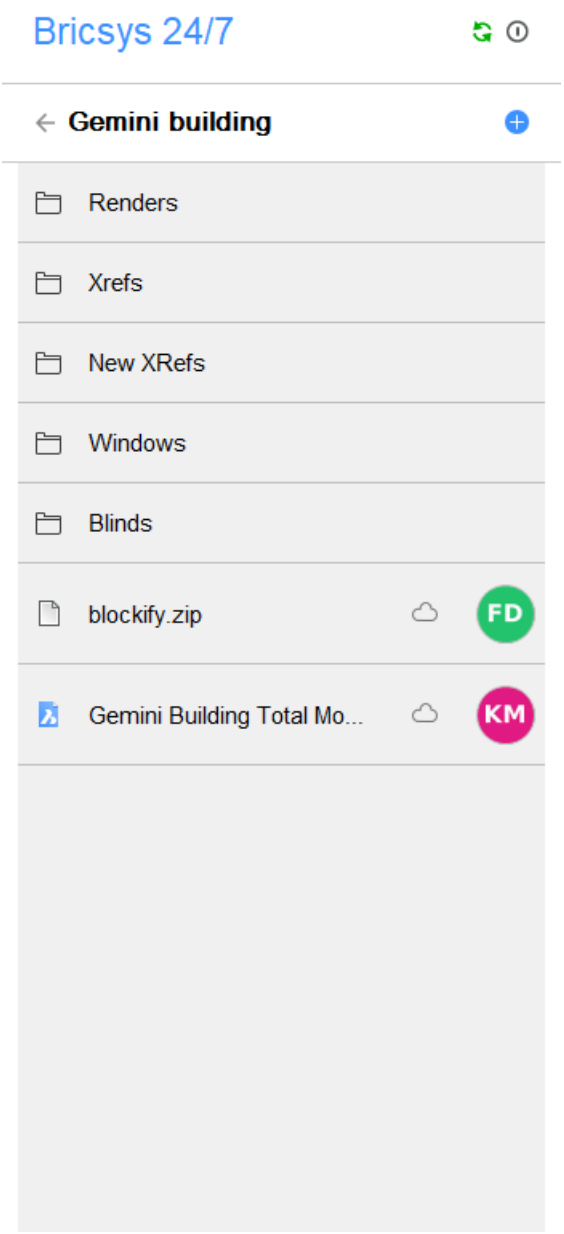
Mechanical Browser

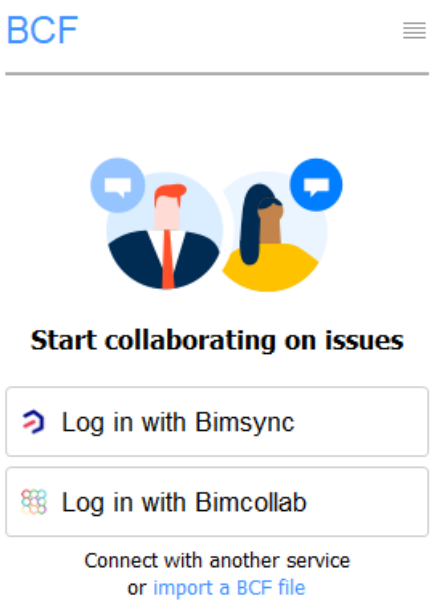
The **Mechanical Browser** panel displays the hierarchy of components, features, arrays, parameters, and constraints for the current drawing.
They are displayed in a tree-like form: you can expand and collapse every node to see its child nodes (for example, nested components).

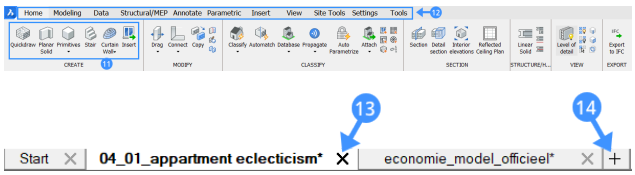


The **Mechanical Browser** allows you to edit properties of a particular node, displayed at the bottom of the panel and calls different tools depending on what type is available in the context menu by right click on a particular node. For example, in the image above you could add a description for the selected window.

Render materials	The **Render materials** Panel contains a library of predefined materials that you can assign to solids by dragging them on. You can also see a list of all the materials that you are currently using in your drawing. See the **Materials** article to see how to edit and create materials.
Report	Some commands use the **Report** panel to report the command output; e.g. all SHEETMETAL (Sm*) commands, DMAUDIT and DMSTITCH.

Bricsys 24/7	Login to your Bricsys 24/7 account here to collaborate on your project in the cloud. BricsCAD 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. When logged on, you can see an overview of your in the cloud Projects, Folders and Documents as shown in the image below:  See the **BricsCAD 24/7** panel article to see in detail how this panel works.

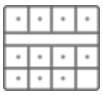
BIM BCF	Here you can import BCF (Bim Collaboration Format) files and connect to a BCF server. The BCF topics are listed in this panel.  The top part of the new **BCF** panel shows all the topics and can look similar to the **Library** panel. The lower part of the new **BCF** panel shows the currently active topic, with a bigger snapshot and the most common metadata: comment, author, status, etc. Double-clicking on an issue activates the topic.
Details	In the **Details** panel you can: • View saved details. • Organize saved details. • Propagate details. • Open details and adjust them in the model space.



20.2.3 Quad cursor

20.2.3.1 About

The **Quad cursor** menu, or **Quad** for short, is an alternative to the Command line or toolbars, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. The rollover tips are a limited set of properties displayed when selecting or hovering over an entity.



To toggle the **Quad** on/off, do one of the following:

- Click the **Quad** field in the Status bar.
- Press the F12 function key.
- Edit the QUADDISPLAY system variable in the **Settings** dialog.
- Type QUADDISPLAY in the Command line, then choose an option.

To toggle the **Rollover Tips** on/off, do one of the following:

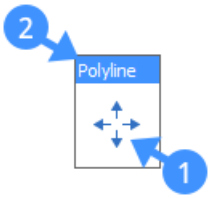
- Click the **RT** field in the Status bar.
- Type ROLLOVERTIPS in the Command line then set a value.

20.2.3.2 Quad layout

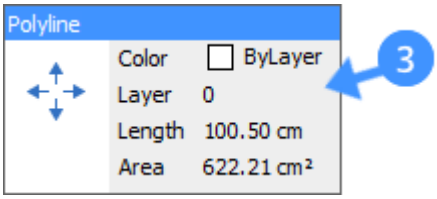
The **Quad** has a floating feature set that adjusts its content, depending on what you are or are not highlighting, and what you may have selected in the current workspace.

- Based on the highlighted entity type, the **Quad** displays the most recently used tool (1) with the entity name (2).

  Nota: When multiple entities are under the cursor, hit the TAB key to cycle through the different entities.

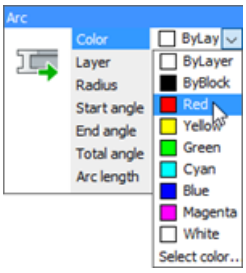


- When the **Rollover Tips** setting is on, a limited set of properties (3) of the entity under the cursor display in the **Quad**.



  The property values in the **Rollover Tips** can be edited.

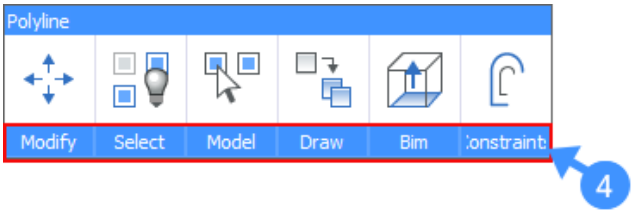
  The following figure shows the color of an arc being changed using the **Rollover Tips**.



Nota: To display the **Rollover Tips**, simply click the title bar of the **Quad** after expanding it.

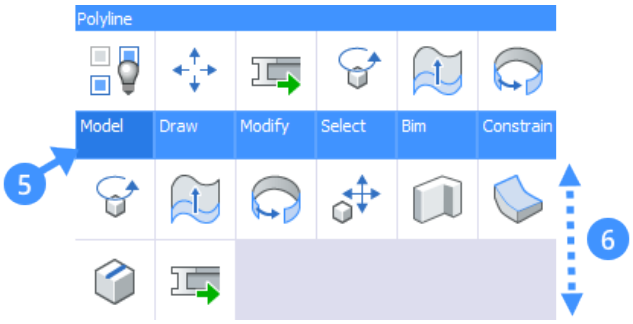
- When the cursor is moved on the **Quad**, a number of tool tabs display at the bottom (4).

  Nota: The number of tool tabs is controlled by the **Quadwidth** user preference (default = 6).

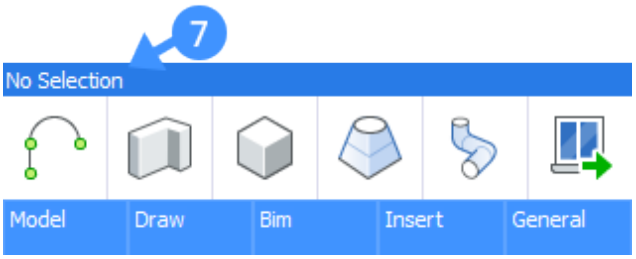


- When you move the cursor over the tool tabs title bar (5), the tool groups expand (6) under the cursor, which provides you with more tool options.

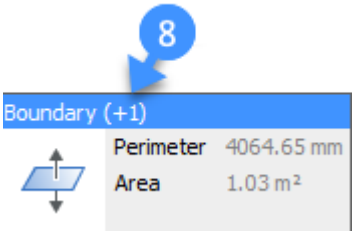
  Nota: The number of available tool groups and the number of tools within a tool group depend on the entity type under the cursor.



- The **Quad** can also be accessed in the no selection If the cursor is on an entity, hold down the right mouse button longer to display the **No Selection** Quad (7). Or when there is no selected/highlighted entity in the drawing, right-click to display the **No Selection** Quad.



- When an entity is selected and another one is highlighted, next to the entity type (title bar on the top) a plus one (8) appears in the parentheses. The count, regardless of the type, increases each time you select an entity. To reveal the selected the entity types which are hidden in the parentheses, expand the quad by moving your cursor on it.

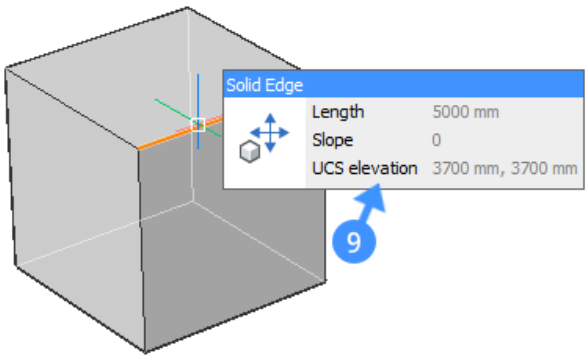


Nota: The **Quad** also adapts to the type of entities that have been selected. If you select two surfaces, you will be provided with different tool options in the Quad, then if you selected a solid and a surface.

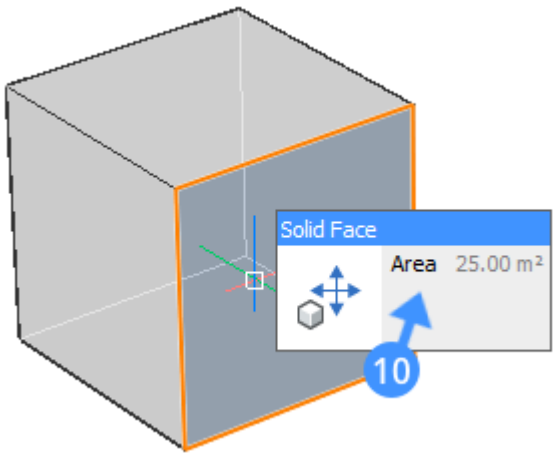
20.2.3.3 About: selecting entities

The **Quad** adapts to the type of entities that you have selected. If you select a surface, you will be provided with different tool options in the Quad, then selecting a solid or edge.

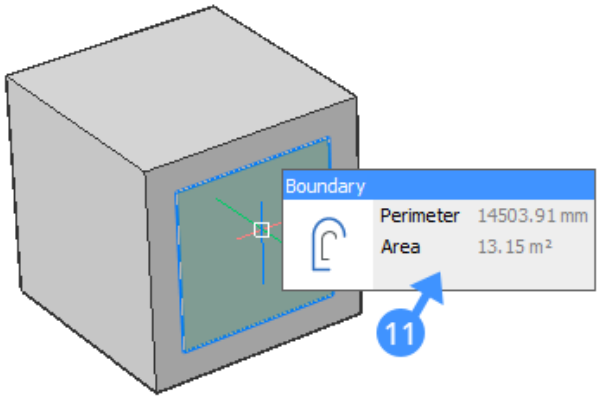
- Selecting an **edge**, the **Quad** appears showing the edge length, slope, and elevation (9).



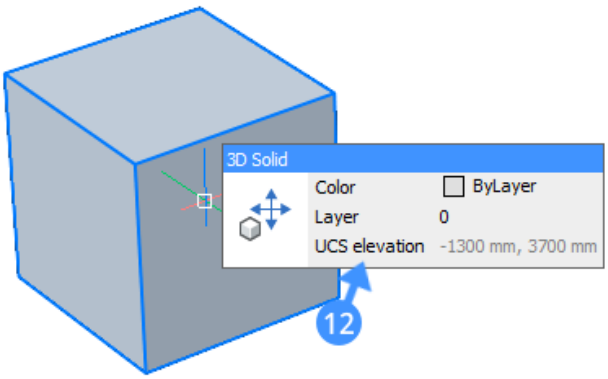
- Selecting a **solid face**, the surface area is shown in the **Quad** (10).



- Highlighting a **boundary**, the **Quad** displays the perimeter and area of this boundary (11).



- Highlighting a **solid**, the **Quad** displays the color, layer, and elevation of the solid (12).



20.2.3.4 Procedure: using the quad

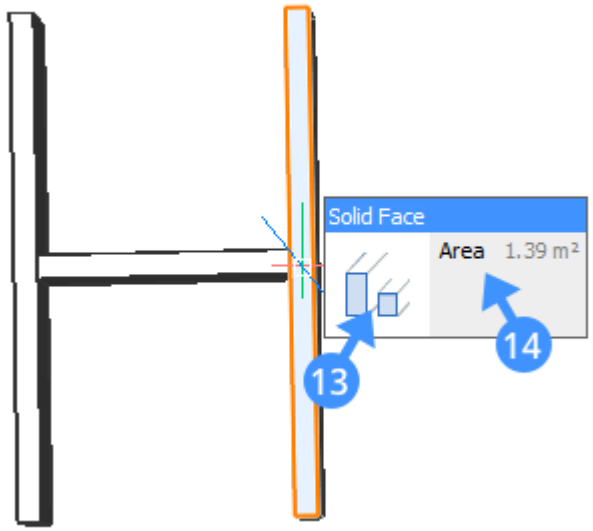
The following are general steps to use the **Quad** cursor.

Nota: The **Quad** and **Rollover tips** are both enabled.

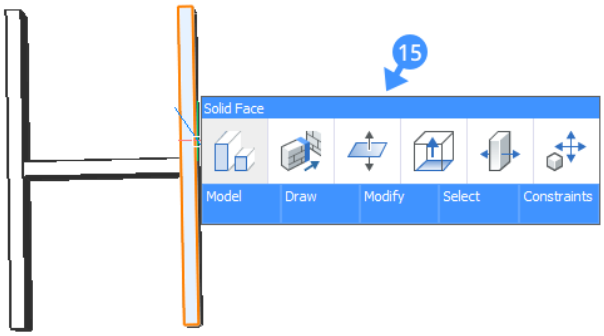
1 Hover the cursor over the entity you want to manipulate.

1 The entity under the cursor highlights.

The **Quad** displays a single icon (13), which is the most recently used tool with this entity type, and a limited set of properties (14) of the highlighted entity.



2 Right-click to launch the recently used tool. Or move the cursor to the single icon to expand the **Quad** (15), then select one of the available tools to manipulate the entity.



3 Move the cursor over the blue field to further expand the **Quad**.

All Quad tool groups for the current workspace display. When the first tool group is expanded, click an icon to use a tool or move the cursor to a different tool group.

20.2.4 Viewing entities

20.2.4.1 Commands

HIDEOBJECTS, ISOLATEOBJECTS, UNISOLATEOBJECTS

20.2.4.2 About

If you work with many entities in a drawing, everything that you have in the drawing is visible in the view. When you want to temporarily make only a few entities or a particular entity visible and manage them in a view, you can use the **Hide** or **Isolate** tools.

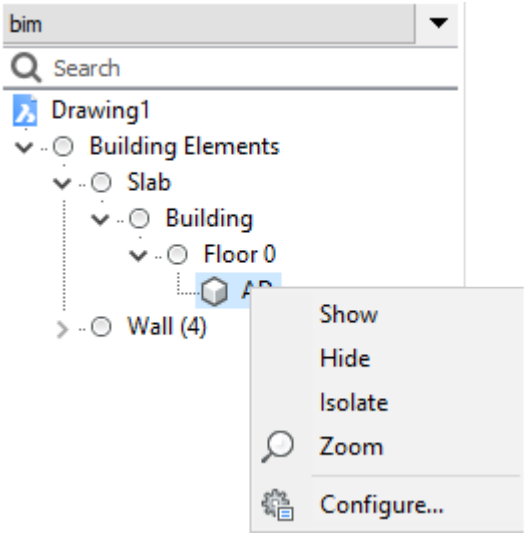
20.2.4.3 Hiding entities

The **Hide** tool temporarily hides the selected entities in the view.

For more information about this command, visit the Command Reference article HIDEOBJECTS.

(**Optional**) **Structure Browser** can also be used to hide and isolate selected entities as well as revealing all hidden entities in the model space. This is an easy way to quickly hide or isolate a certain category of objects or entities with a certain property value etc.

To do so, right-click on the entity in the **Structure Browser** and choose one of the options in the context menu (see below image).



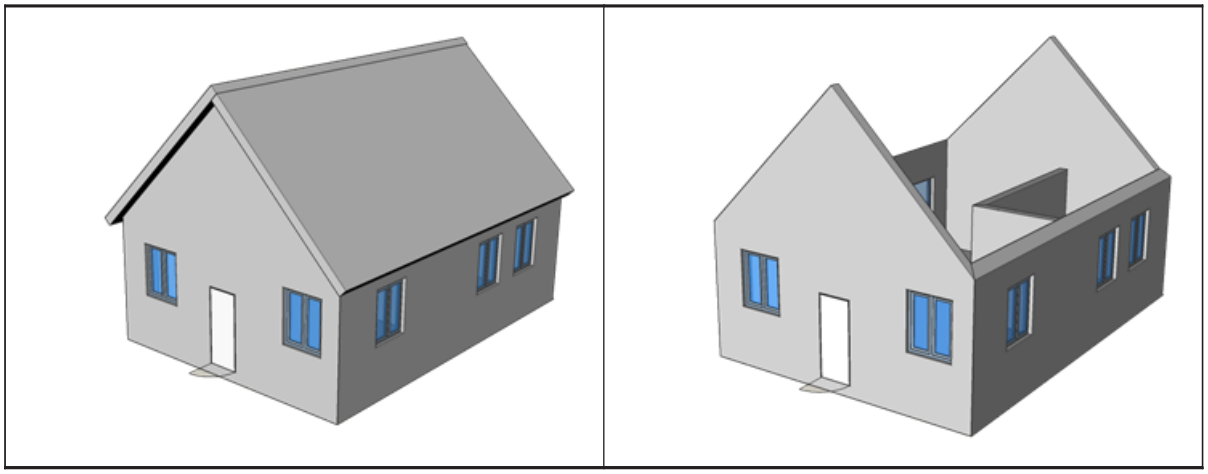
20.2.4.4 Procedure: hiding entities

1 Select one or more entities in the drawing area.

2 Select **Hide Entities** from the Quad.

All entities that have been selected are hidden in the view.

The following illustration shows an entire model on the left. In the second image, the roof was temporarily hidden.



20.2.4.5 Isolating entities

The **Isolate** tool temporarily hides all entities except those that have been selected.

For more information about this command, visit the Command Reference article ISOLATEOBJECTS.

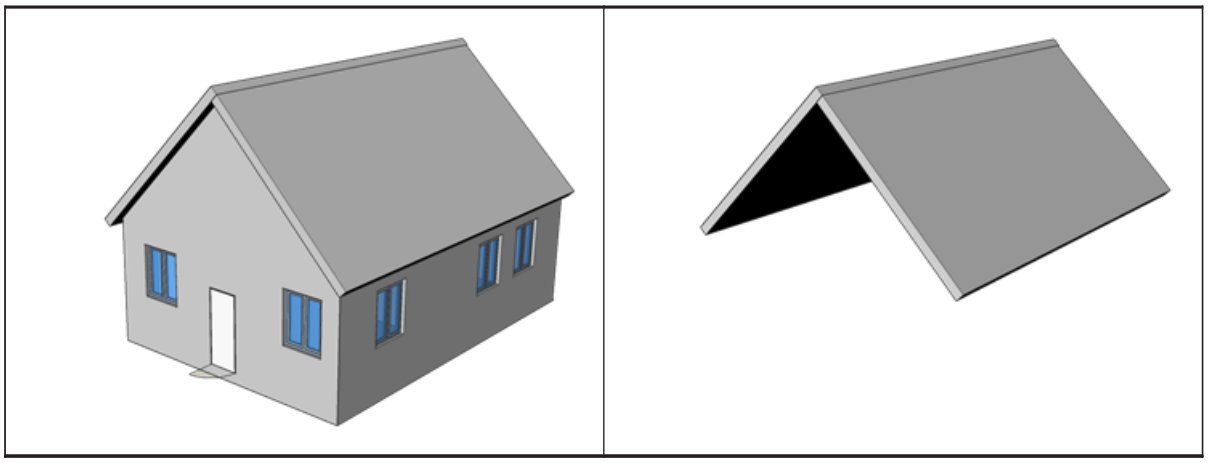
20.2.4.6 Procedure: isolating entities

1 Select one or more entities in the drawing area.

2 Select **Isolate Entities** from the Quad.

All entities that have been selected before are isolated in the view.

The following illustration shows the same building model from the below. In the second image, the roof was temporarily isolated.



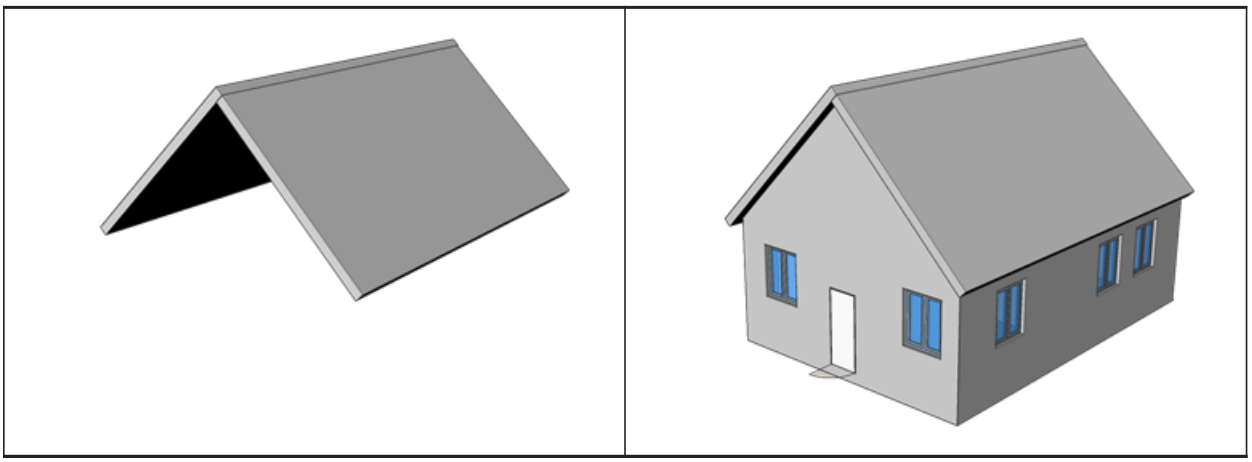
20.2.4.7 Showing entities

The **Show Entities** tool brings back all hidden entities in the view.

For more information about this command, visit the Command Reference article UNISOLATEOBJECTS.

If you have any hidden entities in your drawing, you can reveal them in the view using the show entities tool.

The following illustration shows the isolated roof elements from the above example. The second image shows the result after using the **Show Entities** tool.



20.2.5 Selecting entities

20.2.5.1 About

In BricsCAD, when you hover over an object, it will be highlighted. After highlighting an entity, you left click to select the highlighted entity, then it will be selected.

Highlighting: you only can highlight one entity at a time.

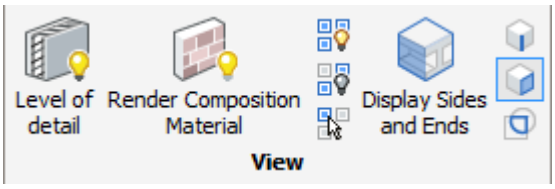
Selecting: you can select as many entities as you want.

Most of the modeling process can be done by highlighting the entity and using the Quad. For example, when you highlight a surface, the Quad will open, pick the command you want, without having to select the surface first.

20.2.5.2 Selection modes

The selection modes allow you to control which sub-entities (faces, edges, and boundaries) should highlight in selection preview, and can be selected.

Ribbon: **Home tab** > **View panel** > **Selection modes**



Nota: To display the **Ribbon**, right-click on a toolbar, then choose **Ribbon** in the context menu.

Settings: **Program options** > **Selection** > **Selection Preview** > **Selection Modes**

Selection modes	0x0007 (7)
1	☑ Select edges
2	☑ Select faces
4	☑ Select detected boundaries

Keyboard: SELECTIONMODES

In the following steps, each selection mode option is given with its figure and explanation.

- **Enable detection of 3D solid edges** ( ): controls whether 3D solid edges are highlighted by selection preview and can be selected. If you enable this option, you can select the edges of 3D solids by clicking it once.
- **Enable detection of 3D solid faces** ( ): controls whether 3D solid faces are highlighted by selection preview and can be selected. If you enable this option, you can select the faces of the 3D solids by clicking it once.
- **Enable boundary detection** ( ): controls whether closed boundaries in XY-plane of the current coordinate system or on the face of 3D solids are detected. If you enable this option, you can select closed boundaries from the faces of 3D solids by clicking it once.
- **Enable selection of Sides and Ends** ( ): controls the bimSides and BimEnds visibility of all linear elements.

Nota: BIM Sides/Ends are only available on classified linear elements with the attached library profile.

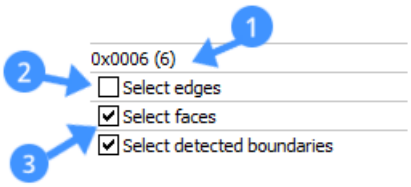
20.2.5.3 Changing selection modes setting

A sub-entity is a face, edge or boundary of a 3D solid. To control which sub-entities you select in your model do one of the following:

Nota: The default value of the SELECTIONMODES system variable is 6, which means the detection of 3D solid edge is disabled, while the detection of 3D solid faces and boundaries are enabled.

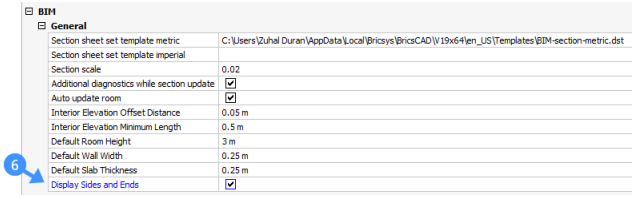
- Click the buttons on the **Settings** panel on the **Ribbon** and find **Selection Modes**.

  The following illustration shows the sum of the values (1) of the SELECTIONMODES system variable when the **Select edges** option is disabled (2) after clicking the checkbox once, the **Select faces** and **Selected detected boundaries** are enabled (3).



- Type SELECTIONMODES in the Command line and change the current parameter.
- Use the CTRL key on the keyboard to temporarily toggle between the different selection modes. These key functions enable you to select any type of entity.
- To display the BimSides/Ends for linear elements you can use DISPLAYSIDESANDENDS system variable in the **Settings** dialog box or set the DISPLAYSIDESANDENDS system variable to 1 in Command line.

  The following illustration shows the **BIM** > **General** system variable when the DISPLAYSIDESANDENDS is enabled (6) after clicking the checkbox once.

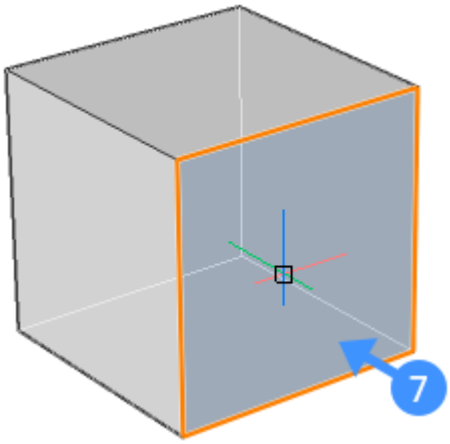


BIM	
General	
Section sheet set template metric	C:\Users\Zuhal Duran\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Templates\BIM-section-metric.dst
Section sheet set template imperial	
Section scale	0.02
Additional diagnostics while section update	☑
Auto update room	☑
Interior Elevation Offset Distance	0.05 m
Interior Elevation Minimum Length	0.5 m
Default Room Height	3 m
Default Wall Width	0.25 m
Default Slab Thickness	0.25 m
Display Sides and Ends	☑

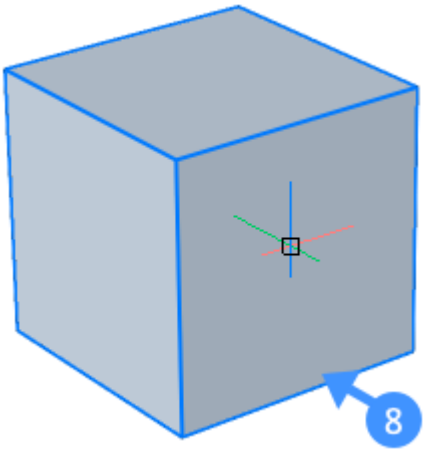
20.2.5.4 Selecting sub-entities

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default), do one of the following:

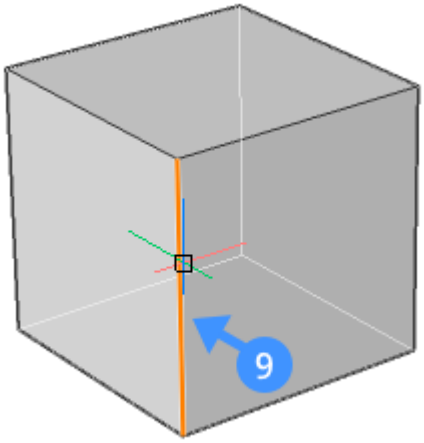
- To highlight a solid face, hover over the face with your mouse cursor. The highlighted face displays in orange (7). To select the face, left-click it when the face is highlighted. To select multiple faces of the solid, left-click them one by one.



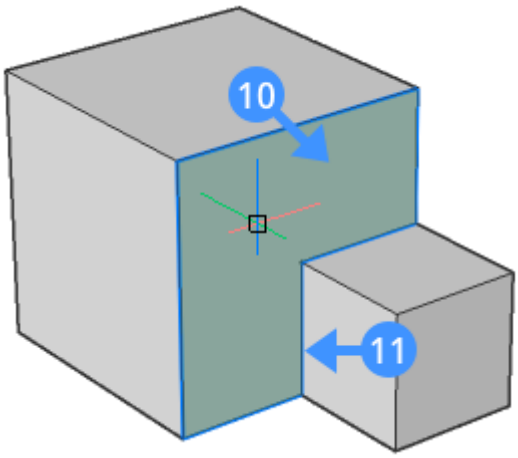
- To highlight a solid, hover over one of its faces while holding down the CTRL-key. The solid displays in blue (8). To select the solid, left-click it while the solid is highlighted. To select multiple solids left-click them one by one.



- To highlight an edge, hover over the edge of the solid while holding down the CTRL-key. The edge displays in orange (9). To select the edge, left-click it while the edge is highlighted. To select multiple edges left-click them one by one.



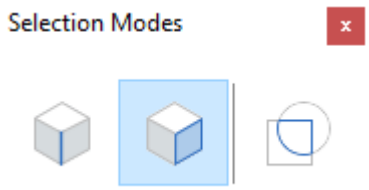
- To highlight a boundary, hover over the detected boundary on the face of the 3D solid.
- When the detected boundary is highlighted, it displays with a color. By default, inside of the boundary displays in green (10) and the outer edge of the boundary displays in blue (11).



Nota: This setting can be changed by typing BOUNDARYCOLOR on the Command line (default value = 95).

20.2.5.5 Procedure: Modify the thickness of a solid

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the face of a solid

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

5 The selected face of the solid is dragged

6 The dynamic dimension field displays the relative displacement of the face (= with respect to its original position).

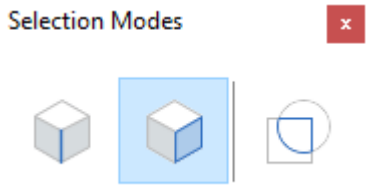
7 (**Optionally**) Press Tab to change the face selected as the reference face for the dynamic dimension.

Nota: When a composition with locked thicknesses (non-variable thickness) is applied to a solid and if the thickness of the solid is not equal to the total amount of the composition thickness, the solid displays with red hatches. Execute the BIMUPDATETHICKNESS command to re-apply the composition to such solid.

20.2.5.6 Procedure: Modify the height of a wall

Modify the dimensions of a single wall

1 Make sure the **Select faces:** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the top face of the wall.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose **Push/Pull** in the Model command group of the Quad.

The selected face moves dynamically with the cursor.

The dynamic dimension fields display the incremental height.

Do one of the following:

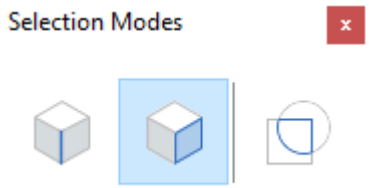
5 Type a value in the dynamic dimension field and press Enter.

6 Press Tab to use another reference face, e.g. the bottom face of the solid, then type a value in the dynamic dimension field.

Modify the height of multiple walls

Use the SELECTALIGNEDFACES command (**Quad** > **Select**:) it is easy to select the top face of all solids that lie in the same horizontal plane.

1 Make sure the **Select faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over the top face of one of the walls.

3 If necessary, press Tab to highlight an obscured face.

4 When the face highlights, choose SELECTALIGNEDFACES in the **Solid/Face Selection** command group in the Quad.

All faces lying in the plane of the selected face are selected.

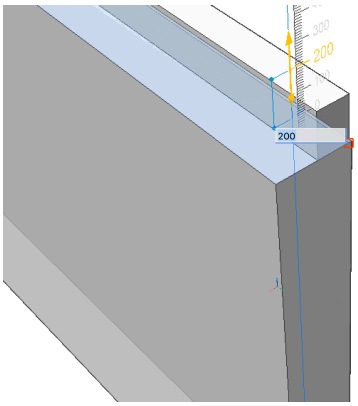
5 Now follow the same steps as for a single wall.

Modify the height ply-by-ply

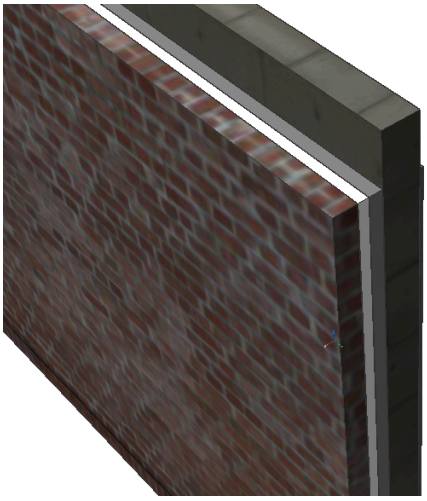
When the **Display Composition** property of a solid is set (LEVELOFDETAIL system variable =2 and, optional, RENDERCOMPOSITIONMATERIAL system variable =1) and the **Select faces** option of the SELECTIONMODES system variable is selected, the height or length of the composition plies can be modified separately.

1 Move the cursor over the ply you want to edit and choose **Push/Pull** in the **Model** command group of the Quad when the ply highlights.

2 Type value in the dynamic distance field.



3 Press Enter to accept the distance.



20.2.5.7 Selecting obscured sub-entities

In your drawing, some entities or sub-entities (faces, edges, and boundaries) might be hidden behind other entities. The Tab key enables you to cycle through sub-entities.

The following steps show how to highlight a hidden face that lies beneath the cursor.

When **Select edges** is off, **Select faces** is on, **Boundary Detection** is on (default).

1 Place your cursor on one of the solid faces.

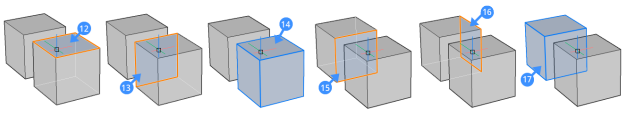
1 In the example below, the top face of the front solid is highlighted (12).

2 Press Tab key to cycle through the hidden entity.

3 Now, the back face of the front solid is highlighted (13).

4 Repeatedly pressing the Tab key, while keeping the cursor at the same position, allows you to highlight the entire solid in the front (14) and the front face (15) and right face (16) of the rear solid. Finally, the entire rear solid is highlighted (17).

5 When the entity or sub-entity that you want to select is highlighted, left-click it to select.



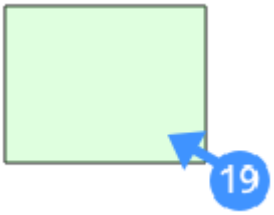
20.2.5.8 Selecting multiple entities using the selection boxes

The selection boxes allow you to select one or more than one entity at a time. To use a selection box, left-click on an empty space in the drawing area, and move the cursor until the selection box covers the entities that you want to select.

There are two types of selection boxes in BricsCAD;

Blue selection box (18) appears when you create a box from left to right. The blue selection box will only select entities that are entirely covered by this selection box.

Green selection box (19) appears when you create a box from right to left. The green selection box will select all elements that are entirely covered by the selection box and all the elements that intersect with one of the edges of the box.



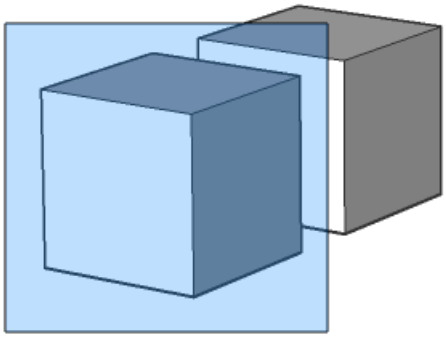
The following steps show how to select multiple entities using the blue selection box:

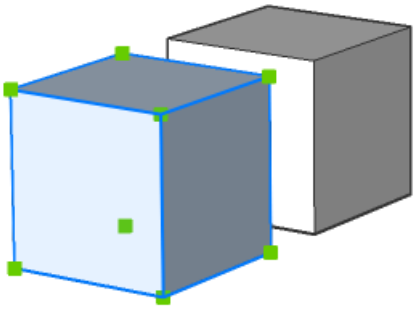
Nota: By default, the selection box only selects entire entities (e.g. solids, lines, polylines, blocks, …) but not sub-entities (e.g. solid faces, solid edges).

1 Click and move the mouse to the right to create a blue box around the entities you want to select. When the entity is completely inside the window, it will be added to the selection set.

1 The example below shows the blue selection box usage when the selection mode is set to **Solids**

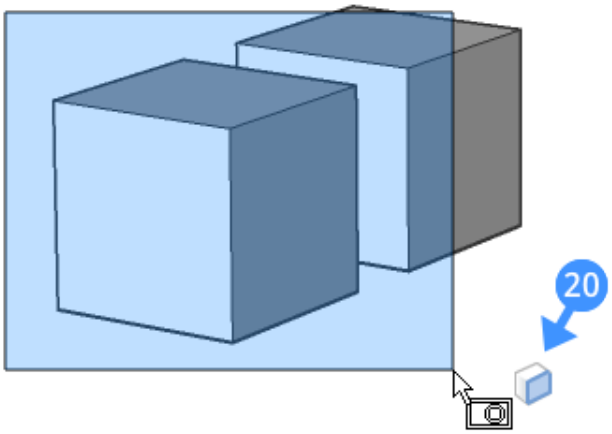
(default).

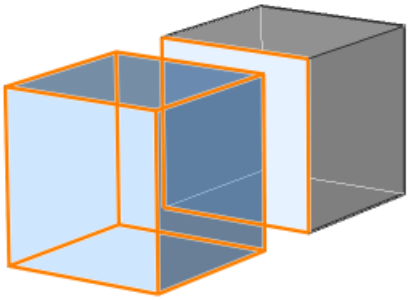




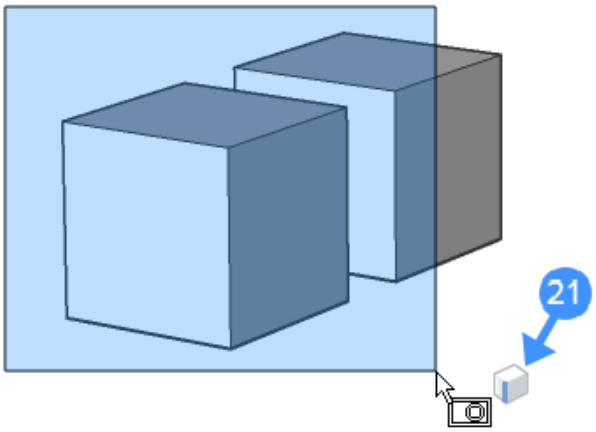
The blue selection box covers the left cube completely and thus only this cube is selected in the model.

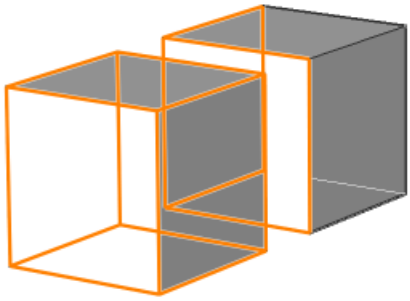
2 To select faces of 3D solids and surfaces, press CTRL key once during window selection. The icon (20) next to the cursor indicates the selection mode. The faces that are completely inside the window will be selected when the selection mode is set to **Faces**.





3 To select edges of 3D solids and surfaces, press the CTRL-key twice during window selection. The 'Edges' icon (21) appears next to the cursor. The edges that are completely inside the window will be selected when the selection mode is set to **Edges**.

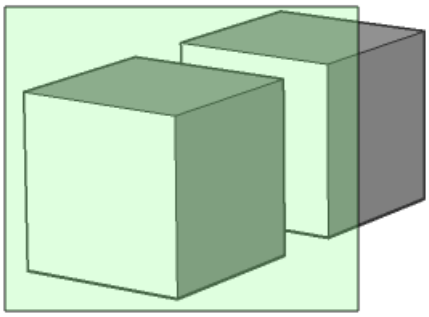


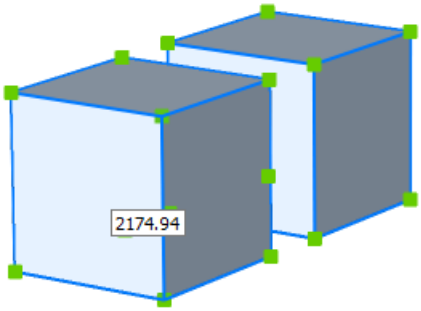


The following steps show how to select multiple entities using the green selection box:

4 Click and move the mouse to the left (crossing window) to create a green box around the entities you want to select. When the entities overlap the window or are completely inside the window, they will be added to the selection set.

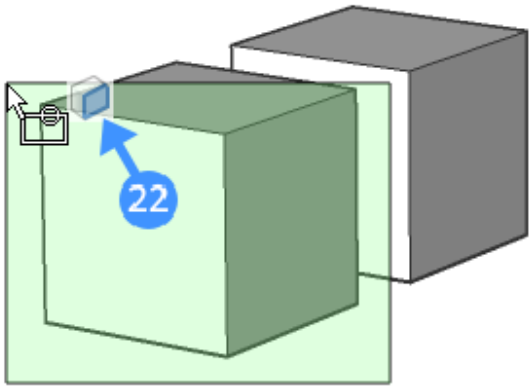
5 The example below shows the green selection box usage when the selection mode is set to default.

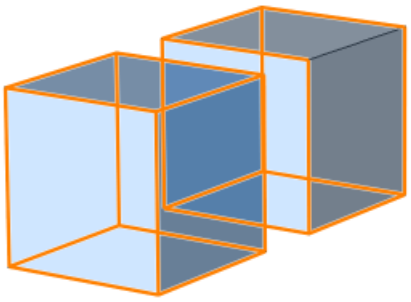




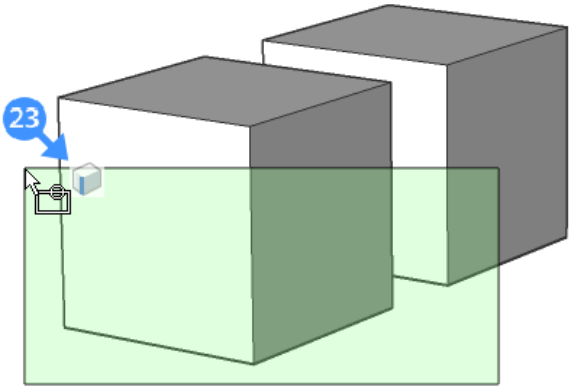
The green selection box selects both cubes in the model as the selection box overlaps with the two cubes.

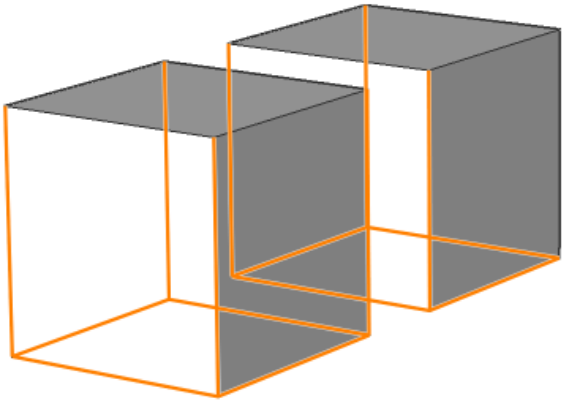
6 To select faces of 3D solids and surfaces, making use of the green selection box, press the CTRL key once during window selection. The 'Faces' icon appears next to the cursor (22). Any faces that are completely or partially inside the crossing window will be selected.





7 To select edges of 3D solids and surfaces, making use of the green selection box, press the CTRL key twice during window selection. The 'Edges' icon appears next to the cursor (23). Any edges that are completely or partially inside the crossing window will be selected.

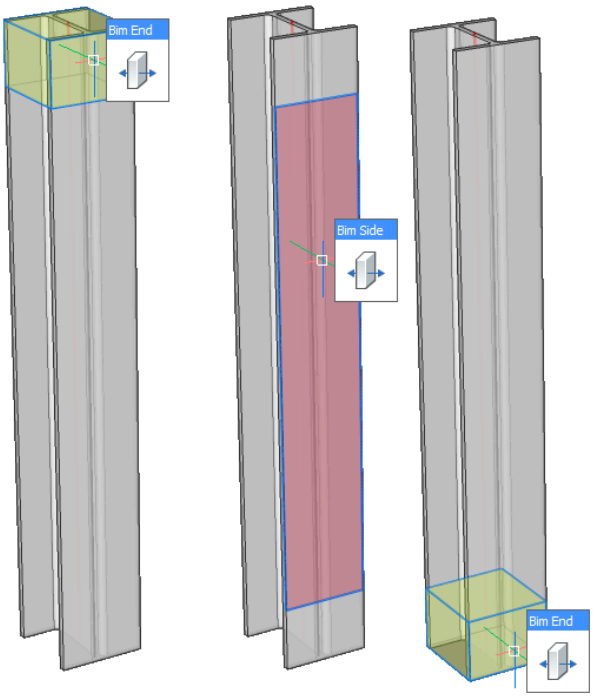




20.2.5.9 Selecting/Highlighting BIM sides and ends on profiles

1 Click on **Enable selection of Sides and Ends** icon in the BIM Toolbar (see below image).

2 Hover over the linear solid in the model space when the cursor is on the top or bottom side of linear solid, **Bim End** face displays in yellow whereas the side face of linear solid displays in red.



20.2.6 Level of detail

The RENDERCOMPOSITIONMATERIAL and LEVELOFDETAIL system variables control the display of materials and multiply compositions.

20.2.6.1 Level of detail

Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
[0] Low	Multiply compositions do not display.
[2] High	The individual plies of the multiply compositions are visible and editable.

20.2.6.2 Render composition materials

Quad: Bim

Ribbon: **Home** | **View**

Toolbar: **BIM**

Click the Icon to toggle the setting.

Option	Description
[0] Off	Materials are not rendered.

Option	Description
[1] On	Materials are rendered. When Level Of Detail = 0 (Low), major faces of solids, that have a multiply composition attached, are rendered in the material of the underlying ply. Minor faces are rendered in the material of the exterior ply.

20.3 Tecniche di modellazione

20.3.1 BIM linear solid

20.3.1.1 Commands

BIMLINEARSOLID, BIMAPPLYPROFILE, BIMPROFILES, BIMADDECCENTRICITY, BIMRECALCULATE

20.3.1.2 About BIM linear solid

The Linear Solid tool allows you to create structural members or flow segments. The tool can be made use of creating a variety of structural member profiles or MEP elements.

For more information about this command, visit the Command Reference article BIMLINEARSOLID.

For more information about **MEP modeling**, visit the help article **MEP Modeling** and **MEP Flow Connection Points**.

For more information about **Structural Steel**, visit the help article **Structural Steel**.

Before starting the process, make sure that Quad is active. Next, follow these steps:

1 Hover over the **Model** tab in the Quad and select the Linear Solid tool.

Nota: You can also launch the **BimLinearSolid** command by entering it in the command line.

You are prompted: Set start point or [Follow/Rectangular/Circular/Library/select in Model/adJust profile] <Last point>:

2 Select a start point.

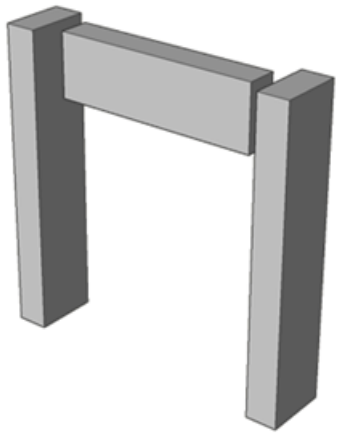
3 You are prompted: Set next point or [Angle/Length/Undo/Quarter turn/Rotate/choose other Profile/adJust profile]:

4 Move the mouse cursor to draw the beam or column.

5 Click to specify the endpoint or enter a value for the distance using the dynamic dimension field.

6 Press Enter key to accept it.

7 The following illustration shows a beam and two columns with the default rectangular profile.

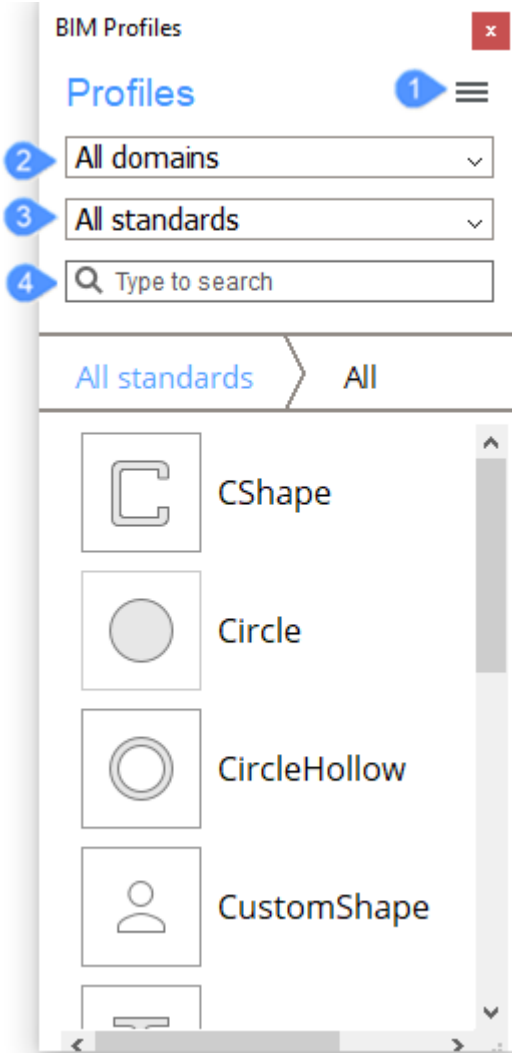


20.3.1.3 About BIM profiles

You can access the **BIM profiles** panel by clicking the I-beam on the right-hand side of your screen. If the **BIM Profiles** toolbar is not displayed on the right-hand side of your screen, right-click on the **Tool** panel at the right-hand side of your screen, choose **Panels** in the context menu, then check **BIM Profiles**.

Nota: You can also find the **BIM profiles** panel by typing BIMPROFILES in the Command line.

The **BIM Profiles** panel will be displayed:



The drop-down box at the top (2) acts as a filter. There are different domains on which you can filter: **Generic**, **Structural Concrete**, **Structural Steel**, **HVAC**, **Piping** and **Electrical**.

Nota: If you don't have any profiles in a certain domain, you will not be able to filter on it.

The second drop-down boxes (3) allow you to filter the profiles by the country standards.

The search box (4) allows you to filter by name.

The menu (1) on the right-hand side allows you to access the **BIM Profiles** dialog box. Clicking the menu icon displays a context menu with a couple of options.

Select **Open profile dialog** on the menu to open the **Profiles** dialog box.

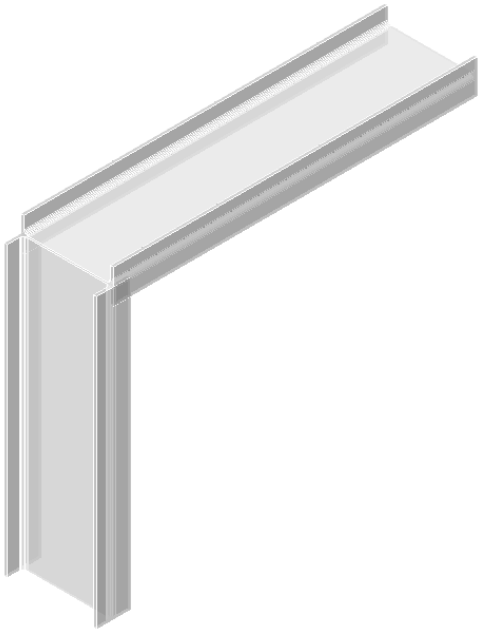
20.3.1.4 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the **BIM Profiles** panel to the model space.

Nota: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



Nota: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

20.3.1.5 About BIMAPPLYPROFILE

The **Apply Profile** tool allows you to change the profile of linear entities and linear solids. Linear solids have a constant cross-section over a linear extrusion path. The tool recognizes clippings and openings. Using the **Apply Profile** tool, you can create columns, railings, beams, pipes, ducts and more.

For more information about this command, visit the Command Reference article BIMAPPLYPROFILE.

How to apply profiles to an existing line or solid: with the BIMAPPLYPROFILE command

1 Select the line or solid you want to apply a profile to and launch the BIMAPPLYPROFILE command.

2 Press Enter and the **Profiles** dialog box, appears.

3 Choose a profile and press the **Select** button in the bottom-right corner.

4 Press Enter to end the command. The profile is applied to the line or solid in your model.

How to apply profiles to an existing line or solid: with the BIM Profiles panel

1 Open the **BIM Profiles** panel.

Nota: You can access the **BIM Profiles** panel by clicking the I-beam icon on the right-hand side of your screen.

2 Select all the entities you want to apply a profile to.

3 In the **BIM Profiles** panel, search for the profile you want apply to the selected entities.

4 Drag the chosen profile onto one of the selected entities.

5 The profile is applied to the line or solids in your model.

Nota: By applying a profile to a solid, the solid will be replaced by the chosen profile. In case of a line, the line will not be replaced, but a profile was added with his axis the same as the selected line.

6 Since the line is no longer needed, you can delete the line.

20.3.1.6 About BIMADDECCENTRICITY

BIMADDECCENTRICITY command allows you to control the relative position of the axis of a linear solid.

For more information about this command, visit the Command Reference article BIMADDECCENTRICITY.

1 Select an existing linear solid or profile in the drawing area.

2 Launch the BIMADDECCENTRICITY command by entering it in the Command line or select the command in the Quad (under the model tab) or in the Ribbon (in the **Structural/MEP** tab).

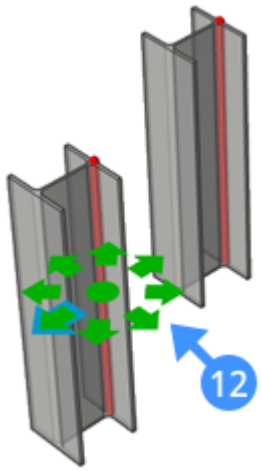
The eccentricity widget appears.

3 To set an eccentricity, click one of the nine locations from the widget, then right-click to confirm.

4 Optionally, type a location in the Command line.

5 The options of the locations are: **TR**: Top Right, **TM**: Top Middle, **TL**: Top Left, **MR**: Middle Right, **C**: Center (resets the axes of the selection set to the default location), **ML**: Middle Left, **BR**: Bottom Right, **BM**: Bottom Middle, **BL**: Bottom left .

6 Press Enter to accept changes. The following illustration shows the Eccentricity widget (12) on the linear solid profile.



20.3.1.7 About BIM recalculate axis

The Recalculate Axis tool recalculates the axis of linear building elements and repositions it back to the centerline of a linear element.

For more information about this command, visit the Command Reference article BIMRECALCULATEAXIS.

Recalculate the axis of linear building elements

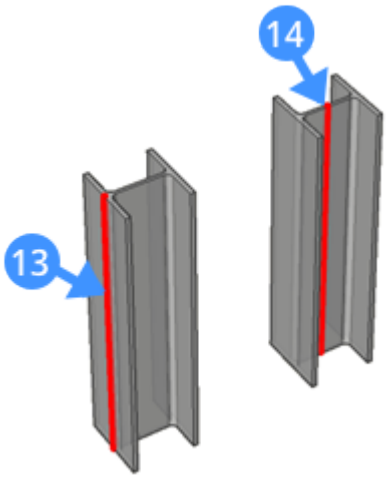
1 Select any linear building element in your model.

2 Launch the BIMRECALCULATEAXIS command.

3 Press Enter.

4 The axis of the element is moved to the default position.

5 The following illustration shows the result of using BIMRECALCULATEAXIS on the linear solid. The axis (13) is moved to the centerline (14) of the linear element.



20.3.1.8 Helpful settings when modeling Linear Solids

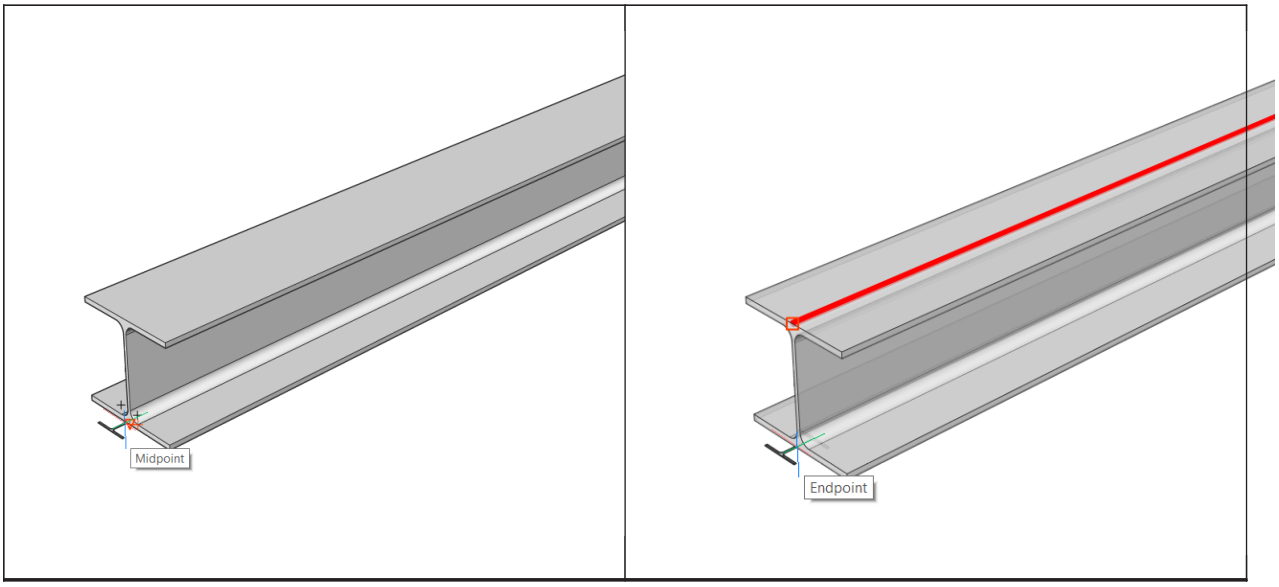
Display Axes

The DISPLAYAXES variable system can hide or view the Axes of linear solids in order to edit the structure more easily.

If DISPLAYAXES is toggled on, the axes of the linear solids are visible as a thick red line. In this case, you will only be able to snap the endpoint of the axis as a starting point to add more linear solids or profiles to your already existing structure.

When DISPLAYAXES is toggled off, no axis is visible in your drawing area and you're able to snap all specific points you have ticked in the context menu of ESNAP.

DISPLAYAXES OFF	DISPLAYAXES ON



BIMOSMODE

The BIMOSMODE variable system gives you the possibility to choose the level of detail you are snapping to.

1 Enter BIMOSMODE in the Command line or in Settings.

1 You are prompted: New current value for BIMOSMODE (0 to 3) <2>:

2 Choose a new value (type a number from 0 to 3).

Value	Snapping detail
0	Turns on all snapping.
1	Turns off all snapping to solids except for the axis of the solid.
2	Turns off all snapping to BIM Grid axes except for axis intersections.
3	Turns off all snapping to BIM Grid axes except for axis intersections. Turns off all snapping to solids except for the axis of the solid.

3 Press Enter

20.3.2 BIM window create

20.3.2.1 Commands

BIMWINDOWCREATE

20.3.2.2 About

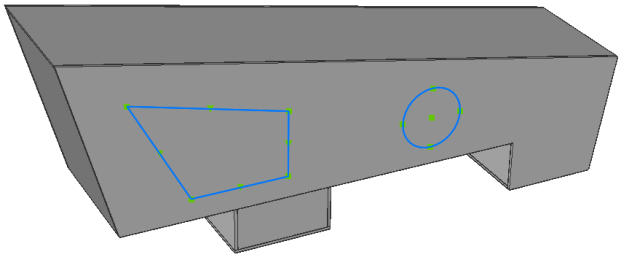
In BricsCAD, the **Window Creator** tool allows you to turn a different set of design ideas into a window. You can use the tool on windows of all shapes and sizes, which enables you to utilize a significant amount of window styles in your BIM project.

When you have a closed 2D entity or boundary on the face of a solid (wall), the window creator tool enables you to use these closed entities to make your window. Using the window creator tool, you can select any window styles in the dialog box and change the properties of this created window in a BIM model.

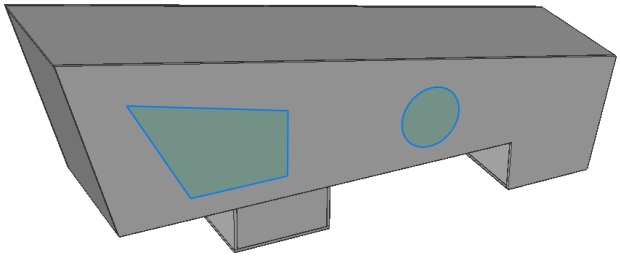
20.3.2.3 Creating a parametric window using boundaries

1 Draw a polygon on the face of a solid (wall).

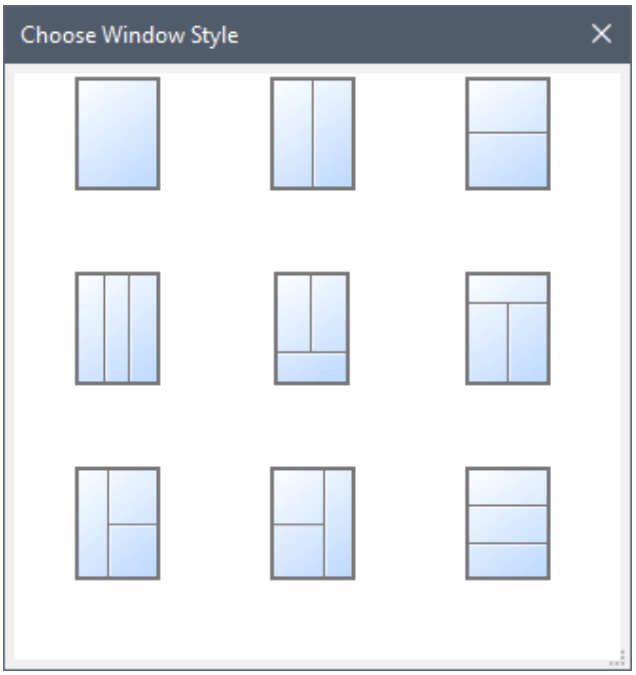
1 Based on this base profile, the window opening will be created.



2 Make sure **Boundary Detection** (select detected boundaries) is enabled into SELECTIONMODES system variable. Hover over the inside of the polygon and select the **Create Window** tool from the Quad.

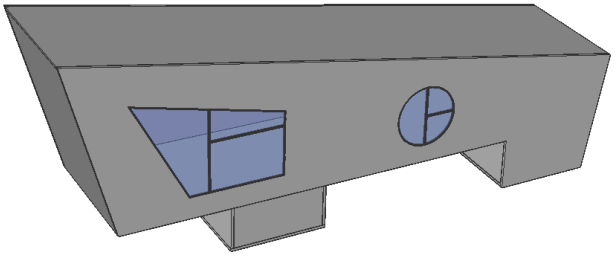


A **Choose Window Style** dialog box displays:

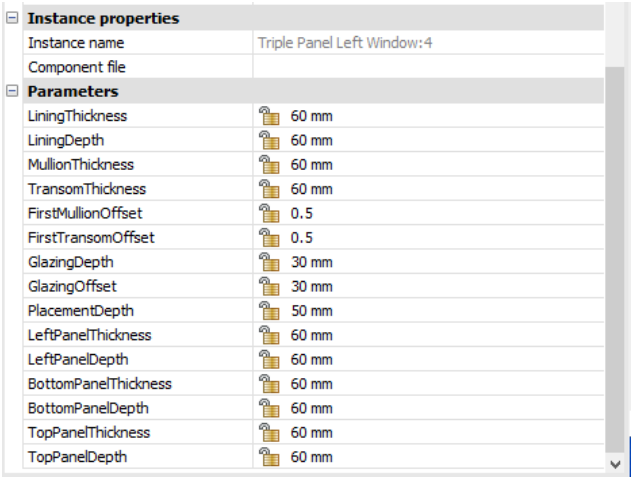


3 Select a window style in the dialog box.

4 The window is created on the wall with the selected style. The following illustration shows the triple panel left window style which was selected to create the windows on the face of the wall.



5 (**Optional**) You can refine the window design by changing the default values of the parameters. To do this, select the window and change the desired parameters in the **Properties** panel.



Instance properties	
Instance name	Triple Panel Left Window:4
Component file	
Parameters	
LiningThickness	60 mm
LiningDepth	60 mm
MullionThickness	60 mm
TransomThickness	60 mm
FirstMullionOffset	0.5
FirstTransomOffset	0.5
GlazingDepth	30 mm
GlazingOffset	30 mm
PlacementDepth	50 mm
LeftPanelThickness	60 mm
LeftPanelDepth	60 mm
BottomPanelThickness	60 mm
BottomPanelDepth	60 mm
TopPanelThickness	60 mm
TopPanelDepth	60 mm

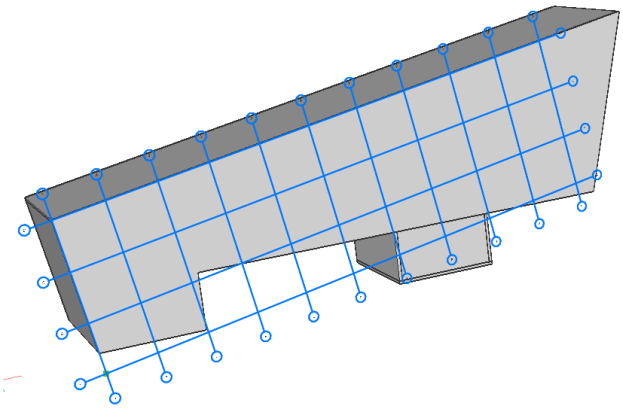
To change one of the default parameters of this window, type a new value in the property field.

Press Enter to accept the new value of the parameter. The window should be updated automatically.

20.3.2.4 Create windows based on a grid

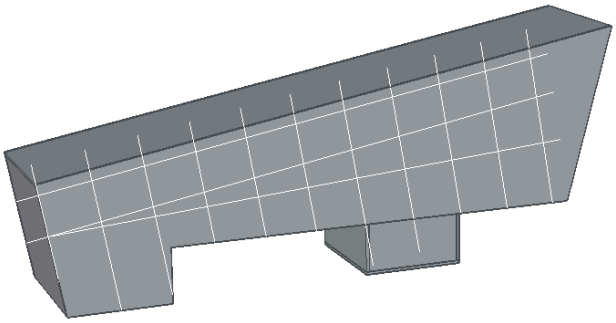
1 Use BIMGRID command to create a Grid on the face of the wall. Choose the basic dimensions in the Command line. Changing the **U-line** and **V-line** panel options allows you to specify the desired size and number of panels on the surface.

Nota: To draw on the face, make sure you first highlight the face and then click. Otherwise, the Grid will be created in the XY-plane.

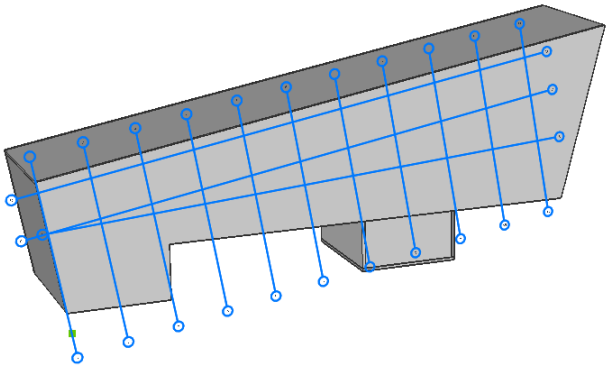


2 Use BEDIT or REFEDIT to edit the Grid.

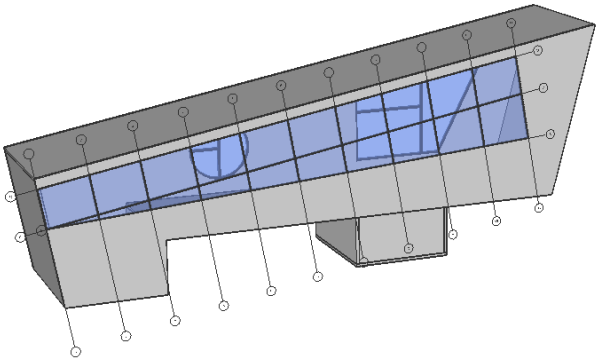
3 Use the Manipulator to change the position of the gridlines on the wall.



4 Select the Grid and select **Create Window** in the Quad.



5 The window is generated based on the gridlines.



Nota: You can edit the parameters in the **Properties** panel. Type a new value in the property field and press Enter. The window will automatically update itself.

Parameters	
FrameDepth	60 mm
GlazingDepth	30 mm
GlazingOffset	30 mm
PlacementDepth	50 mm

20.3.2.5 Creating window openings

BIM Opening Create creates an opening without window geometry. It uses the same command as BIMWINDOWCREATE. The command allows to create an opening at the location of the selected base profile. The base profile can be a boundary or a closed polyline.

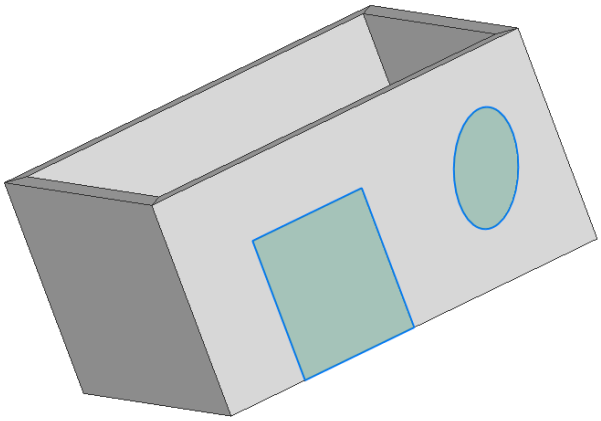
For more information about this command, visit the Command Reference article BIMWINDOWCREATE.

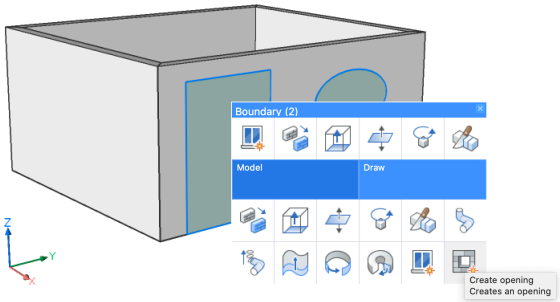
1 Draw one or more closed polylines on the face of a wall.

2 While boundary detection is on, hover over the polygon.

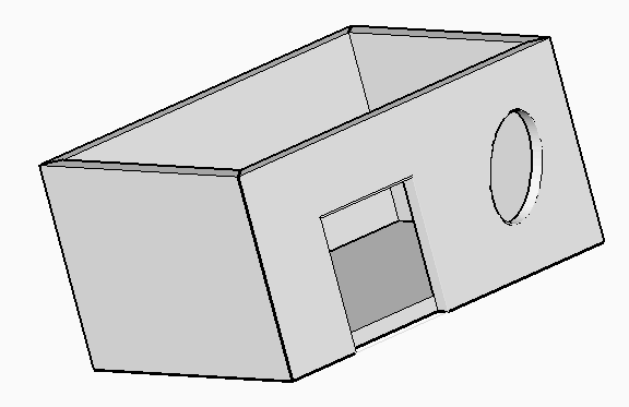
3 Or select the closed polyline.

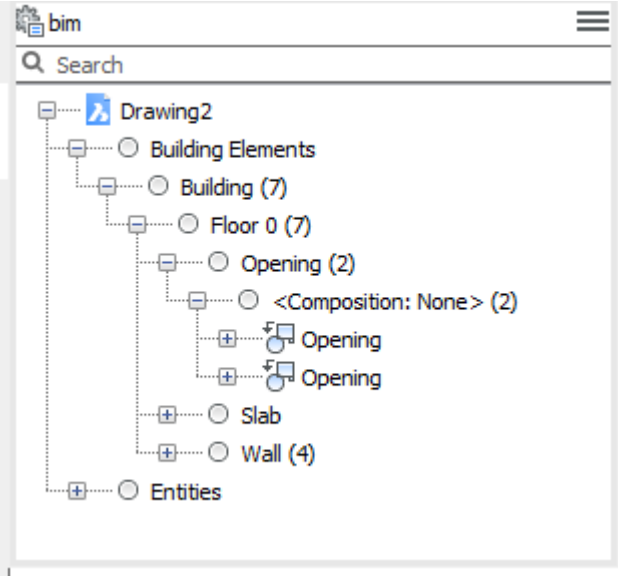
4 Select the **Create Opening** tool from the Quad.



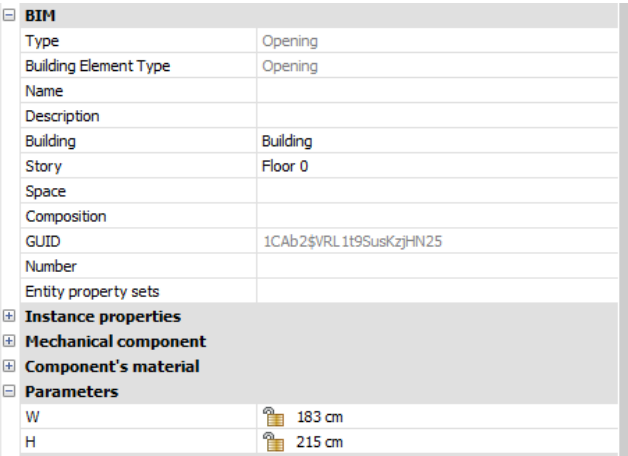


5 The opening is created based on the closed contour. In the **Structure Browser**, the opening is added as a block reference. The block reference is classified as an opening.





If you select the opening from the structure browser, you will see a set of properties related to this opening element in the **Properties** panel.



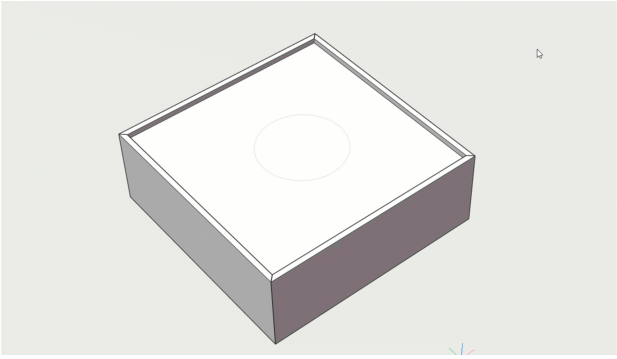
BIM	
Type	Opening
Building Element Type	Opening
Name	
Description	
Building	Building
Story	Floor 0
Space	
Composition	
GUID	1CAb2$VRL1t9SusKzjHN25
Number	
Entity property sets	
Instance properties	
Mechanical component	
Component's material	
Parameters	
W	183 cm
H	215 cm

Nota: If your closed contour has a rectangle shape, the opening element will include Width and Height parameters.

Nota: Opening elements contain subtractor solids. To display the subtractor solid, open the **Layers** panel and select the **BIM_Subtract** layer.

Nota: The associative tags can be generated using the BIMTAG command for the opening element in a sectioned drawing.

The following animated gif demonstrates how to create an opening based on a closed contour.



20.3.3 The library panel

20.3.3.1 Commands

LIBRARYPANELOPEN, LIBRARYPANELCLOSE, CREATELIBRARYBLOCK, -CREATELIBRARYBLOCK

20.3.3.2 About

The **Library** panel is a quick way to keep your BIM blocks at your fingertips or access predefined ones. You can add any CAD model to the **Library**. You can also fully customize the name and category to make it easier to find. You can then insert a library block in your drawing by dragging and dropping. Keep your library clean by deleting the items you no longer need.

20.3.3.3 Content of the Library panel

You can control the content of the **Library** panel in the **Option** menu.

Nota: When the COMPONENTSPATH system variable is empty (all paths deleted), the default **C:\ProgramData\Bricsys\Components** path is added automatically when you close and reopen the **Settings** dialog box.

20.3.3.4 Creating a library block

1 Open or create the drawing which contains the geometry for the new block.

2 Do one of the following:

- Click the **Home** icon ( ).

  This allows you to select the destination category; or you can create a new category.

- Click the **Category** thumbnail.

  The new item will be created in the selected category or in a subcategory of the selected category.

- Launch the CREATELIBRARYBLOCK command.
- The **Add block to library** dialog box displays.
- Select the entities or choose the **Entire drawing** option.

- Type a name for the block.
- (**Option**) Type a new category name or choose a category from the list.
- (**Option**) Click **Show Options** and set the block creation options.
- Click the **Create** button.

20.3.3.5 Managing library blocks

1 In the **Grid view** display, hover over the block icon.

2 Click the **Menu** icon (...).

3 (**Option**) Click the **File Open** icon (1) to open the block file.

Edit the file, then save the changes.

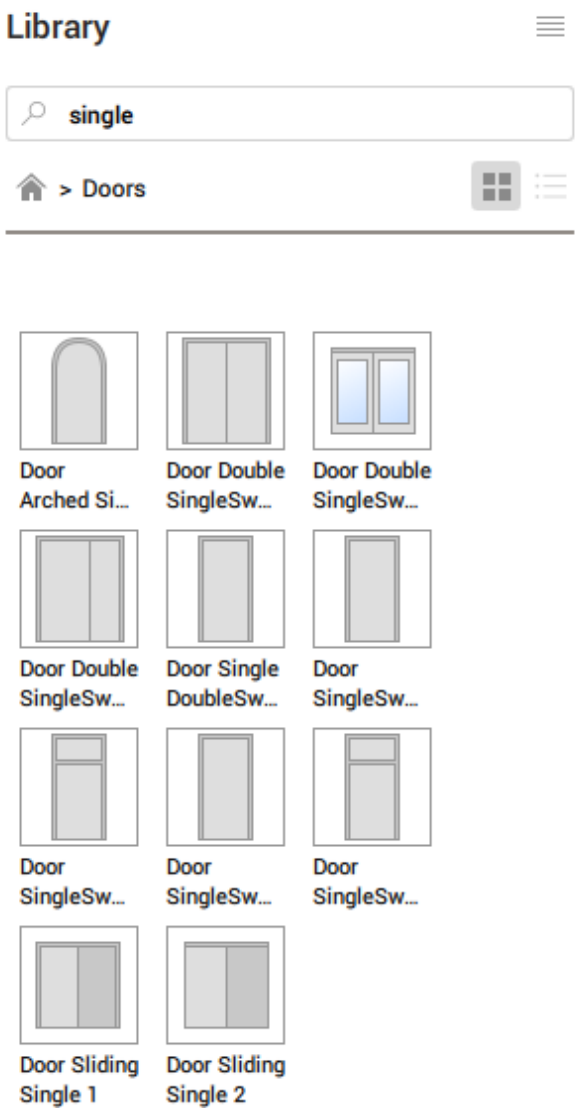
4 (**Option**) Click the **Delete** icon (2) to delete the block file.

Click **Yes** to confirm.

20.3.3.6 Searching the block libraries

Type a search string in the search field.

Only matching library blocks in the current category display.

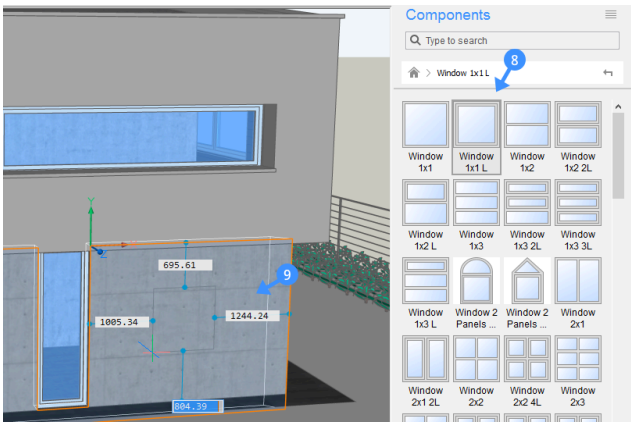


20.3.3.7 Inserting a BIM window into your 3D model

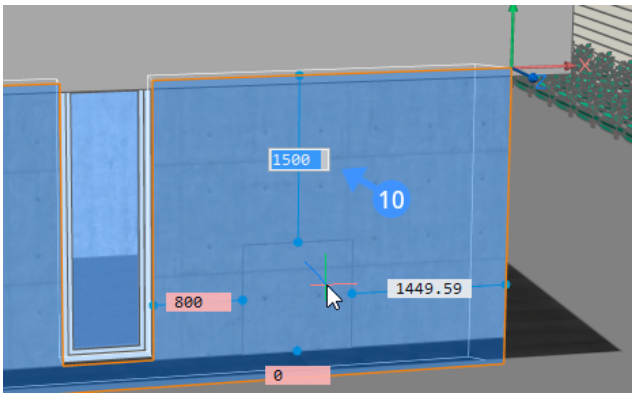
1 Click the **Windows** category tile in the **Library** panel.

2 Click a window (8).

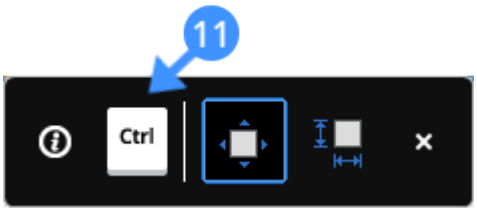
3 Drag and drop the window onto the wall (9).



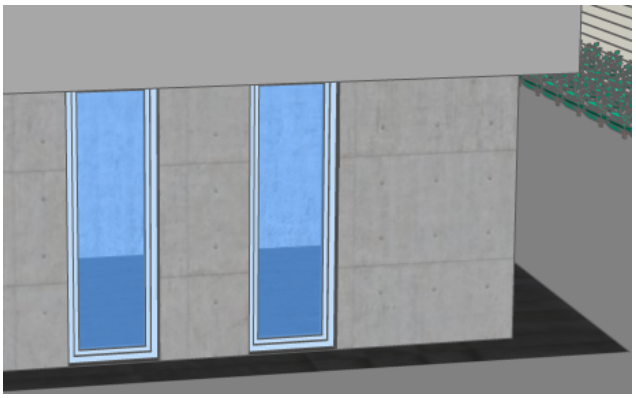
4 If Dynamic dimensions (**DYN**) is switched on, set the window location by entering the dimensions into the DYN fields (9). Press the Tab key to cycle through the DYN fields (10). A DYN field turns red to indicate the distance is locked.



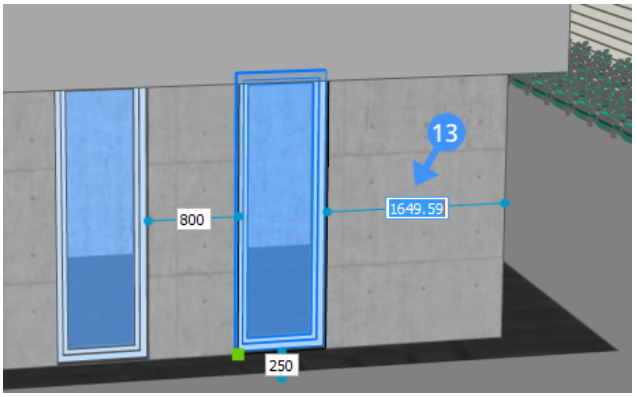
5 To edit the dimensions of the window press Ctrl (11) once and then use the Tab-key to cycle through the DYN fields (12).



6 Press Enter. The window is created. This same technique can be used for most of the blocks in the **Library**.



7 (**Optional**) Once you have a window you can adjust the position by selecting the window and then clicking on the dimension lines and editing the dimensions (13).



8 (**Optional**) To adjust the dimensions of the window after the window has been created, go to the **Properties** panel. Scroll down to the **Parameters** section and change the parameters as required. All windows and doors in the Library are parametric.

Parameters	
Name	Window_1x1_L:9
File	
Physical material	
W	800 mm
H	250 cm
Rebate	0 mm
PlacementDepth	90 mm
FrameThickness	60 mm
FrameDepth	50 mm
LeafThickness	45 mm
LeafDepth	60 mm
PaneOffset	20 mm
LeafOffset	20 mm

20.3.4 Connecting solids

20.3.4.1 Commands

LCONNECT, TCONNECT, BIMSTRUCTURALCONNECT

20.3.4.2 About

Walls, profiles, plies, etc. are all created as native 3D solids. So, any of the 3D Solids Editing commands can be used to modify them. Use **BIM Connect** commands to connect two solids in different ways.

The LCONNECT command creates an L-connection (bisector or parallel) between two solids.

The TCONNECT command creates a T-connection between two solids.

The BIMSTRUCTURALCONNECT command connects structural profile solids such as beams and columns.

The BIMFLOWCONNECT command creates a connection between flow segments, such as pipes or HVAC ducts.

The procedures below use the Quad cursor menu.

20.3.4.3 Procedure: creating L-connections

1 Make sure SELECTIONMODES system variable = 0.

2 Type **LConnect** in the Command line or select it via the Quad.

3 Select two solids or faces to connect.

4 The two selected solids are connected: the default connection type is L-bisector.

5 If HOTKEYASSISTANT system variable is ON (HKA is switched on), the Hotkey Assistant widget displays.

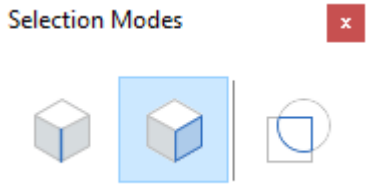
Use the CTRL key to cycle between the connection types **or**
You are prompted: Select base solid [Switch] <Accept>. Type **A**to use the selected connection, type **S** to switch to the desired connection type.
The 3D model updates dynamically when cycling the L-connection types.

Nota: L-connections can also be created between non-vertical solids such as roof slabs.

20.3.4.4 Procedure: Creating T-connections

The following procedure allows connecting minor faces of a solid to the major faces of other solids, regardless of what the orientation of the faces is. As a result, you can use TCONNECT command to T-connect solids.

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.

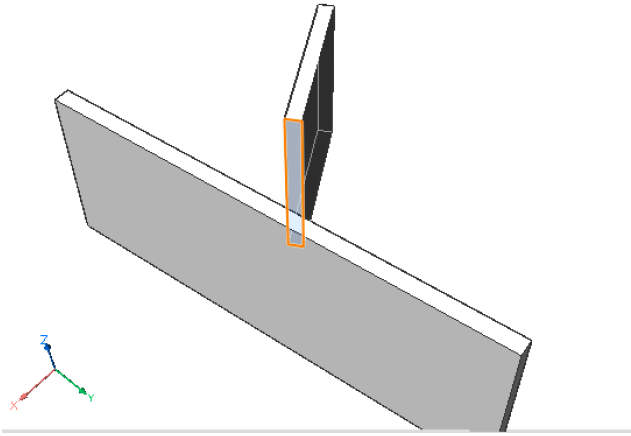


2 Type TCONNECT in the Command line or select it via the Quad.

3 Move the cursor over the minor face of the first solid at the end that needs to be connected.

4 Press Tab to highlight obscured faces.

Click when the face highlights.



5 (**Optional**) Repeat the previous step to select more faces, then press Enter.

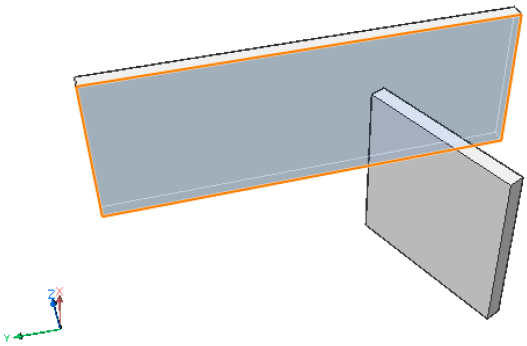
Suggerimento: The SELECTALIGNEDFACES command in the **Select** command group in the Quad selects all faces in the 3D model that are in the same plane (coplanar) with a selected face.

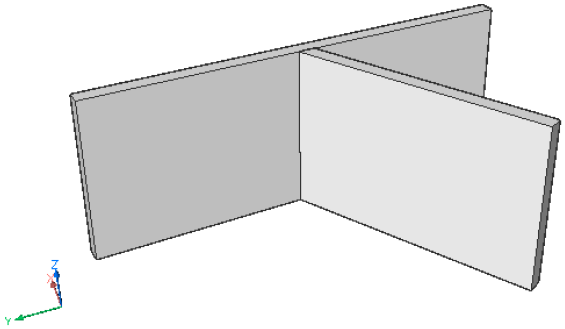
6 You are prompted: Select entities to connect to or [Connect to nearest/Disconnect/selection options (?)] <Connect to nearest>:

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.

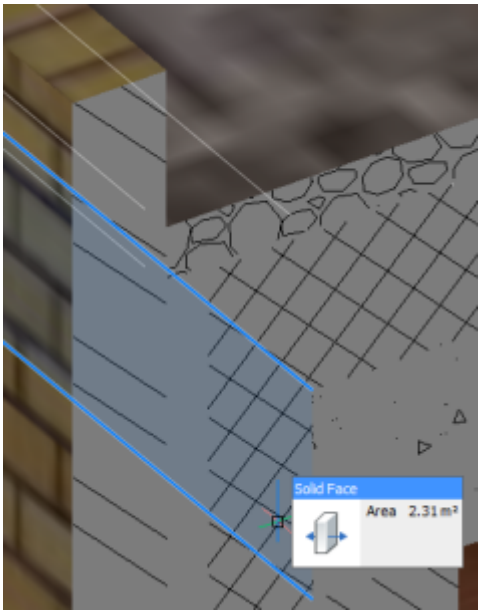
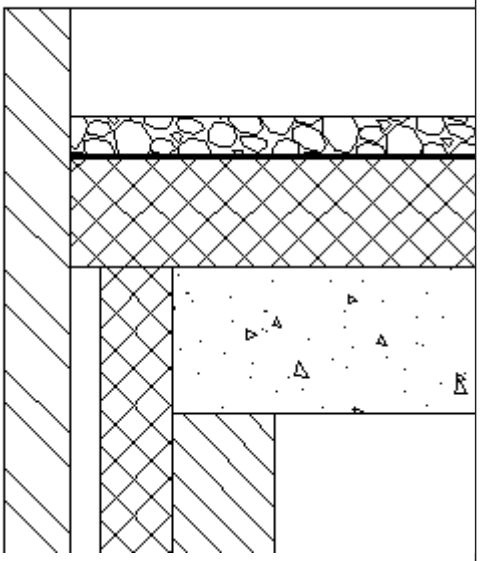
  Right click to stop selecting faces.





Suggerimento: Hover over the minor face of the solid and choose **Connect with Nearest in the Model** command group of the Quad.

Creating T-connections ply-by-ply

	
3DModel	Calculated Section

To control the connection of composition plies between two building elements, you can **Push/Pull** a ply in a clipped BimSection (if Show Composition is ON). For more in-depth information, visit the BIM Ply Editing article.

To create a T-connection ply-by-ply in a section, do the following.

1 Place the cursor over the face of the ply you want to connect. If necessary, repeatedly press Tab to select an obscured face.

2 When the ply face highlights, choose **Push/Pull** in the Quad.

3 Specify to which point the selected ply should reach by moving the cursor or typing in the distance.

4 Right-click or press Enter to confirm.

5 The ply of the first solid is connected to the ply of the second solid. The connected ply is subtracted from the second solid.

6 Optionally, use the BIMPROPAGATEPLANAR command to apply the connection to similar locations in the model.

20.3.4.5 Procedure: Setting the distance between parallel solids

1 Select the solid of which the position must be fixed.

2 Select the solid which will be moved when defining the new distance.

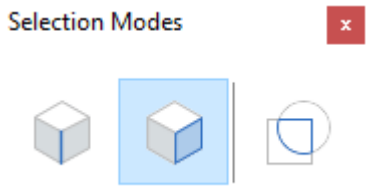
3 The nearest distance between the solids displays.

4 Type a value in the distance field and press Enter.

20.3.4.6 Procedure: Creating a copy of a solid

Creating a parallel copy of a solid

1 Make sure the **Select Faces** option of the SELECTIONMODES system variable is selected.



2 Move the cursor over a face of the solid to be copied.

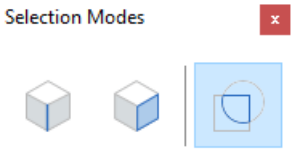
3 When the face highlights, choose **BimCopy** in the **Model** command group of the Quad.

A copy of the solid displays dynamically. The ruler indicates the current distance.

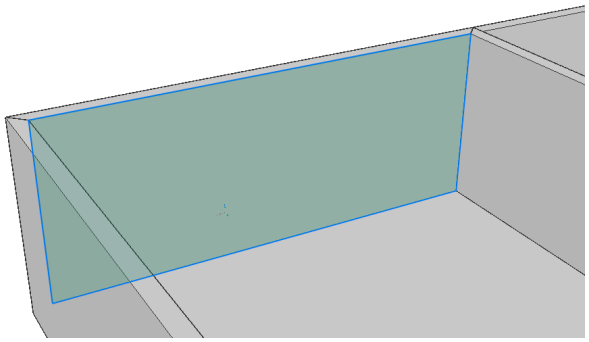
4 Type a value in the dynamic entry field and press Enter.

Creating a partial copy of a solid

1 Make sure the **Enable boundary detection** option of the SELECTIONMODES system variable is selected.



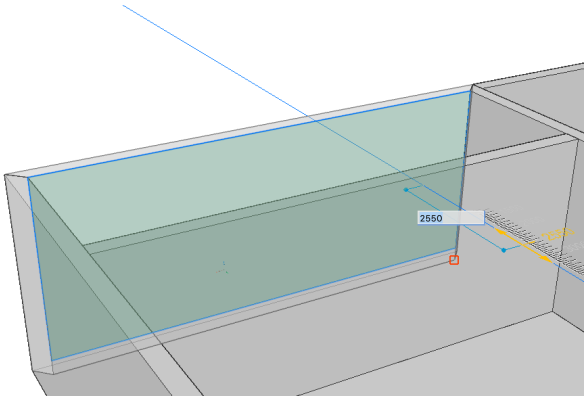
2 Highlight the part of the boundary for the part of the solid you want to copy.



3 When the face highlights, choose **BimCopy** in the **BIM** command group of the Quad.

A partial copy of the solid displays dynamically. The ruler indicates the current distance.

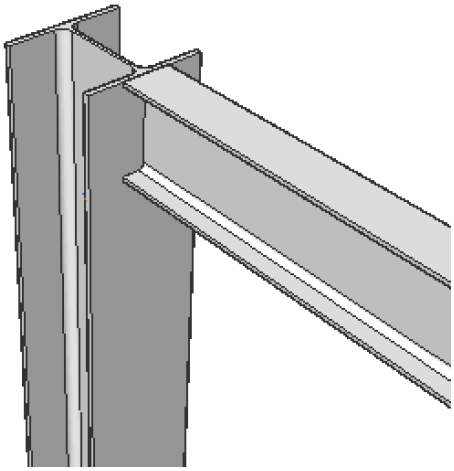
4 Type a value in the dynamic entry field and press Enter.



20.3.4.7 Procedure: disconnecting two connected solids

In some cases, you need to disconnect 2 solids of a node to do some further direct modeling.

You select two solids and it breaks the connection by separating the two solids physically and removes all end clippings at the side of the connection. As such, these solids can direct modeled upon again.

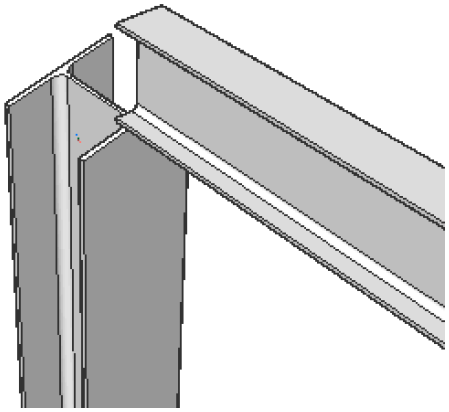


1 Type LCONNECT in the Command line.

2 Select the two solids.

3 The connection is automatically made, press CTRL until the disconnect icon is highlighted.

4 Press Enter.



Nota: You can use Propagate to disconnect similar nodes.

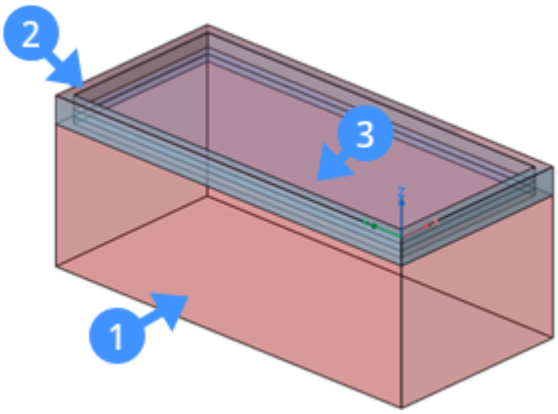
20.3.5 Creating a custom window

20.3.5.1 Creating a custom window

In BricsCAD, you can create any geometry or component from scratch. Once you have your geometry in 3D you can insert and use it in your BIM model. You can make changes or add more details to your window at any time.

This process is explained in 4 parts to make the steps of creating a 3D window geometry straightforward.

- Create the file
- Create the subtractor solid (1)
- Create the window frame (2)
- Create the (Glass) panes (3)

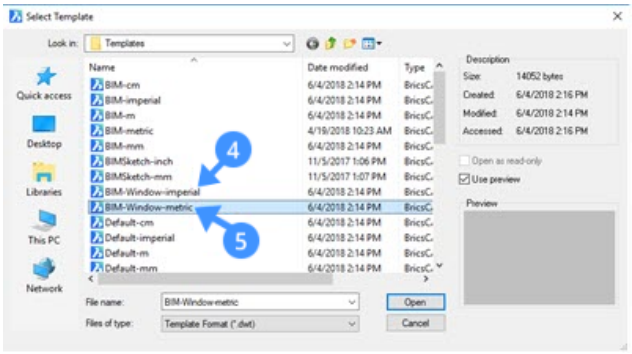


Step 1: Creating the file

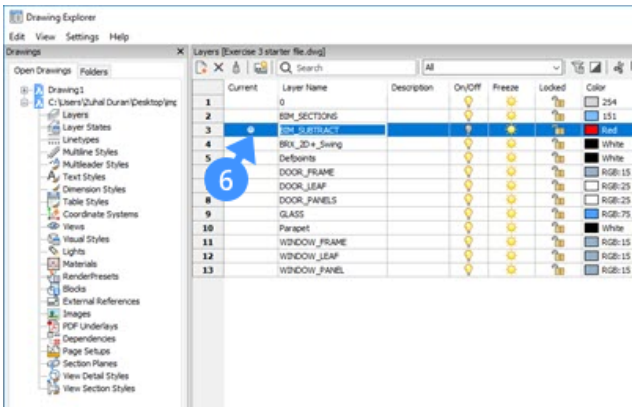
1 Click **New** in the **File** menu.

2 Select one of the pre-defined templates to create a simple window specifying the unit system of your drawing.

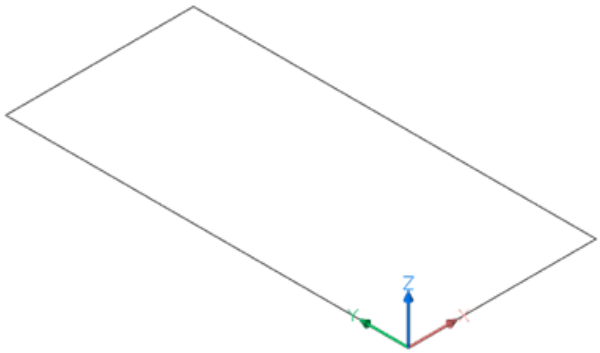
Nota: The pre-defined templates contain the layers needed to create the geometry of the window or door. These .dwt file format templates are **BIM-Window-imperial** (4) and **BIM-Window-metric** (5) which assure the required layers and properties are displayed in your drawing.



3 Set the **BIM_Subtract** layer current (6) either using the drawing explorer or layer field in the **Properties** panel or **Layers** panel on the right-hand side.



4 Create a closed polyline (e.g. a rectangle) that represents the form of the opening for the insert.



5 Save the file in any location. The library of predefined windows and doors is located in the support folder.

6 By default this is: C:\Users<user_name>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support

\Bim\Components\Windows This path can also easily be found by typing SUPPORTFOLDER in the Command line.

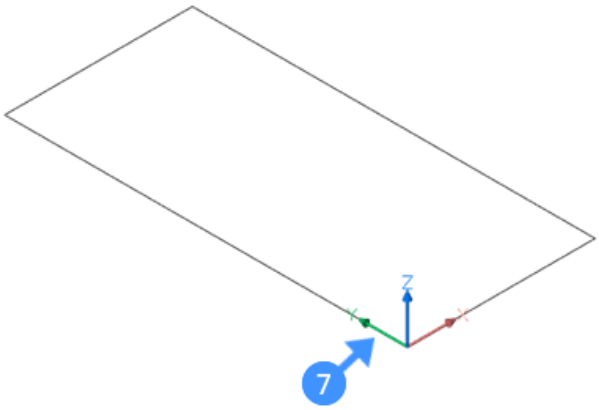
Step 2: Creating the subtractor solid

The subtractor solid will be used to create an opening.

Nota: **BIM Subtract** layer must be a current layer.

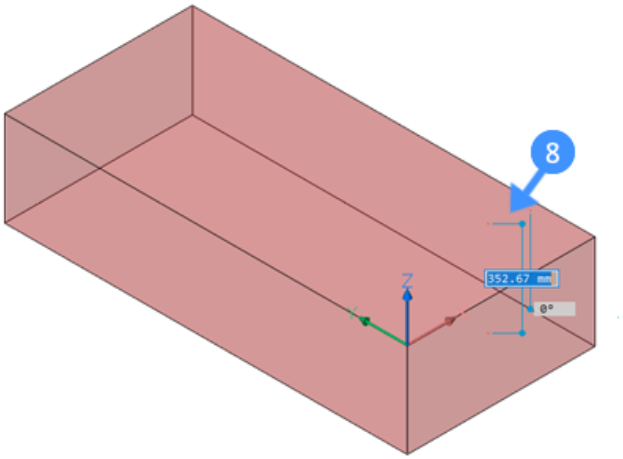
1 Use the outline of the solid which you have previously created (e.g. a rectangle).

1 The outline lies in the first quadrant of the XY plane of the World Coordinate System (WCS).



The lower left corner of the rectangle lies at the origin (0,0,0) of the WCS (7).

2 Select the **Extrude** tool and move the cursor downward (8) to extrude your solid outline in the negative Z direction.



3 Do one of the following to finish the subtractor solid creation.

- Click a point.
- Type a value in the dynamic dimension fields, then press Enter.

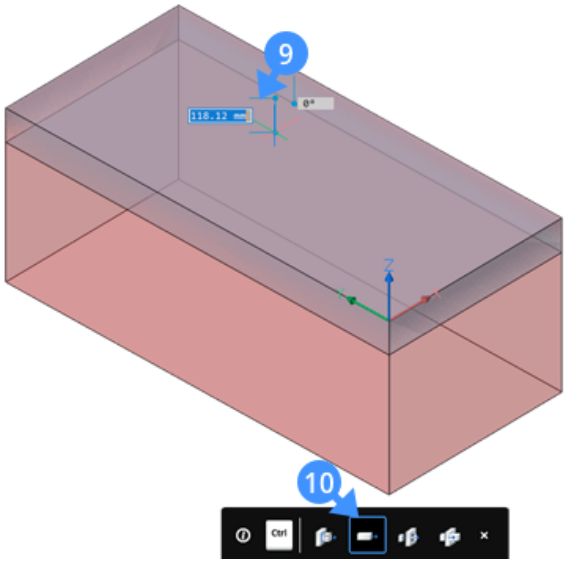
Step 3: Creating the fixed frame

The outline of the fixed frame coincides or is parallel (in case of a rebate) with the outline of the subtractor solid.

1 Set the **Window_Frame** layer current.

2 Highlight the top face of the subtractor solid.

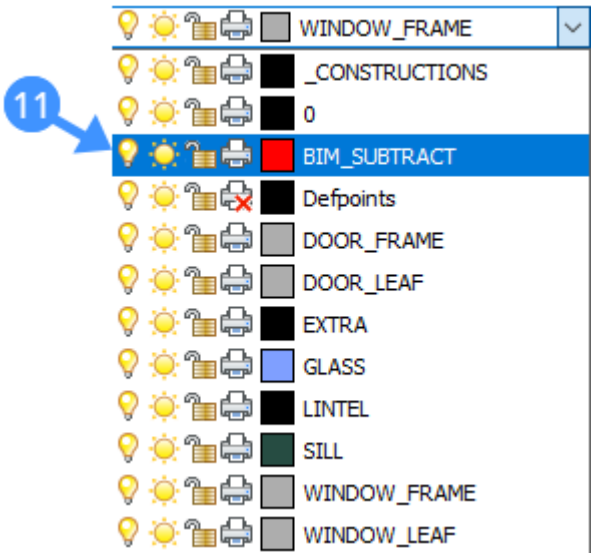
3 Next, extrude the top face of the subtractor solid in the negative Z direction (9) using the **Extrude** tool 'Create' (10) option by pressing the CTRL key once.



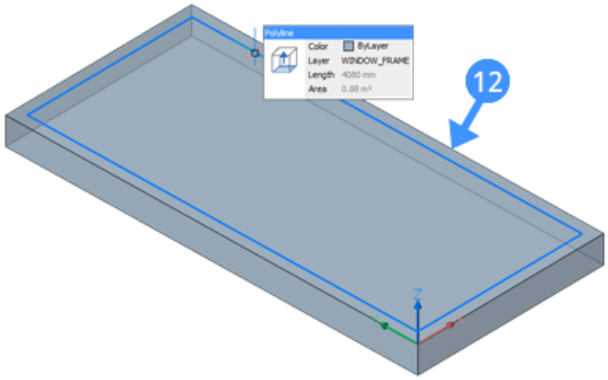
4 Type a value in the dynamic dimension field to specify the height of the fixed frame.

Nota: The height of the extrusion will be the thickness of the fixed frame.

5 Turn off **BIM_SUBTRACT** layer by clicking the light bulb (11) next to the layer name.

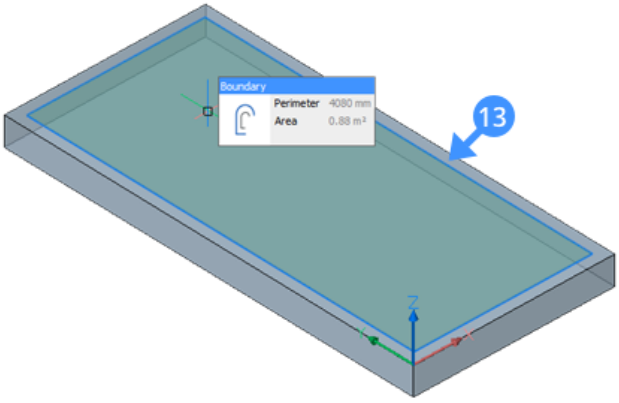


6 Use the **Offset** tool to create a parallel offset of the outline of the fixed frame (12).

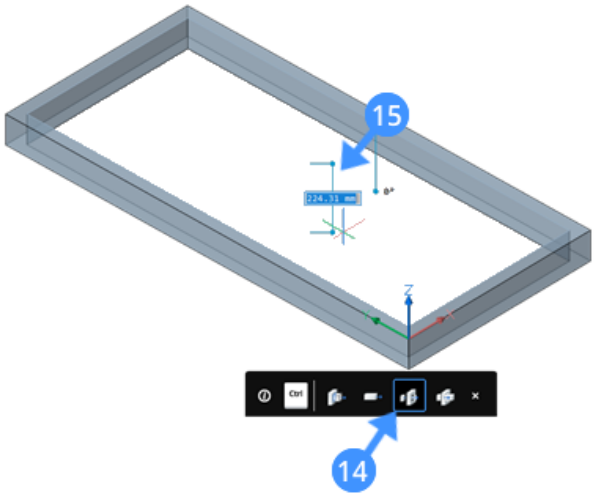


Nota: The distance of the offset equals the width of the frame.

7 Hover over the inside boundary of the parallel offset (13) and select **Extrude** tool from the Quad.



8 When the extrude mode is on the **Subtract** option (14) move your cursor downward (15) to create an opening in the fixed frame.



9 Do one of the following to finish the creation of the opening in the fixed frame.

- Click a point.

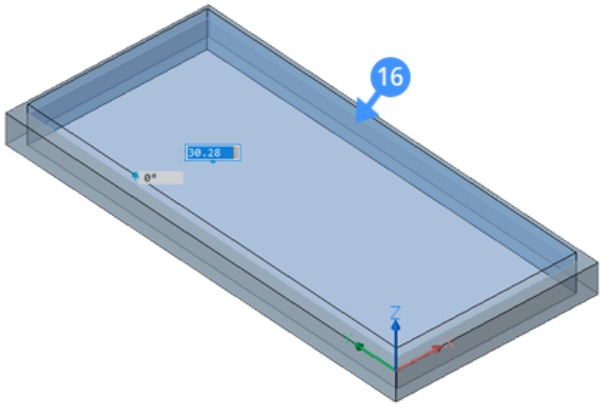
- Type a value in the dynamic dimension fields, then press Enter.

Step 4: Creating the glass panes

1 Set the **Glass** layer current.

2 Use the DMEXTRUDE command to extrude the inside boundary of the window frame.

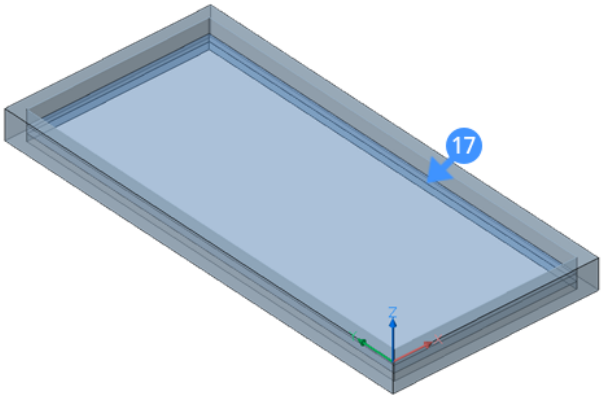
Nota: The height of the extrusion equals the thickness of the glass pane. The following illustration shows the inside boundary of the window frame after using DMExtrude to create a window glass pane.



Note that to create a glass pane, the create option should be selected from the Tooltip of **DMExtrude**.

3 Move the glass pane solid down in the negative Z-direction if the outside face of the fixed frame does not coincide with the outside face of the solid (wall) the insert will be placed in.

4 The following illustrations show the glass pane solid after moving it (17) down.



5 Optionally, before saving the drawing, classify the BIM as a window element.

6 To do so, follow these steps:

- Type BIMCLASSIFY in the Command line.
- Press 'I' for window and press enter to accept it.
- Next, press 'D' for Drawing and press the Enter key until the command is executed.

The drawing is classified as a window. When you insert this block in your drawing, it will automatically

be classified as a window.

7 Optionally, when no selection is active, you see a set of BIM properties on the right-hand side of your screen. Here you can assign information about the thermal transmittance coefficient of the window materials, fire rating according to the national fire safety classification, and so on. The following illustration shows the BIM properties of the window in the properties panel.

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m³/s
Thermal transmittance	0 W/m²·K
Glazing area fraction	0
Smoke stop	Off

8 Save the drawing.

Adding 2D entities

Optionally, windows and doors drawings can contain 2D symbols. These 2D symbols will either replace the actual **3D Solid** section of the window or door or be added to the **3D Solid** section, depending on their layer:

- Symbols on a layer with prefix BRX_2D_will be used to replace the geometry resulting from the actual section.
- Symbols on a layer with prefix BRX_2D+_will be added to the section result. Only those 2D entities in a plane parallel to the section plane are.

20.3.6 Creating roofs

20.3.6.1 The roof creation workflow consists of the following steps:

1 Creating the roof footprint.

2 Extrude the footprint and create a hip roof (DMEXTRUDE command).

3 Adjust the slope of the roof faces if necessary (**Manipulator** widget).

4 Defining the thickness of the roof solid (**Body/Shell** option of the SOLIDEDIT command).

5 Create additional roof components.

20.3.6.2 Creating the roof footprint

You can create the roof footprint:

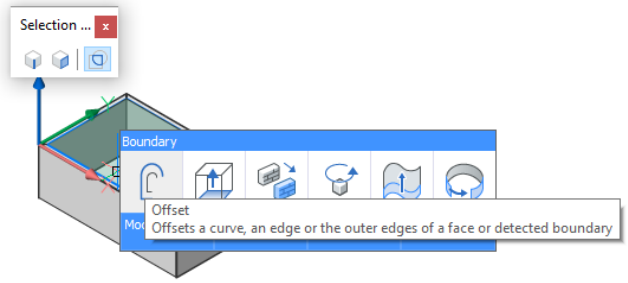
- From the exterior walls boundary.
- Creating a closed polyline by clicking points or other 2D procedures.

To create the roof outline from the exterior walls boundary

1 Isolate the exterior walls. Isolating entities is easily done using the **Structure** Panel. See also the ISOLATEOBJECTS command.

2 Set the coordinate system to the top face of the exterior walls:
   - Type UCS and press Enter.
   - You are prompted: Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:
   - Snap to a top corner of an exterior wall, then press Enter.

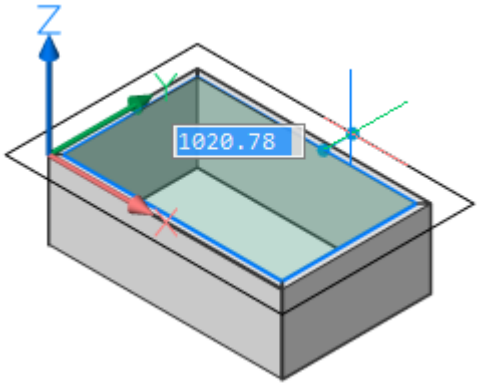
3 Set the **Enable boundary detection** ON in the **Selection Modes** toolbar or include the **Select detected boundaries** into SELECTIONMODES system variable.



4 Move the cursor to avoid area inside the exterior walls.

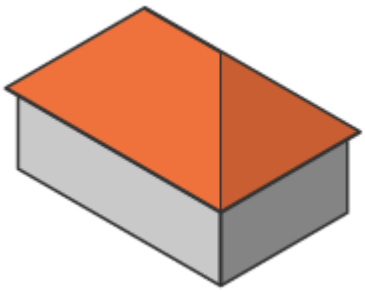
5 The inside boundary highlights.

6 Choose **Offset** in the **Draw** tab of the Quad.



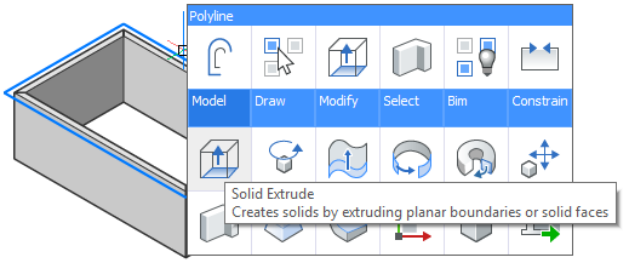
7 Do one of the following:
   - Type a value in the dynamic distance field.
   - Click to accept the value.

20.3.6.3 Creating a hip roof

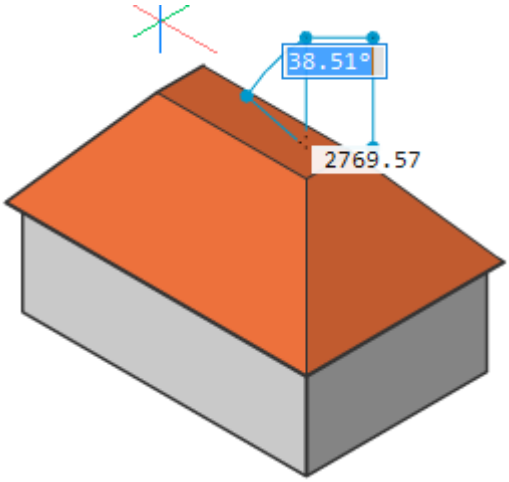


1 Create the roof footprint.

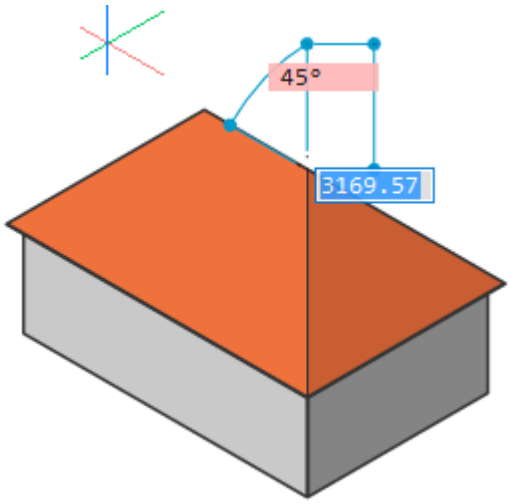
2 Move the cursor over the roof outline and choose **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the Tab key.



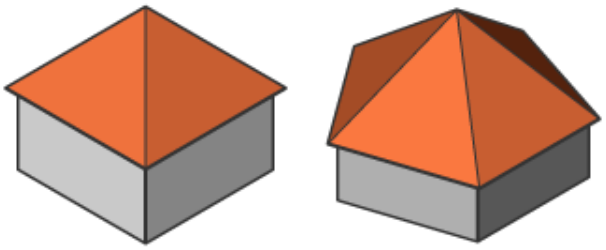
4 Type a value in the **Taper Angle** field and hit the Tab key.



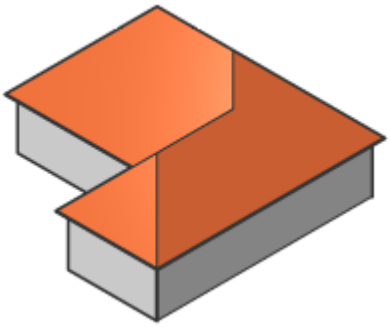
5 Do one of the following:
   - Hit the Tab key and type another value in the **Taper Angle** field.

     The **Taper Angle field** is the complement of the inclination angle of the roof.
   - Press Enter to create the roof.

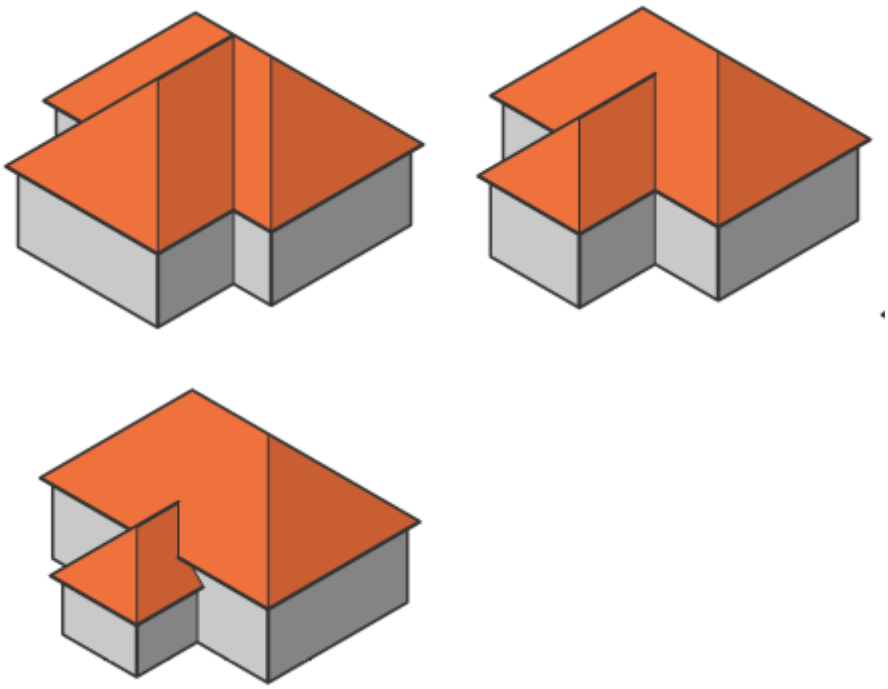
6 Depending on the shape of the roof outline, the result is:
   - Pyramid roof:



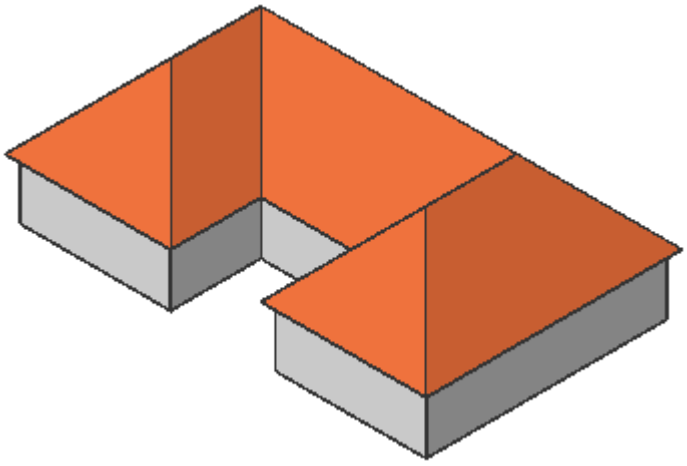
   - L-shaped hip roof:

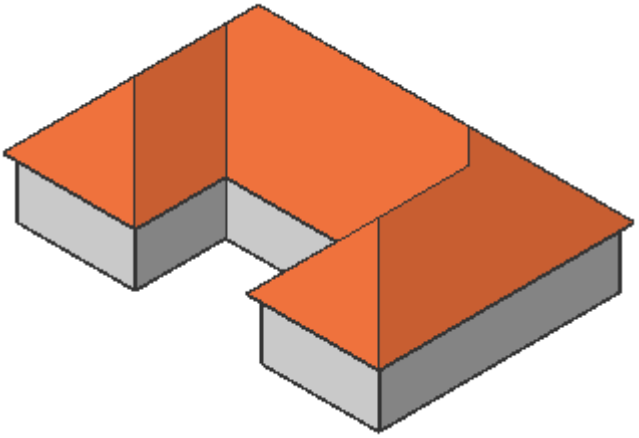


   - T-shaped hip roof:

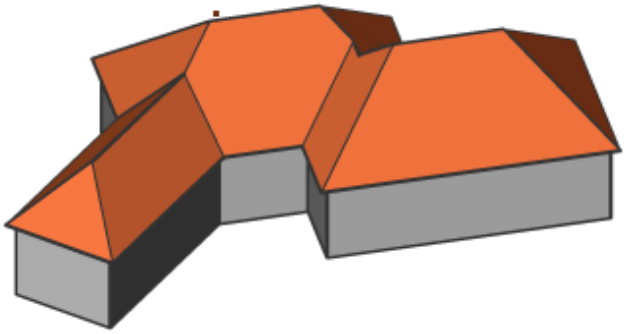


- U-shaped hip roof:





- Complex hip roof:



7 To define the thickness of the roof solid we will use the **Body/Shell** option of the SOLIDEDIT command.

Do one of the following:

a Choose **Shell** in the **Solid Edit | 3D Solid Editing** ribbon panel.
Or Choose **Shell** in the **3D Solid Editing** toolbar.

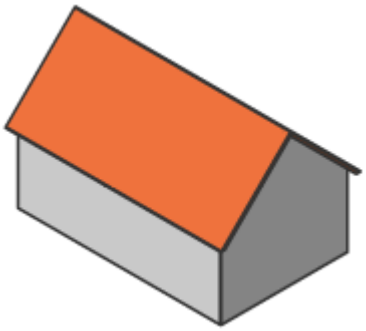
b You are prompted: Select 3D solid: Select the roof solid.

c You are prompted: Select face to remove or [Add/Undo/ALL]: Select the bottom face of the roof

solid.

d You are prompted: Enter the shell offset distance: Type a value to define the thickness of the roof solid.

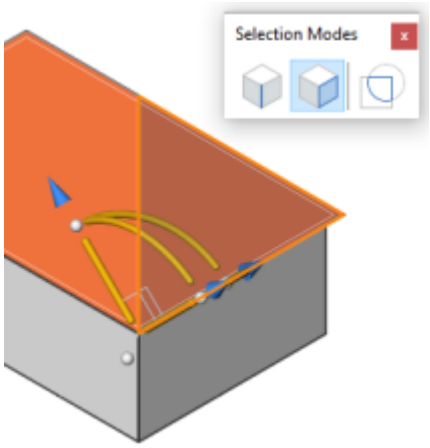
20.3.6.4 Creating a gable roof



1 Create a hip roof.

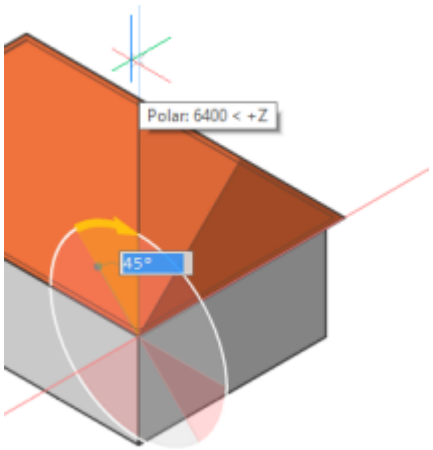
2 Make both end faces vertical:

- Move over the end face then hold down the left mouse button about half a second; the Manipulator widget displays (set the value of the MANIPULATOR system variable = 2).



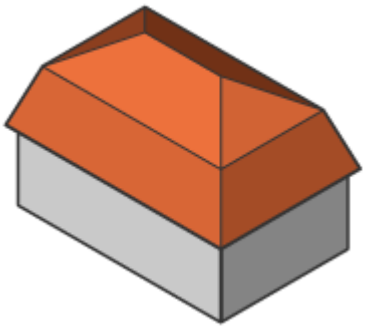
- Place the cursor over the vertical rotation arc. When the arc highlights, move the cursor.
- The selected face rotates dynamically.

  Click when the face is in a vertical position.



3 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

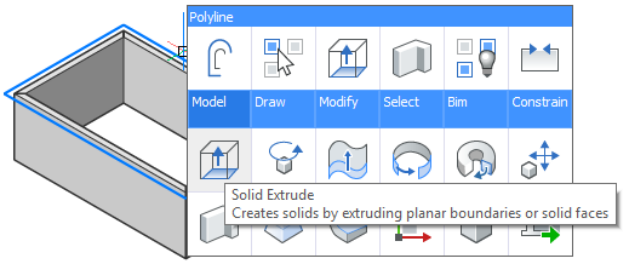
20.3.6.5 Creating a mansard roof



A mansard roof is composed of two solids, which we will create as two separate extrusions.

1 Create a roof footprint.

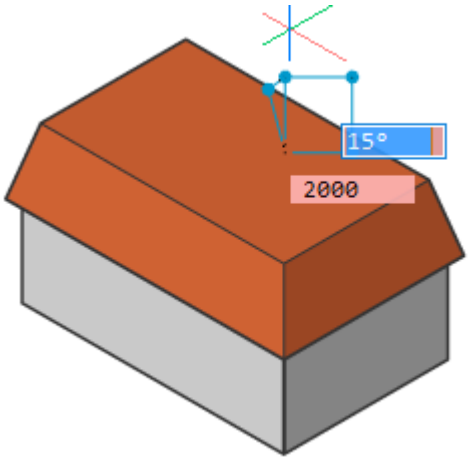
2 Move the cursor over the roof outline and choose to **Extrude** in the **Model** tab of the Quad.



3 Move the cursor vertically, then hit the Tab key.

4 Type a value in the **Taper Angle** field then hit the Tab key.

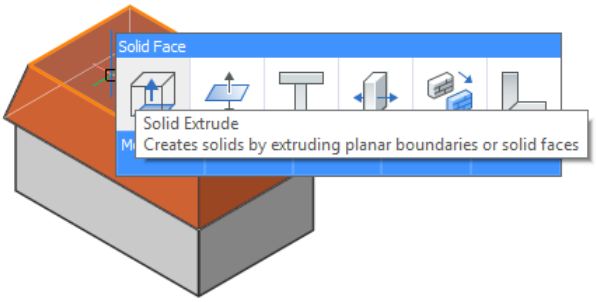
The **Taper Angle** field is the complement of the inclination angle of the roof.



5 Type a value in the **Extrusion Height** field, then do one of the following:
   - Hit the Tab key and type another value in the **Taper Angle** field.
   - Press Enter to create the first solid.

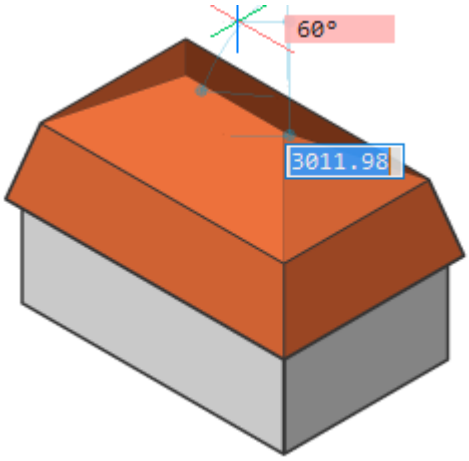
6 Optionally hit the Tab key to go back and forth between the entry fields and press Enter when you are satisfied with the result.

7 Move the cursor of the top face of the first roof solid and choose **Extrude** in the Quad.



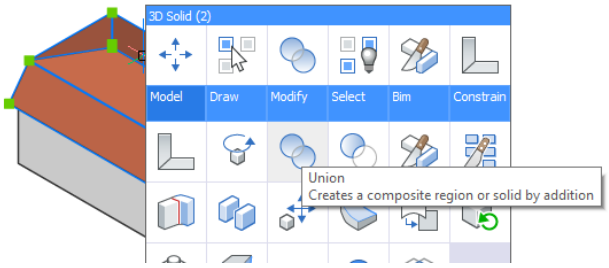
8 Move the cursor vertically, then hit the Tab key.

9 Type a value in the **Taper Angle** field then hit the Tab key.



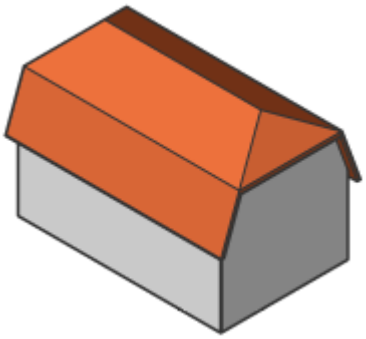
10 Do one of the following:
   - Hit the Tab key and type another value in the **Taper Angle** field.
   - Press Enter to create the first solid.

11 Select both solids and choose **Union** in the **Model** tab in the Quad.



12 Repeat step 7 of the Create a hip roof procedure.

20.3.6.6 Creating a jerkinhead roof



A jerkinhead roof is composed of two solids, which we will create as two separate extrusions.

1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

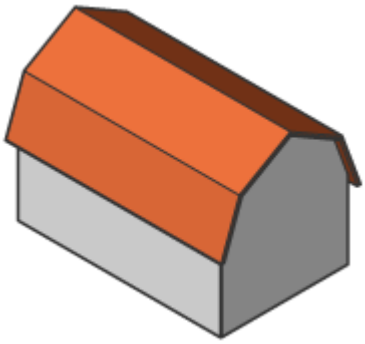
2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

3 Execute steps 7 to 11 of the Creating a mansard roof procedure.

4 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

20.3.6.7 Creating a gambrel roof

A gambrel roof is composed of two solids, which we will create as two separate extrusions.



1 Execute steps 1 to 6 of Creating a mansard roof procedure, to create the first solid.

2 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

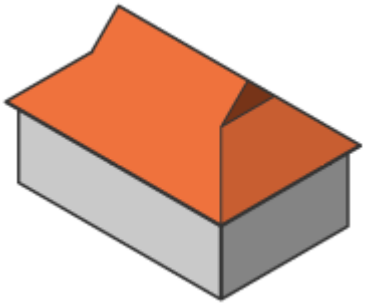
3 Execute steps 7 to 10 of Creating a mansard roof procedure to create the second solid.

4 Execute step 2 of Creating a gable roof procedure to set the end faces vertical.

5 Execute step 11 of Creating a mansard roof procedure.

6 Execute step 7 of the Create a hip roof procedure. Remove the vertical faces of the roof solid.

20.3.6.8 Creating a dutch gable roof

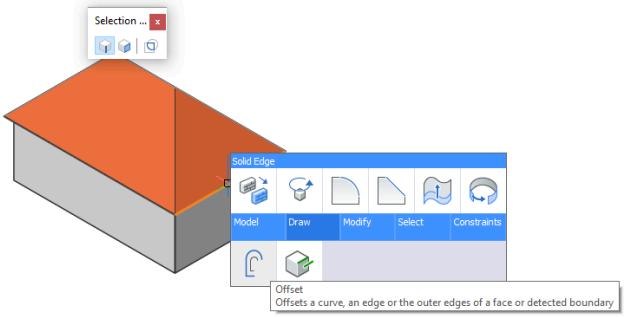


A Dutch gable roof is based on a hip roof.

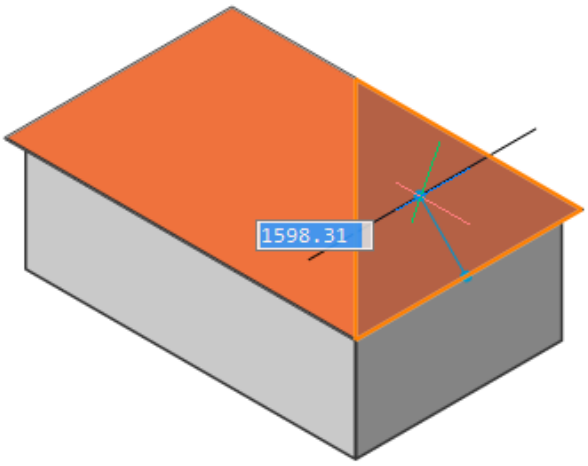
1 Create a hip roof.

2 Set the **Select Edges** option of the SELECTIONMODES system variable ON.

3 Move the cursor over the bottom edge of the roof face, then choose **Offset** in the **Draw** tab in the Quad.



4 Move the cursor and type a value in the dynamic distance field or click a point.



5 Do one of the following:
   - Choose **Imprint** in the **3D Solid Editing** toolbar.
   - Choose **Imprint** in the **Solid Edit | Solid Editing** ribbon panel.

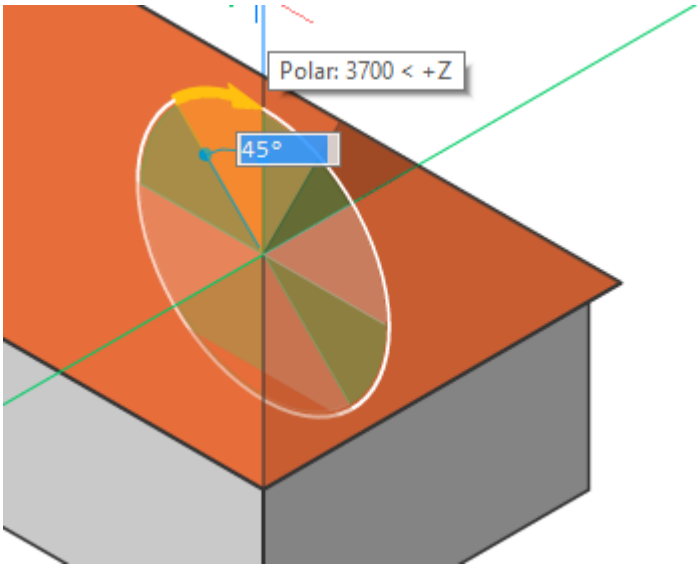
6 You are prompted: Select 3D solid, surface or region: Select the roof solid.

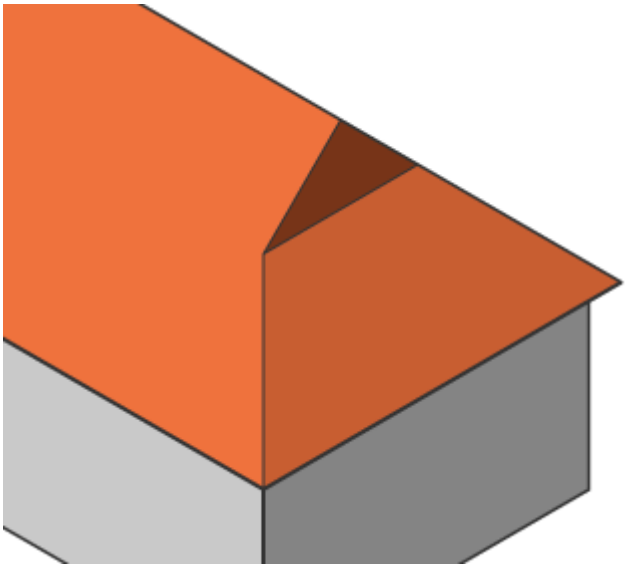
7 You are prompted: Select an entity to imprint: Select the offset roof edge.

8 You are prompted: Delete the source object[Yes/No] <No>: Choose Yes.

9 A triangular face is created.

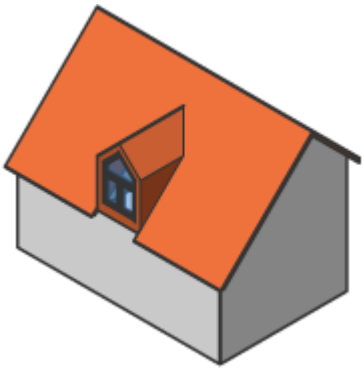
10 Execute step 2 of Creating a gable roof procedure to set the triangular face vertical.





11 Execute step 7 of the Create a hip roof procedure to set the thickness of the roof solid.

20.3.6.9 Creating dormers



The procedure to create a dormer consists of the following steps:

- Create a box solid at the bottom edge of the roof plane and connect the box to the roof.
- Define the position and dimensions of the dormer.
- Create a dormer roof.

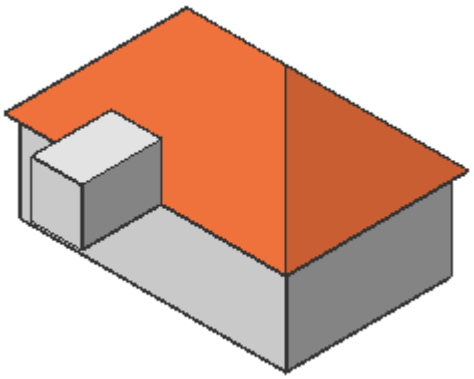
1 Right-click, then choose **Box** in the **Model** tab of the Quad.

   You are prompted: Set corner of box or [Center] <0.00,0.00,0.00>:

2 Click a point at the bottom edge of the roof plan.

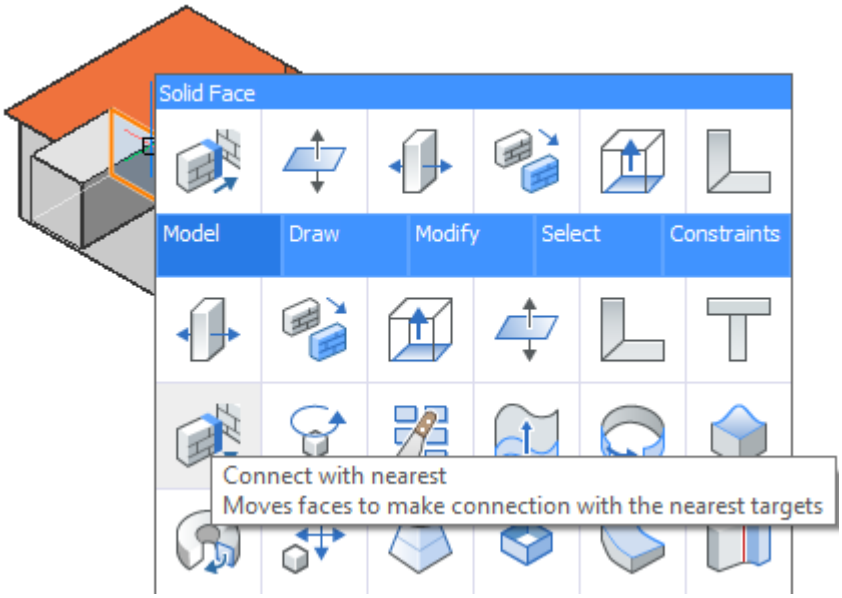
3 You are prompted: Set opposite corner or [Cube/Length of side]:

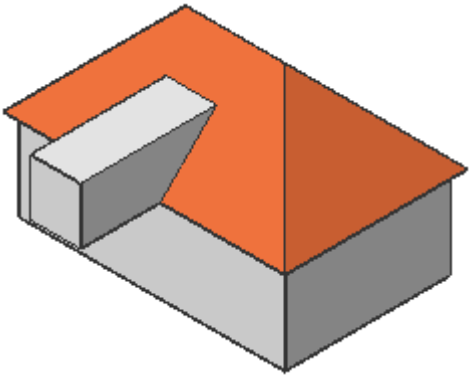
4 Click a point, then define the height of the cube: either click a point or type value.



5 Select the back face of the cube: move the cursor over the cube at the location of the back face, then hit the Tab key until the face highlights (Selection Modes: Select Faces = ON).

6 Choose Connect with nearest in the Model tab in the Quad.



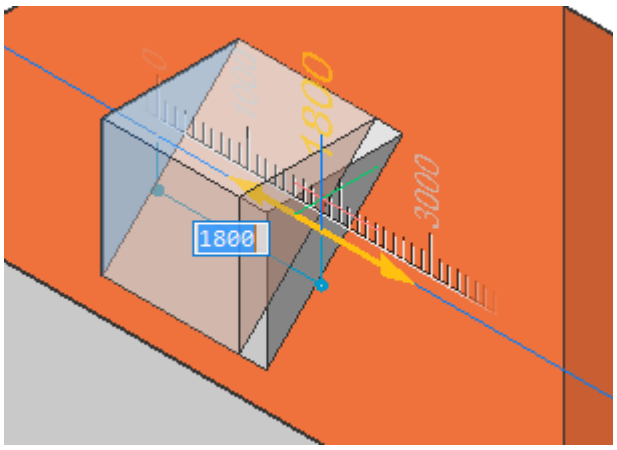


7 Select the front face of the box, choose **Push/Pull** in the **Model** tab in the Quad and move the face to the roof plane.

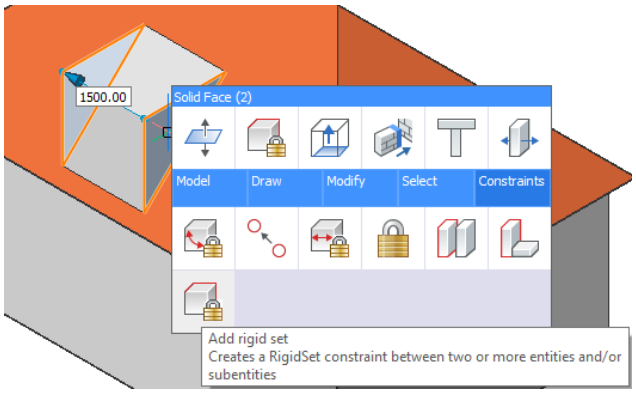
8 Use PUSHPULL command on the front, top and side face of the dormer solid to define the dimensions

and the position of the dormer on the roof:

- Select a side face, then hit the Tab key to define the width.



- Select both side faces, then choose to **Add rigid set** in the **Constraints** tab in the Quad to fix the width.



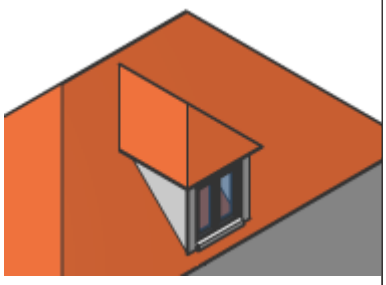
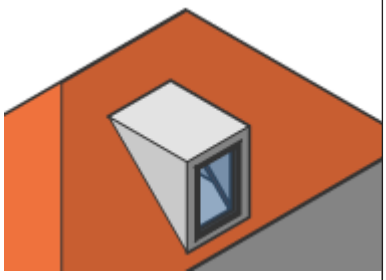
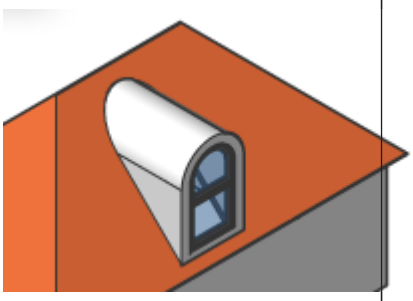
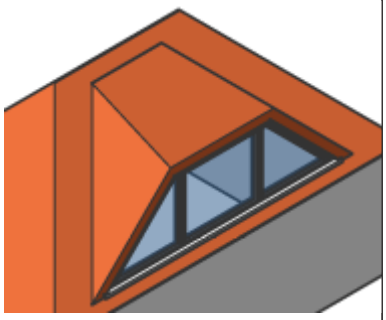
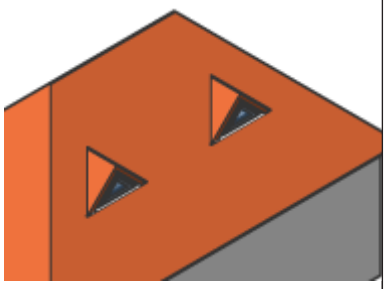
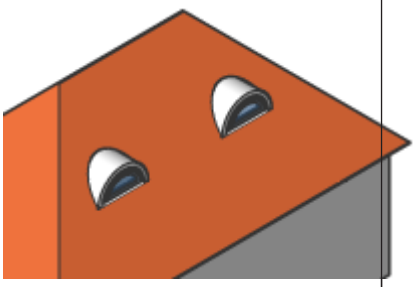
- Drag one of the side faces to position the dormer. Do one of the following:
  - Type a distance in the dynamic entry field.
  - Click to accept the value.
  - Move the cursor over a face that is parallel to the face being dragged and hit the Tab key. The distance field now shows the distance to this face.
- Drag the top face to define the height of the dormer.
- Drag the front face to set the distance from the roof eave.

Move the cursor over the exterior face of the wall beneath the dormer, then hit the Tab key. The distance field now shows the distance to this face.

9 Modify the shape of the dormer:

- Add a hip roof.
- Add a hip roof, then rotate the front face vertical to create a gable roof.
- Add a vault.
- Rotate top and side faces.

- Starting from a gable roof move the front face of the dormer backward until the front face is triangular.

		
Hip	Flat	Vault
		
Special	Triangle	Eyebrow

20.3.7 Creating slabs

20.3.7.1 Commands

DMEXTRUDE, POLYSOLID

Nota:

The DMEXTRUDE command allows to extrude a closed 2D entity such as polylines, circles, ellipses and extrude a detected boundary, enclosed by walls and/or 2D entities.

The POLYSOLID command allows to create pitched roofs.

20.3.7.2 Creating floor slabs between walls

Make sure that:

- The **Enable Boundary Detection** option of the SELECTIONMODES system variable is ON.
- The bottom faces of the surrounding walls are in the XY-plane of the current coordinate system.
- If necessary, move the origin of the coordinate system.
- Dynamic dimensions are ON.

1 Move the cursor inside the surrounding walls.

2 When the boundary highlights, choose **Extrude** in the **Model** command group in the Quad.

The slab is extruded dynamically.

3 Do one of the following:

- Type the thickness of the slab in the dynamic dimension field.
- Pick a point.

20.3.7.3 Creating a pitched roof slab

Make sure that:

- **Dynamic UCS** is ON.
- **Dynamic dimensions** (DYN) is ON; it is recommended to have the **Tracking dynamic dimensions** option of the DYNMODE system variable set.
- **Entity Snap Tracking** (STRACK) is ON.
- **Endpoint entity snap** is ON.

1 Launch the POLYSOLID command.

1 You are prompted: Start point or [Height/Width/Justification/Entity/Separate solids/Dynamic] <Entity>:

2 Move the cursor over the outside face of a wall which is perpendicular to the roof slab you want to create.

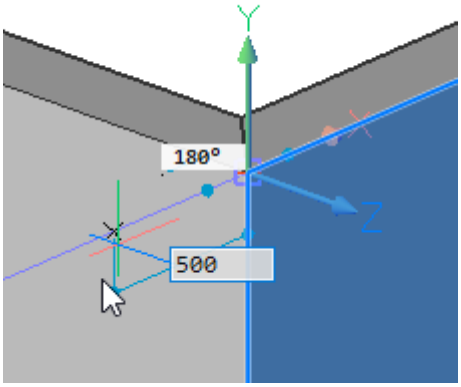
3 When the face highlights, hit the Shift key to lock the **dynamic UCS**.

4 Do one of the following:

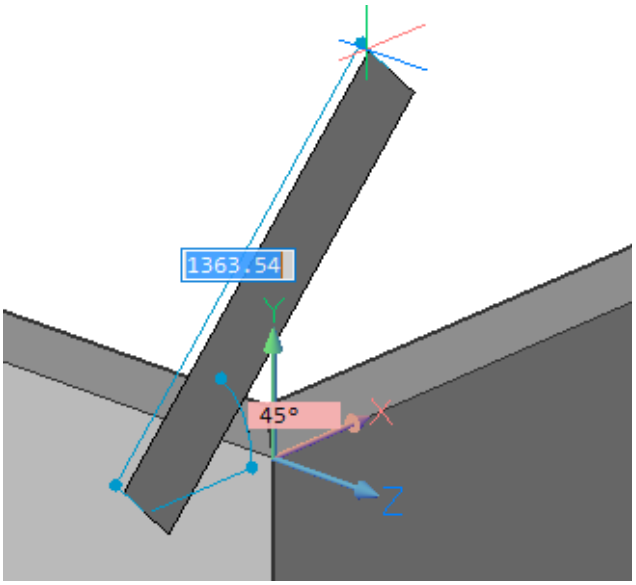
5 To create the roof at the top edge of the wall, snap to the exterior top corner of the wall.

To create a roof overhang (eave):

- Move the cursor over the exterior top corner of the wall to require a snap tracking point.
- When a small red cross indicates the tracking point is acquired, move the cursor to the outside and type the overhang distance in the dynamic entry field.



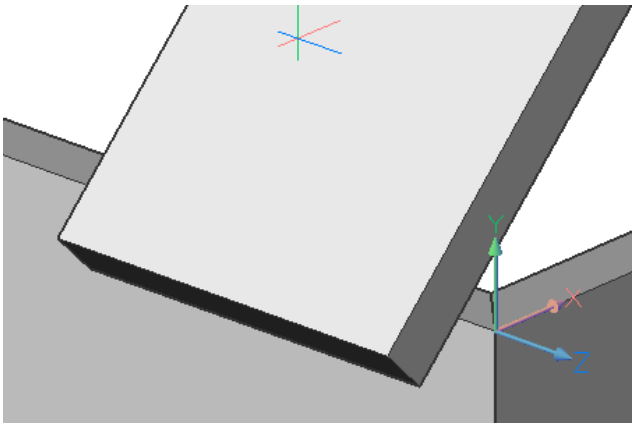
6 Hit the TAB key to activate the Angle field, then type the pitch angle and hit the TAB key again to jump to the Distance field.



7 Specify the height of the roof slab, then press Enter twice: the first Enter creates the footprint of the roof slab, the second Enter interrupts the creation of a second solid.

8 The roof slab is extruded dynamically.

9 Specify a point to define the length of the extrusion.



10 Repeat the previous steps to create more roof slabs.

11 Use CONNECTWITHNEAREST to connect the walls to the roof slabs.

20.3.8 Creating walls

20.3.8.1 Commands

POLYSOLID, LCONNECT, TCONNECT, BIMQUICKDRAW

20.3.8.2 About the Polysolid tool

The **Polysolid** tool allows you to create wall solids by selecting a start point, an endpoint and entering a height. Note that, in BricsCAD BIM, you always start by creating the geometry first and adding materials or compositions later. This way there is no need to worry about details when modeling a preliminary design.

The use of Dynamic Dimensions and Polar Tracking is recommended. Dynamic Dimensions can be used to specify the length of the wall and Polar Tracking controls the direction.

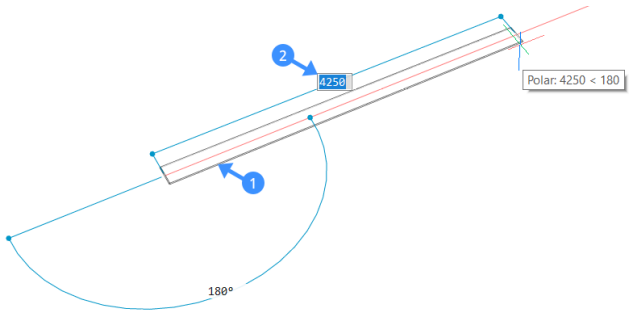
For more information about this command, visit the Command Reference article POLYSOLID.

20.3.8.3 Procedure: Create a simple wall

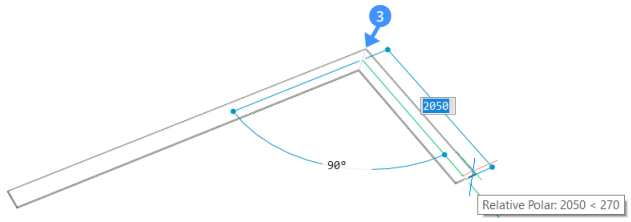
1 Launch the **Polysolid** tool.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

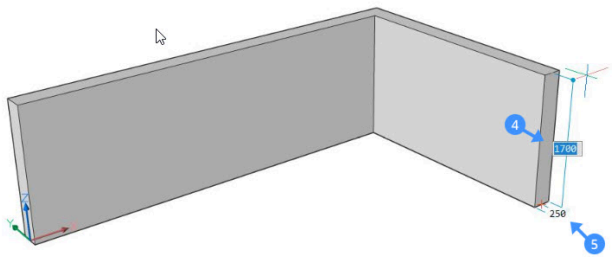
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



4 Select an endpoint or enter a value in the dynamic input field and press Enter. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.



5 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing Tab.



6 To change the justification of the wall hit CTRL while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.

Nota: The width and height of the previous wall will be the new default values of the next Polysolid.

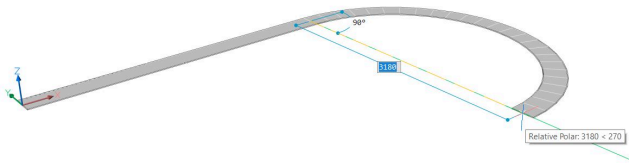
20.3.8.4 Procedure: creating a curved wall

1 Launch the **Polysolid** tool.

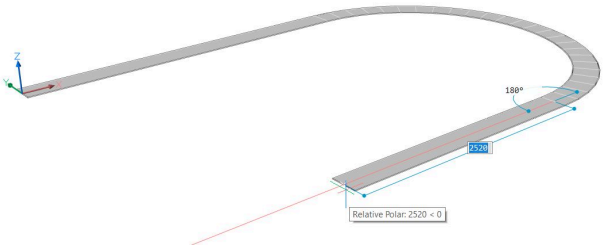
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press Enter.

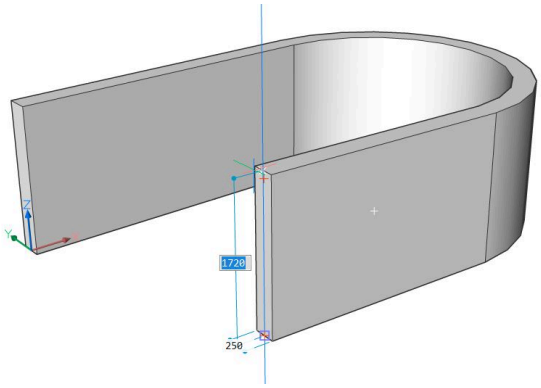


5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.



6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width

field. You can switch between the two dynamic input fields by pressing Tab.



20.3.8.5 Procedure: using an existing 2D plan to create walls

Using an existing 2D plan view enables you to create a wall element in 3D.

1 Click the **Polysolid** tool from the Quad and type E on the Command line or select **Entity** in the prompts menu. Using the **Entity** option in **Polysolid** enables you to create 3D geometry from a 2D layout.

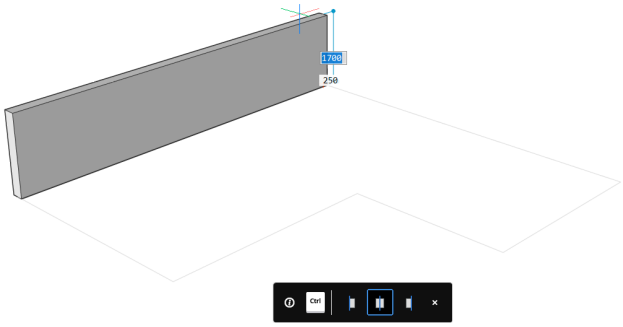
2 Choose a 2D linear entity which will be the Polysolid base.

3 Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the Width Field. Press Tab switch between the Height and the Width field.

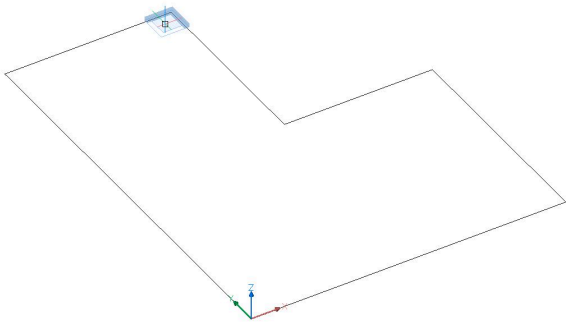
6 The Hotkey Assistant appears and displays the possible justification options. Press CTRL to cycle through the wall justification options.



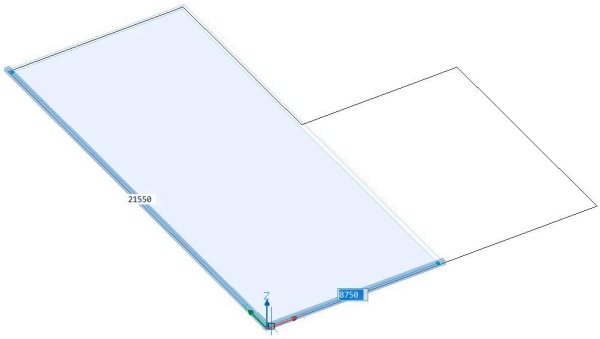
It is also possible to use an existing 2D plan to create walls using the **BimQuickDraw** tool:

1 Open the **Quickdraw** tool from the Quad or type Quickdraw in the Command line.

2 The cursor automatically snaps to the lines of your 2D drawing. Place your cursor where you want to place your first corner and left-click.



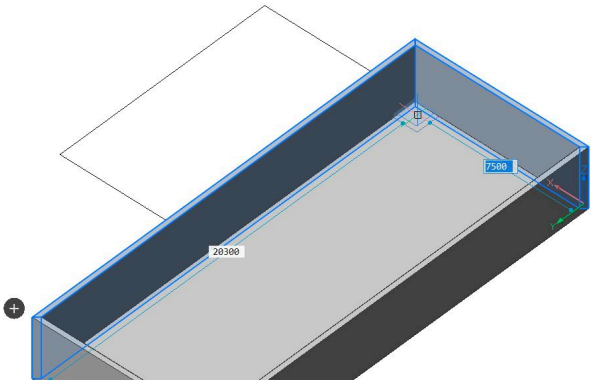
3 Hover your mouse to the opposite corner and left click again or type the values of the lengths of the sides in the respective fields and press Enter.



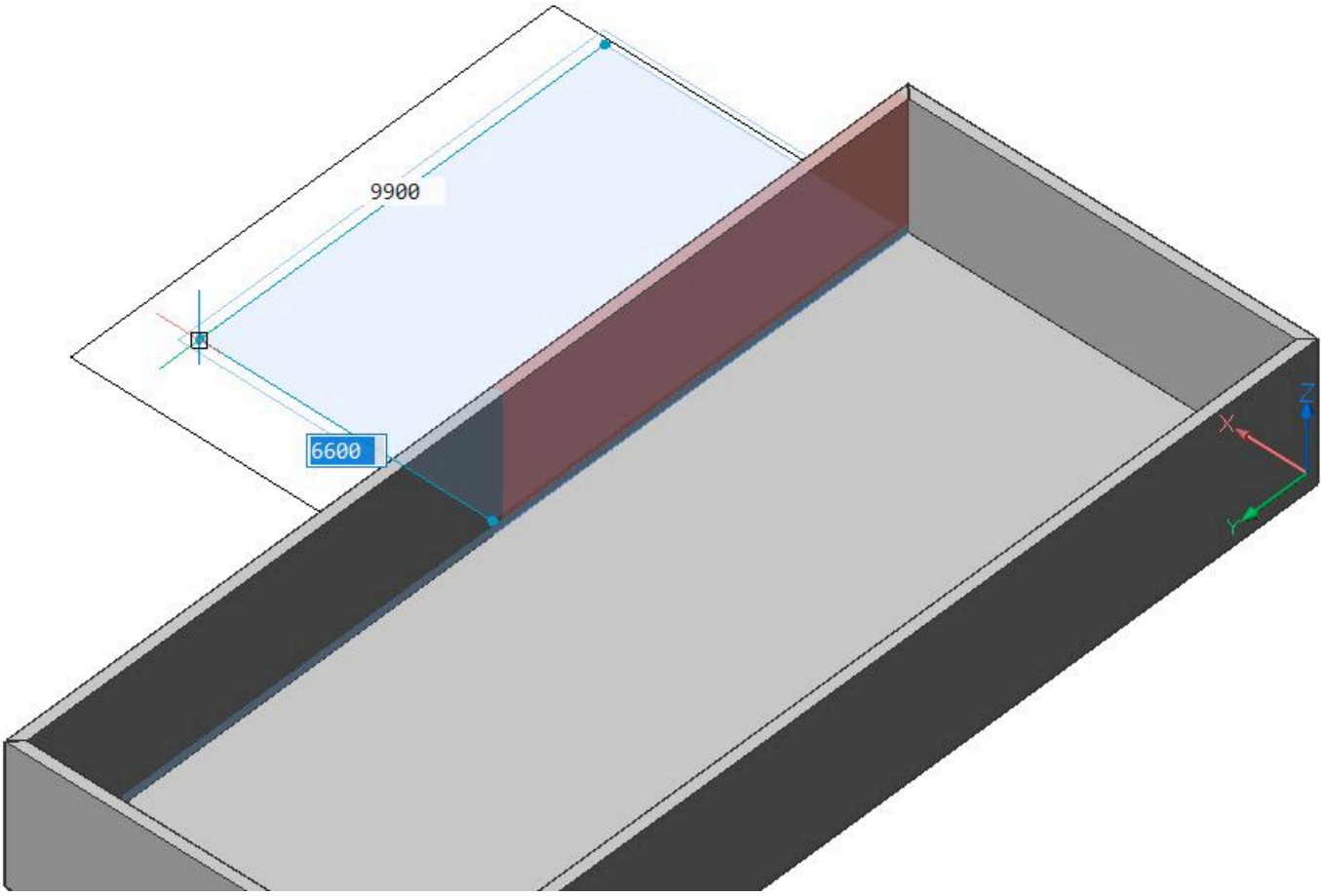
4 The **Quickdraw** tool always uses the default height and width settings.

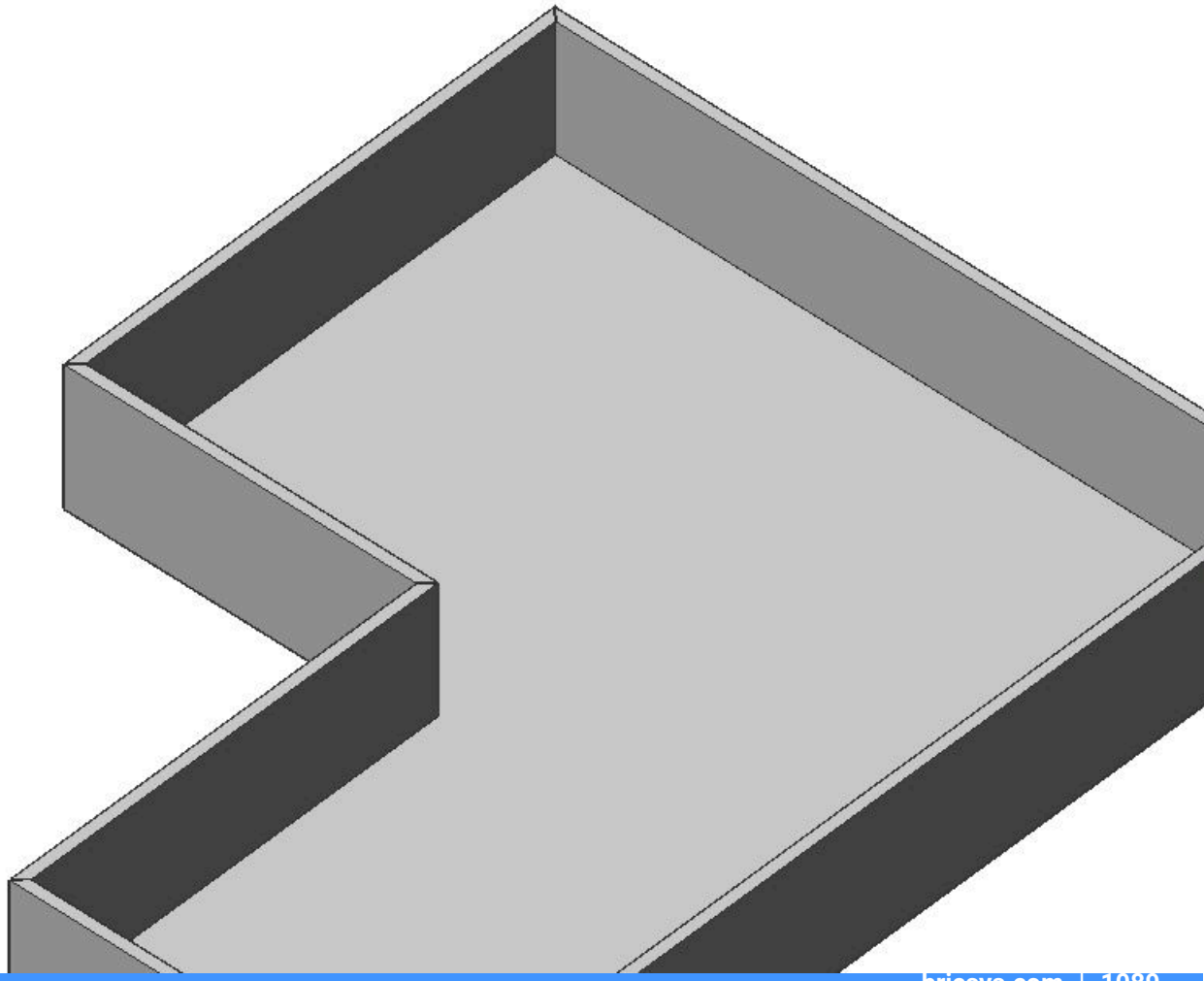
To change these values type S and press Enter to open the Quickdraw Settings. Change the values in this menu.

5 To finish the L-shaped form, you can draw a new rectangular on the second part of the figure. The common wall of the two rectangles will automatically be removed if you draw a second rectangular starting from the interior side of the first rectangular as indicated below:



6 While drawing the second rectangular, you can see the wall that will be deleted is displayed in red.





For more info about **Quickdraw**, please read the article Using BIM Quickdraw.

20.3.8.6 About the BimConnect tool

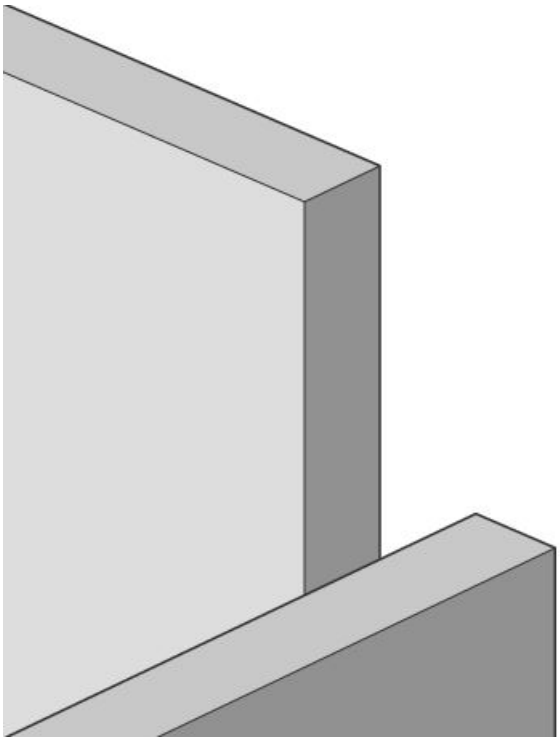
The **BimConnect** tool allows you to connect one wall to another element.

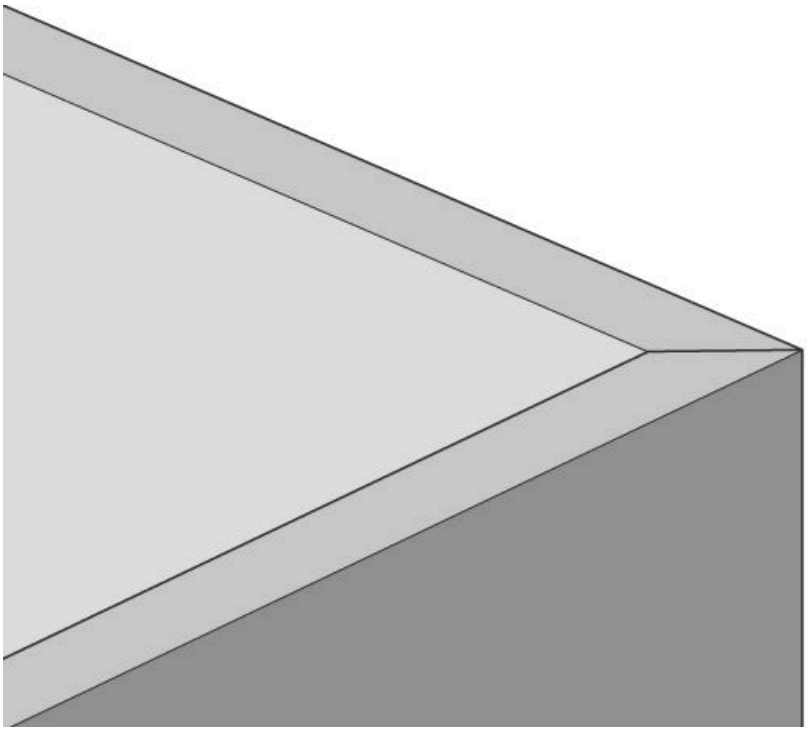
For more information about this command, visit the Command Reference article LCONNECT.

20.3.8.7 Procedure: connecting two walls

1 Select two wall elements and apply the **LConnect** tool.

When two walls intersect, a mitered connection will be created by default.





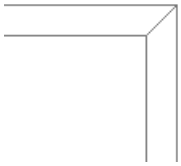
2 Change the layout of the wall connections using the ‘LConnect HotKeyAssistant widget’.

3 The selected connection type is indicated by a blue frame.

The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.
Switch between the options by pressing CTRL.

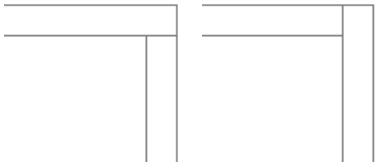
- **Bisector L-connection (miter)**

  When the **bisector L-connection** is selected, the walls are connected at an angle of 45°.



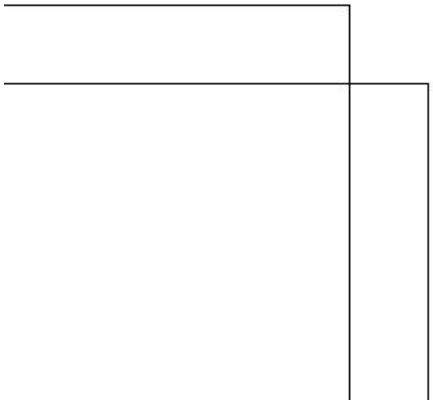
- **Parallel L-connection 1 Parallel L-connection 2**

  When the L parallel type 1 or type 2 connections are selected, the walls are connected at an angle of 90°.



- **Disconnection**

  When the disconnected option is selected, the existing connection between the walls is removed.



Press Enter to accept the current connection type or press CTRL to cycle through the possible connection options.

20.3.8.8 Connecting walls to other elements

The **Connect with Nearest** tool allows you to connect the minor face of a wall to the major face of another element, regardless of the orientation of the faces. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or the side face of a slab to a wall.

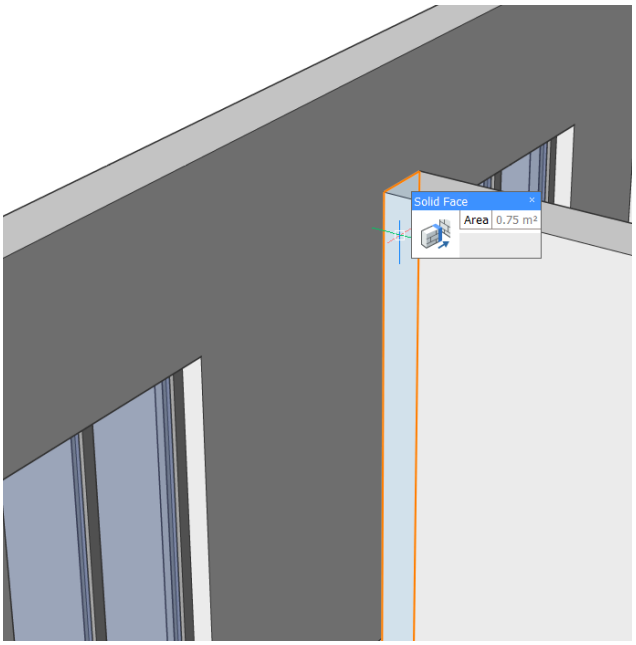
To connect the face to the nearest object, select the face of the object that you would like to connect with the nearest object and choose the Connect With Nearest tool in the Quad.

Nota: Before using the **Connect with Nearest** tool, make sure the **Select Faces** option is enabled in the selection mode settings.

For more information about this command, visit the Command Reference article TCONNECT.

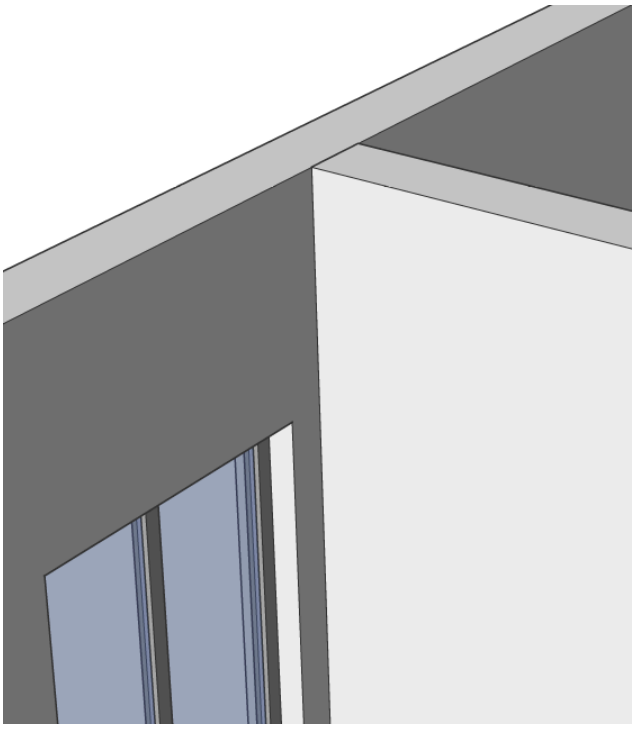
20.3.8.9 Procedure: creating T- connections between two walls

1 In the drawing area, highlight the face you want to connect to another wall.



2 Use the **Connect with Nearest** tool from the Quad.

The **Connect with Nearest** tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



20.3.8.10 About the select aligned faces tool

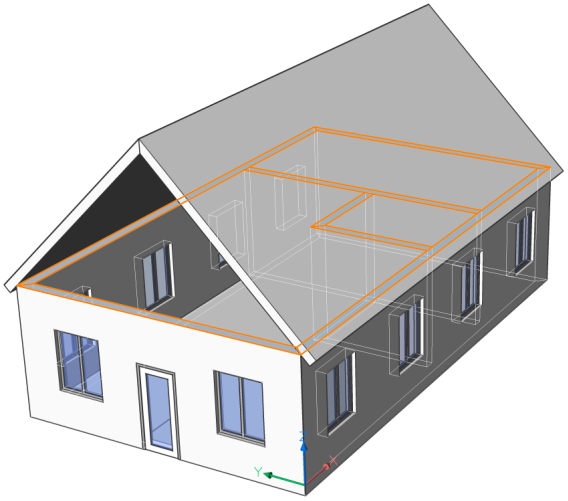
The **Connect with Nearest** tool can also be used to connect multiple faces of a wall to the nearest face of another element. To do this use the **Select aligned faces** tool. This tool enables you to highlight all faces that are coplanar with the current selection.

For more information about this command, visit the Command Reference article SELECTALIGNEDFACES.

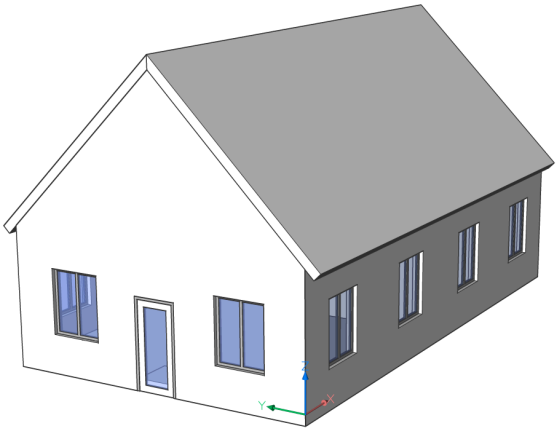
20.3.8.11 Procedure: connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click

the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



20.3.9 Curtain wall

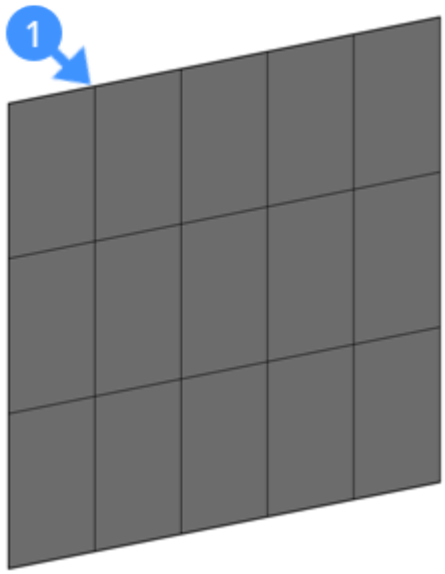
20.3.9.1 Commands

BIMCURTAINWALL

20.3.9.2 About

In BricsCAD, curtain walls are created from the face of a 3D solid or surface as a block by using the **Curtain Wall** tool.

- Curtain walls can be created from any planar and curved surface.
- Window Frames are placed on a BIM Grid when creating a Curtain Wall.
- Changing the U-line and V-line panel options allows you to specify the desired size and number of panels on the planar or curved surface.



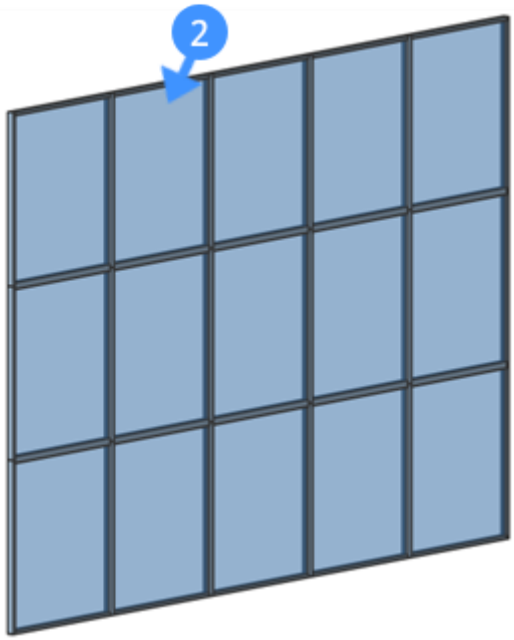


Figure 1, Curtainwall grid (1).

Figure 2, Curtainwall with glazing (2).

The curtain wall system consists of panels of glass. This is a panel infill for a curtain wall.

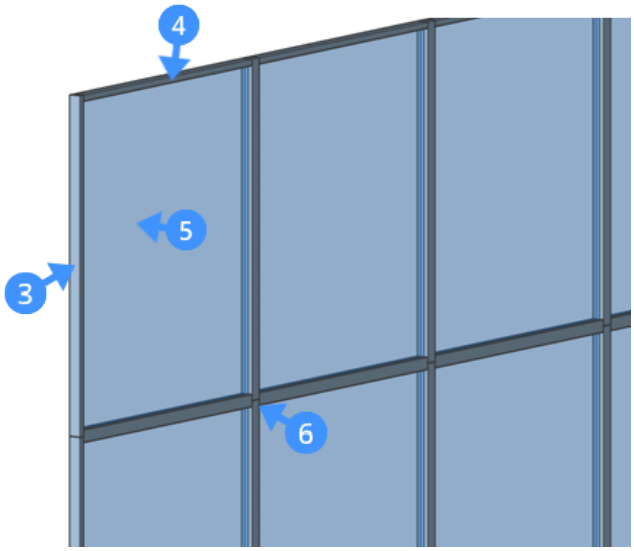
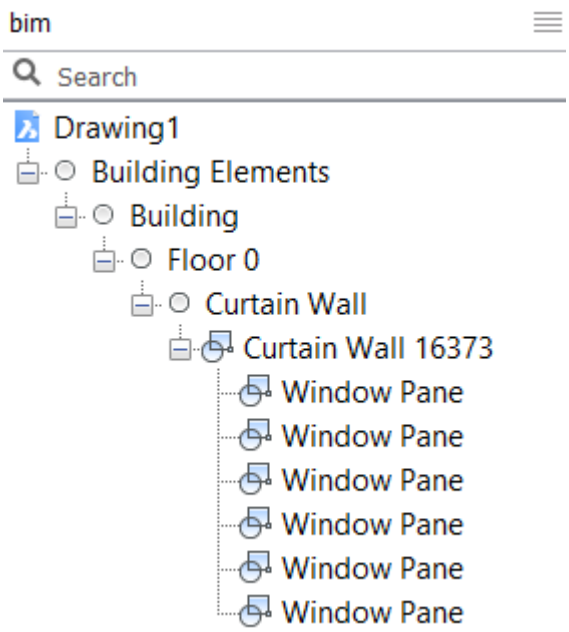


Figure 3, A curtain wall. The frame that coincides with the grid vertical (U) panels (3), the frame that coincides with the grid horizontal (V) panels (4), and glass in-fill (5) are shown. By default, the straight connection (6) is made between the beams.

Curtain walls are automatically classified as a **Curtain wall** in the **Structure browser**.

If you explode the Curtain wall, the sub-entities are classified as windows and as frames. The frames are linear solids. This allows you to calculate the length of a curtain wall.



20.3.9.3 Procedure: creating a curtain wall

1 Launch the BIMCURTAINWALL command in the Command line.

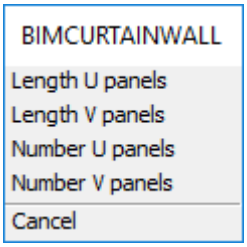
1 You are prompted: Select a face:

2 Select a face in the drawing area to create a curtain wall.

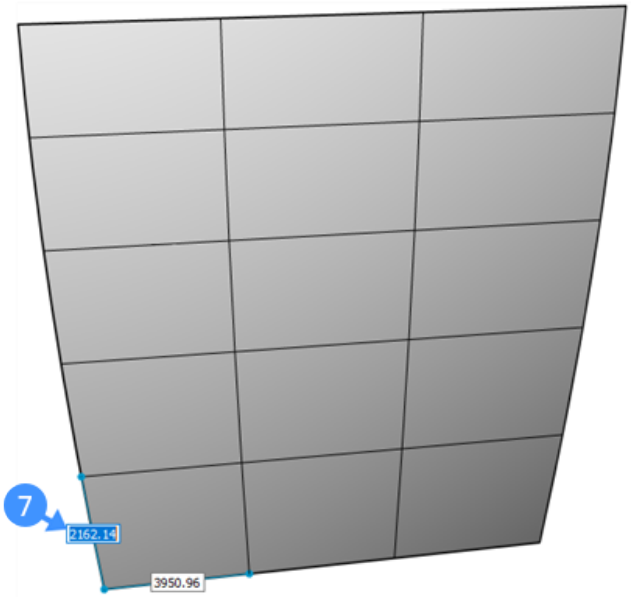
3 The curtain wall grid is automatically generated on this face.

You are prompted: Create grid or [Lenght U panels/Length V panels/Number U panels/Number V panels]:

4 Change the length of the U or V panels and number of U or V panels using the prompt menu and the Command line.



Or, if Dynamic dimension (DYN) is on, the length of the panels can be changed using the DYN fields (7).



5 Hit the TAB key to switch between dynamic dimension boxes.

6 Or Press Enter to accept the current value.

Nota: The maximum deviation from planarity is displayed in the command line.

Optional: If the value of the max deviation from planarity is different to "0", the prompt menu (8) will appear and the Command line prompts a message (9).

The prompt menu and the Command line allow you to planarize the grid cells.

Click "Yes" to planarize the grid cells. By default, the target deviation from planarity is set to "0.00001".

7 Press Enter to accept the default value.

8 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

The prompt menu:

(**Optional**) To change the frame width, select the **Width** option in the prompt menu or type "W" in the Command line.

Default frame width = 60.000000

(**Optional**) To change the frame depth, select the **Depth** option in the prompt menu or type "D" in the Command line.

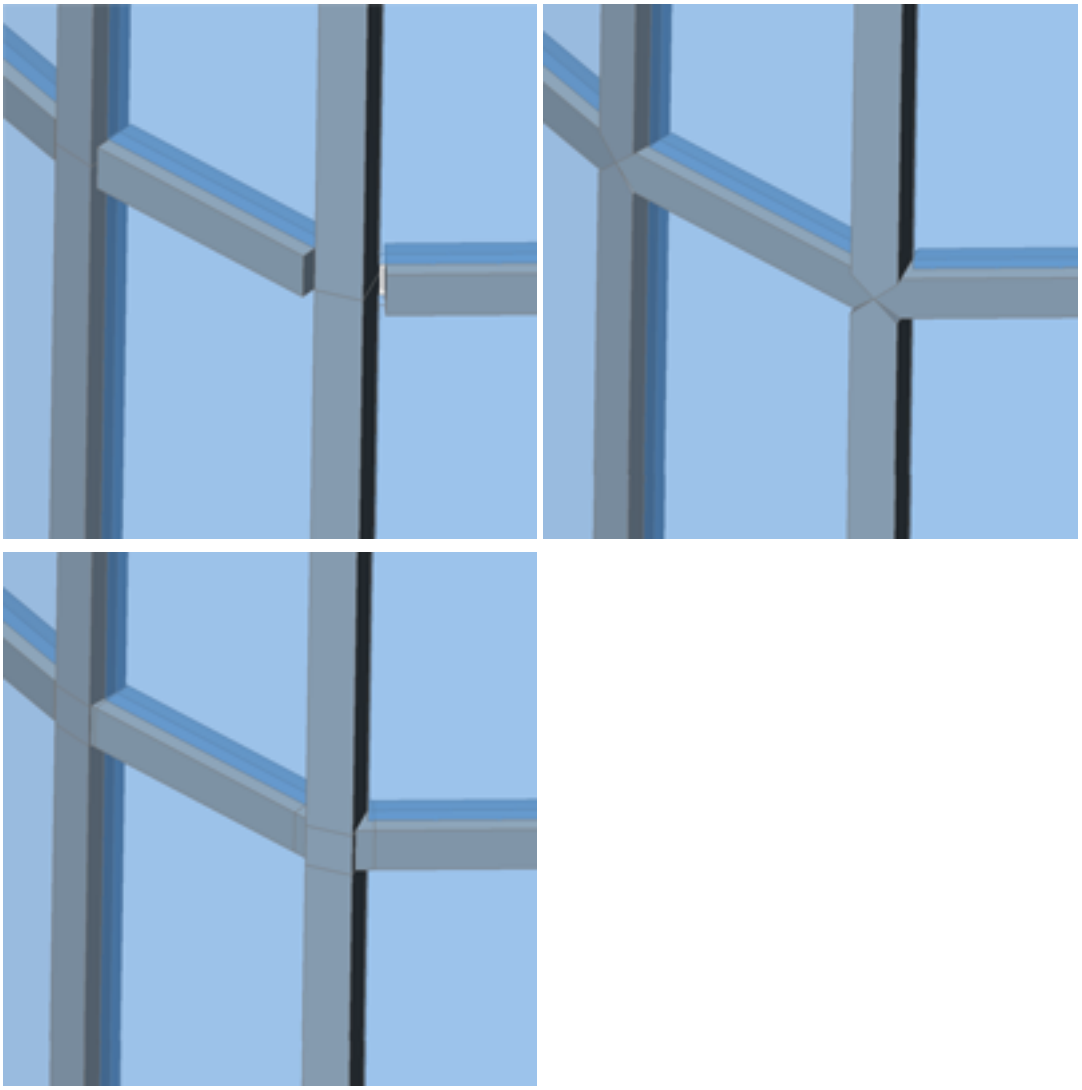
Default frame depth = 60.000000

(**Optional**) To change the glass thickness, select the **Glass thickness** option in the prompt menu or type "G" in the Command line.

Default glass thickness = 20.000000

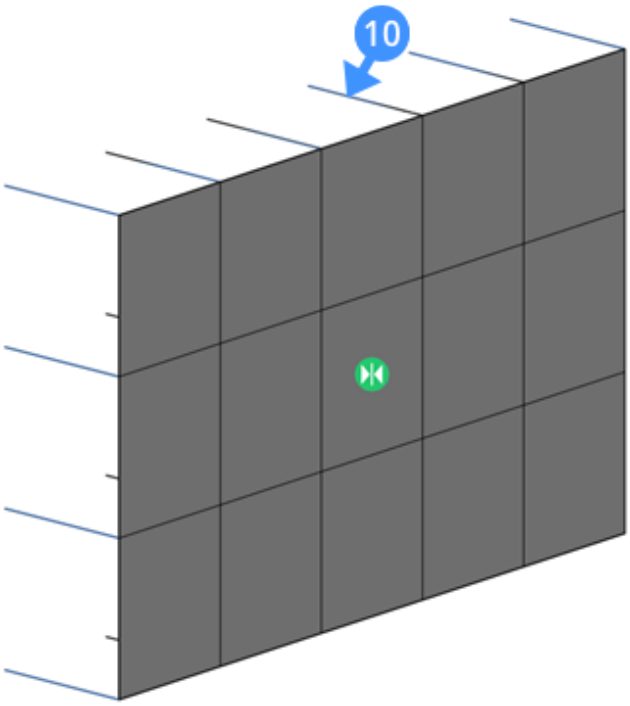
(**Optional**) To choose a connection type between, select the **Connections type** option in the prompt menu or type "C" in the Command line. For smooth connections, select the **Smooth connection** option or type "S" in the Command line. For nodes connection, select the **Nodes connection** option or type "N" in the Command line.

The default connection is straight.



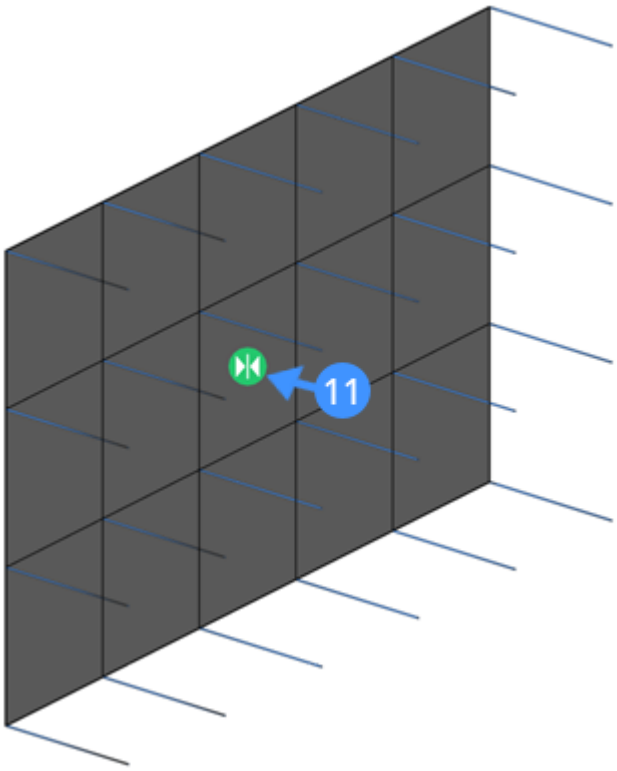
There are three types of connection. From left to right: Straight - Smooth - Nodes.

9 Blue lines (10) appear on the face to show the side a curtain wall will be created.



Nota: Step 6 and step 7 are skipped when the face is selected from a solid. In this case the inside normal is chosen by default.

10 Use the widget (11) to flip the curtain wall.

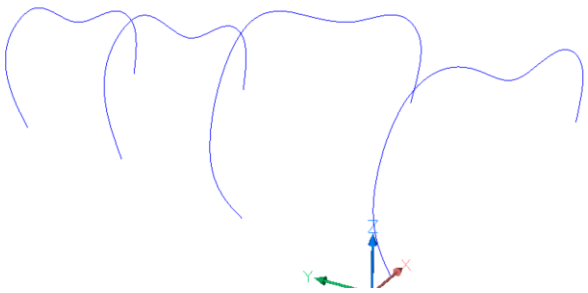


11 Press Enter to accept.

Nota: The surface or solid used to create the curtain wall will be deleted by default. If you don't want this, set the **DELOBJ** value to '0' in **Settings**.

20.3.9.4 Procedure: Creating a curtain wall from a free form surface

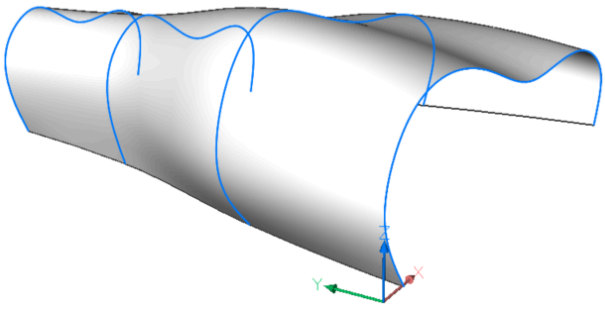
1 Start with 4 splines.



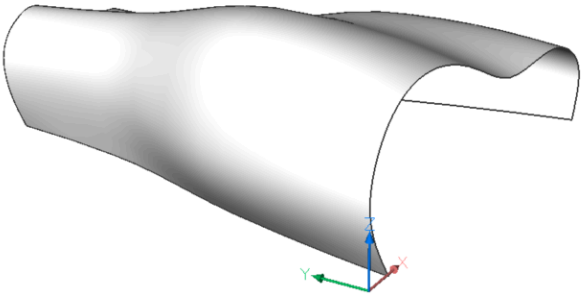
2 Launch the LOFT command in the Command line.

3 You are prompted: Select cross-sections in lofting order [MOde]:

Select the 4 splines in lofting order.



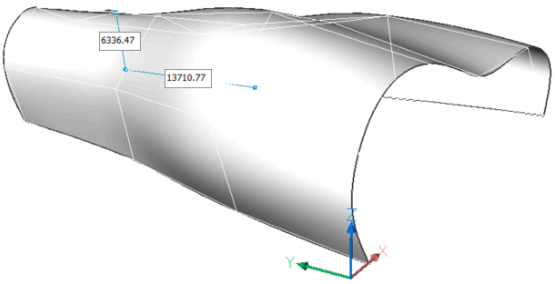
4 Press Enter twice to end the command.



5 Launch the BIMCURTAINWALL command in the Command line.

6 You are prompted: Select a face

Select the lofted surface.



7 A curtain wall grid is automatically generated from this surface.

8 You are prompted: Create grid or [Lenght U panels/ /Length V panels/Number U panels/Number V panels].

Change the number of the U and V panels in the prompt menu or the Command line. In this example: U = 20 and V = 8.

9 You are prompted: Planarize [Yes/No] <Yes>

10 Click "Yes" to planarize the grid cells.

11 You are prompted: Enter target deviation <0.00001>:

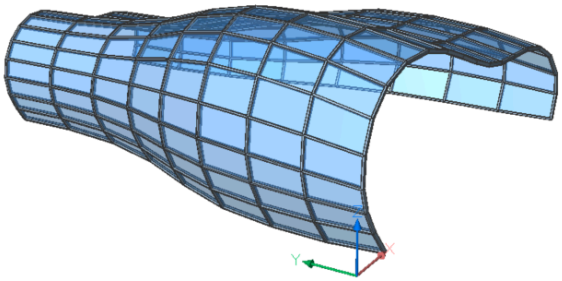
12 By default, the target deviation from planarity is set to "0.00001".

Press Enter to accept it.

13 You are prompted: Change parameters or accept [Accept/Width/Depth/Glass thickness/Connections type] <Accept>:

14 Change the parameters to:
- Width: 150
- Depth: 300
- Glass thickness: 20
- Connection type: Smooth

15 Press Enter to accept. A curtain wall is created.



20.3.10 DmExtrude

20.3.10.1 Commands

DMEXTRUDE

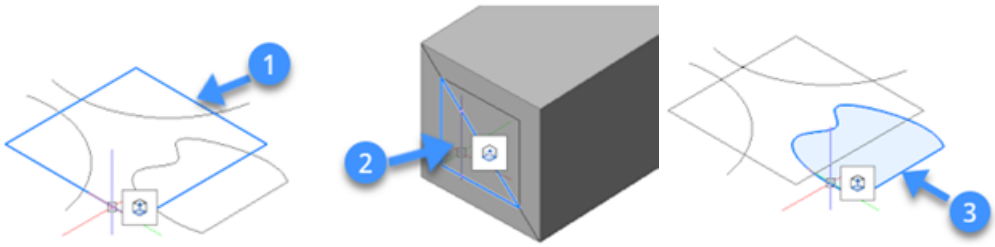
20.3.10.2 About

The **Extrude** tool allows you to create 3D solid(s) by extruding closed 2D entities, faces of 3D solid(s), regions or closed boundaries. The **Extrude** tool enables you to cut the solid model partially by removing the volume from the main solid and it can also create new solid(s) by adding volume to the main solid model.

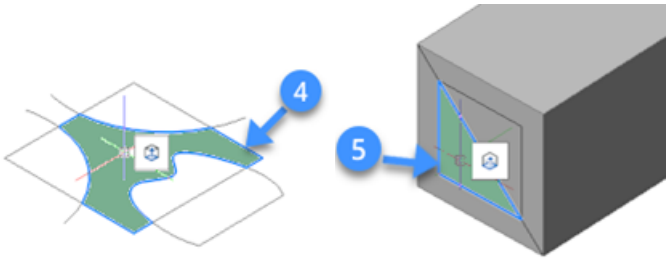
For more information about this command, visit the Command Reference article DMEXTRUDE.

The following illustration shows the selected entities to be extruded.

Closed entities (1) (2) are shown. A region object (3) is shown in the figure.



The following illustration shows a closed boundary in the XY-plane of the current coordinate system (4) and the closed boundary on the face of the solid (5) is highlighted while hovering over it.



The closed boundaries are recognized automatically in BricsCAD, when the **Enable Boundary Detection** option on the **Selection modes** toolbar is active.

Nota: As a closed entity only closed polylines, circles, and ellipses are accepted.

20.3.10.3 Procedure: using DMEXTRUDE in solid mode

The following are general steps to add and remove volume from the solid(s) using the **Extrude** tool.

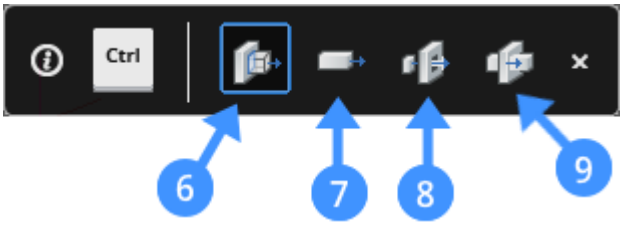
1 Hover the cursor over an entity or sub-entity in the drawing area.

2 Select the **Extrude** tool in the **Model** tab from the Quad.

You are prompted: Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>.

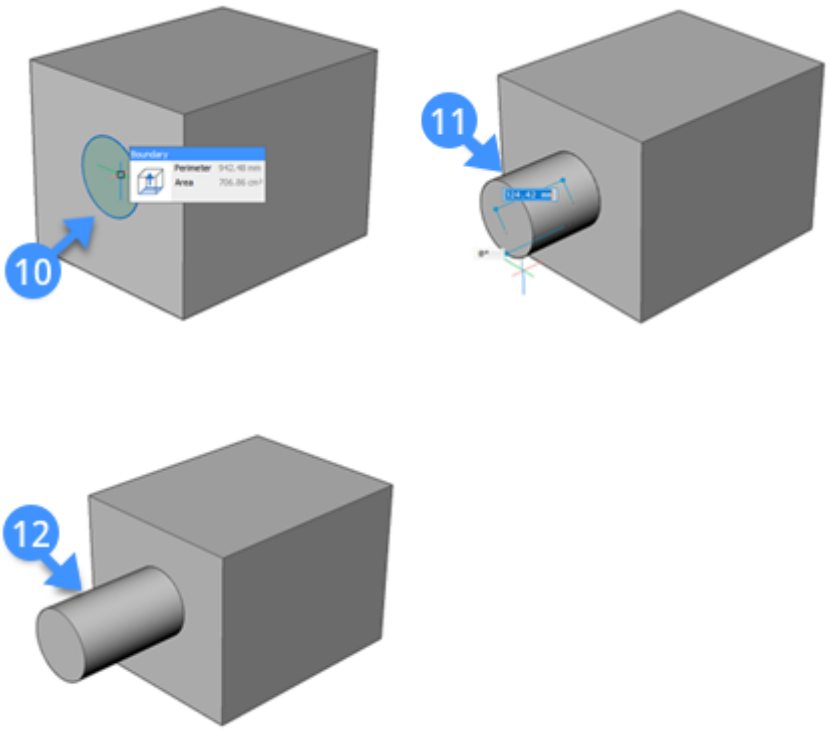
3 The “DmExtrude Hotkey Assistant” appears at the bottom of the screen with a set of extrusion options.

Nota: The default mode is indicated by a blue frame.

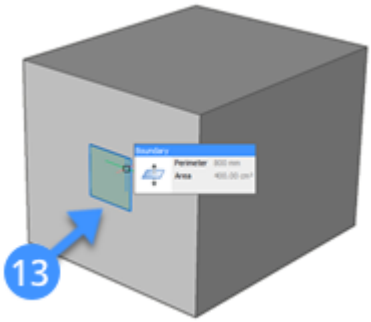


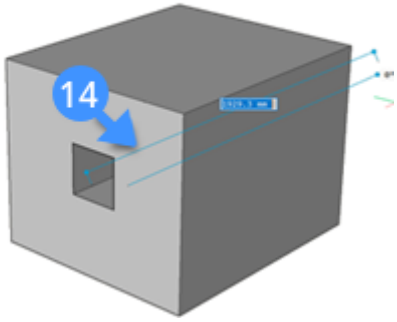
- **Auto (6)**: The result of the auto option depends on the extrusion direction of the 3D solid and the value of the DMEXTRUDEMODE system variable value. Note: By default, the value of the DMEXTRUDEMODE is set to “3”. The following illustration shows the extruded entity from the main solid while the default value is set to “3”. When you highlight an entity (10) and extrude it

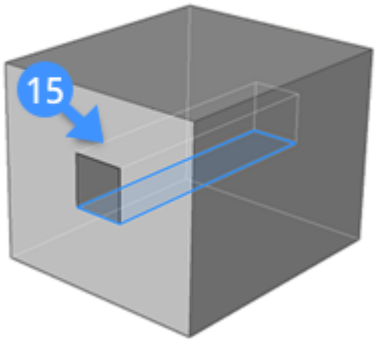
to the outward (11), a new volume is added (12) to the main solid model.



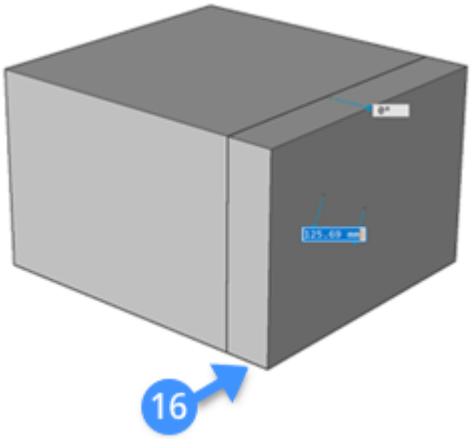
The following illustration shows the subtracted entity from the main solid. When you highlight an entity (13) and extrude it to the inward (14), a volume is subtracted (15) from the main solid.





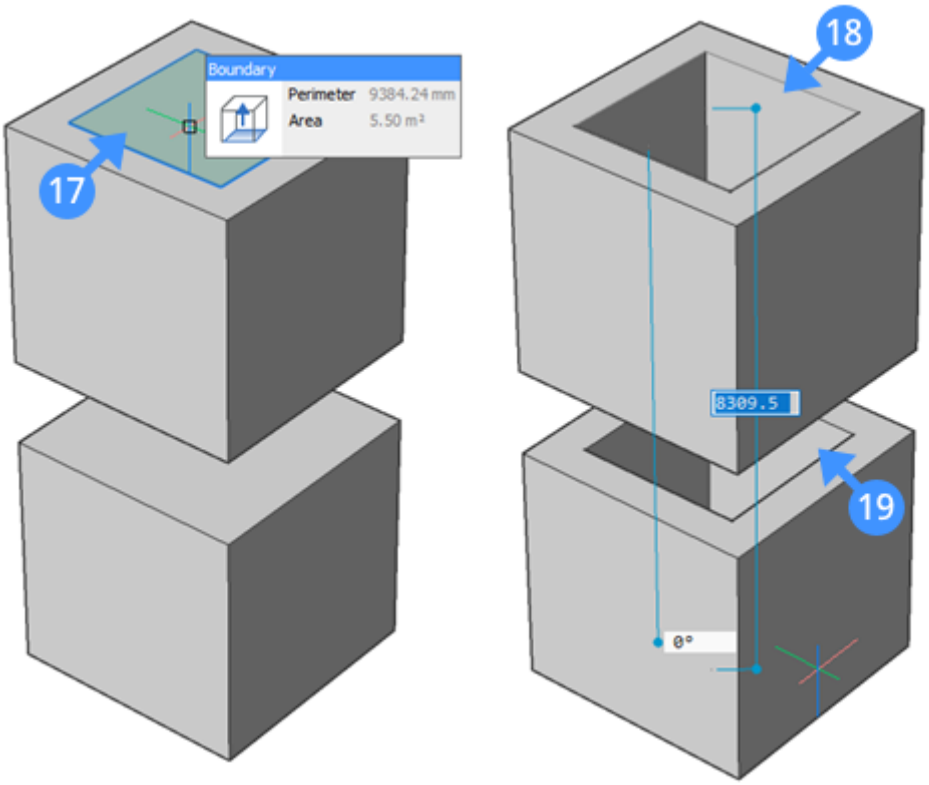


- **Create (7)**: Regardless of the extrusion direction, a new volume is created (16) from the existing solid.



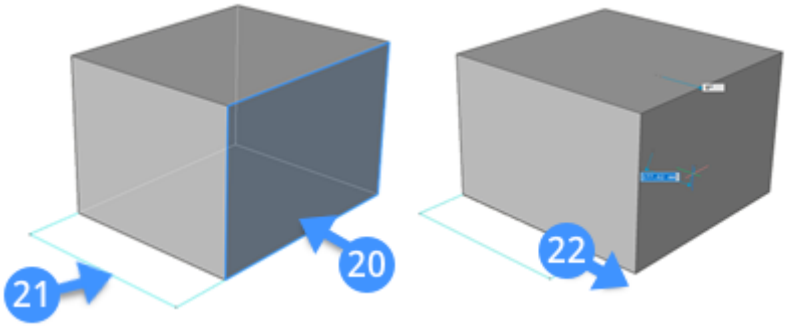
- **Subtract (8):** The 3D solid is subtracted from each interfering existing solid.

  The following illustration shows the subtracted solids (18,19) using the highlighted boundary (17).

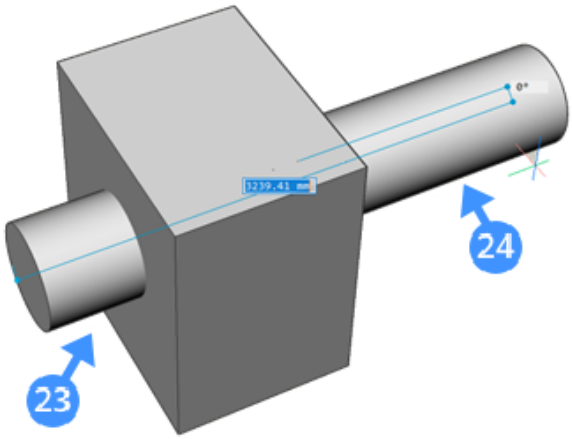


- **Unite (9):** The new volume is unified with each interfering existing solid.

  The following illustration shows the before and after when using the extrusion tool unite option. The solid surface is highlighted (20) and the distance between the left and right face is shown with the aligned line (21). The new volume is unified (22) with the first solid.



- **Both sides**: New symmetric volumes are created (23), (24) in both directions from the main solid.



To create symmetric volumes from the existing solid, type “B” on the the Command line which corresponds to Both sides option of the **Extrusion** tool.

- Select one of the available options to operate the extrusion function on your solid.
- Next, move the cursor in the desired direction to specify the extrusion direction. The selected entity or sub-entity is extruded dynamically.
- Define the height of extrusion by specifying a point in the drawing, entering a value in the dynamic dimension field or choose the limit option.
- Press Enter or right-click to accept the extrusion from your existing solid.

20.3.10.4 DMEXTRUDE in Surface Mode

The **extrude** tool allows you to create 3D surface(s) by extruding 2D entities such as lines or polylines.

For more information about this command, visit the Command Reference article DMEXTRUDE.

20.3.10.5 Procedure: using the extrude tool in surface mode

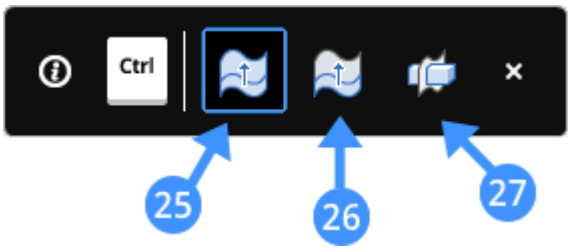
The following are general steps to create 3D surface(s) using the **Extrude** tool in the Surface mode.

1 Move the cursor over an entity or sub-entity in the drawing area.

2 Select the **Surface Extrude** tool in the **Model** tab from the Quad.

You are prompted: Specify height of extrusion or set [Auto(subtract or create)/Create/Subtract/Unite/Both sides/Taper angle/Direction/Limit] <Auto>.

3 The “DM Extrude Surface Hotkey Assistant” appears at the bottom of the screen with a set of surface extrusion options.



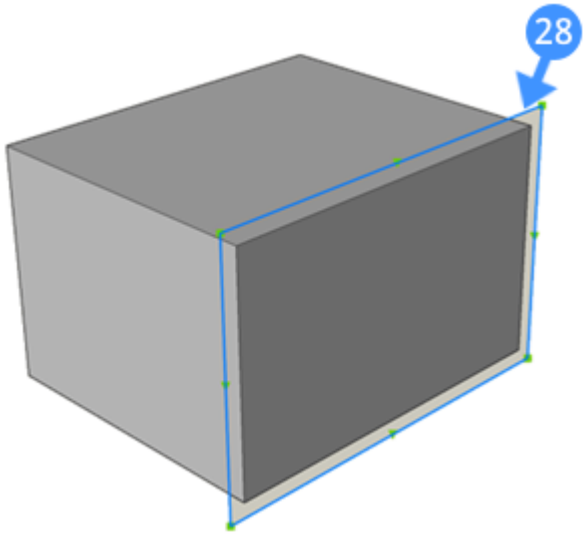
Nota: The default mode is indicated by a blue frame.

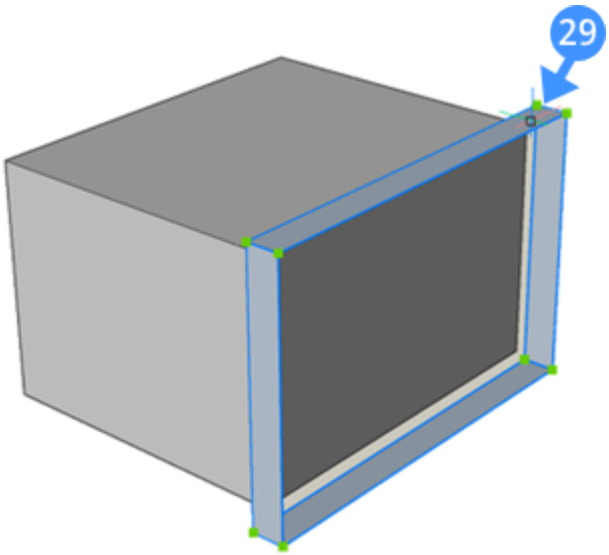
- **Auto (25):**Select either a surface or the face of a solid element. When the surface's edge is used as a contour for extrusion, the created surface is stitched with the base surface. When the surface's face is used for extrusion, the surface results in not stitched with the base surface.
- **Create (26):**Create new surfaces.
- **Slice (27):**Cut through solids with the extruded surface. This option is much like the SLICE command.

4 Select one of the options and move the mouse to give the extrusion direction, entering a value in the dynamic dimension field.

5 Press Enter or right-click to accept the extrusion for the surface.

6 The following illustrations show the created surface (29) by extruding the selected 2D entity (28) in the drawing.





20.3.11 DmPushPull

20.3.11.1 Commands

DmPushPull

20.3.11.2 About

The **Push/pull** tool allows you to add and remove volume from solids by moving highlighted faces.

Nota: **Push/pull** tool applies to planar, cylindrical, spherical, conical and toroidal faces of a solid.

For more information about this command, visit the Command Reference article DMPUSHPULL.

20.3.11.3 Procedure: using Push/Pull tool to add or remove volume to the solid

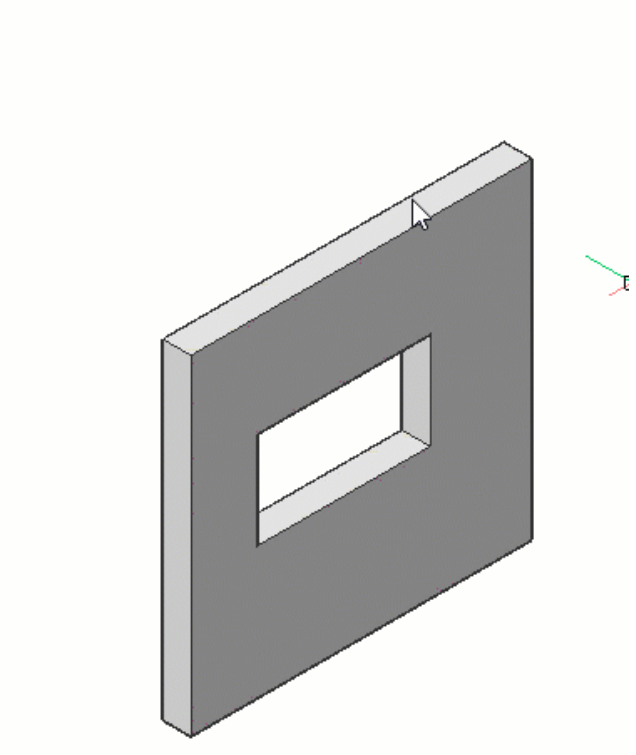
The following are general steps to add or remove volume from a solid(s) using the **Push/Pull** tool.

1 Highlight any face(s) of the solid(s) in the drawing.

2 Select the **Push/pull** tool in the **Model** tab from the Quad.

3 Move the cursor to pull the highlighted face.

4 The following movie shows the highlighted face behavior. The selected face moves dynamically, adding or removing volume.



5 Specify a point in the drawing or enter a value in the dynamic dimension field, then right click to accept changes.

20.3.12 Editing grids

20.3.12.1 Commands

BIMGRID, BEDIT, NUMBER, PROPERTIES

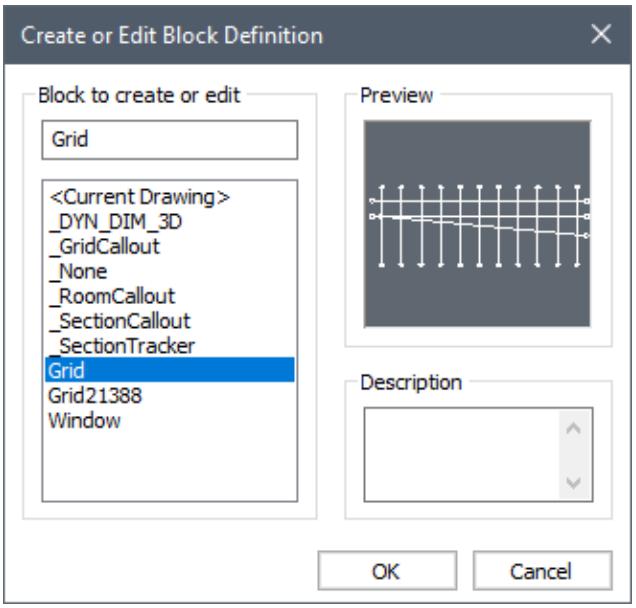
20.3.12.2 About

BIM grids are created as a block. The BEDIT (Block Edit) command allows to edit a BIM grids.

20.3.12.3 Editing a rectangular grid

1 Double click the grid.

1 The **Create or Edit Block Definition** dialog box displays.



2 Select the grid, then double click or click the OK button.

3 A **Block Edit** session is started.

4 Edit the grid axes.

5 You can move, rotate, copy, stretch, grip-edit or delete grid axes.

6 Renumber grid axes or edit axis labels if necessary (see the procedures below).

7 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

20.3.12.4 Renumbering the grid labels

Renumbering grid labels is needed:

- If you want a different numbering style.
- When grid axes are added (copied) or deleted.

Do the following:

1 Double click the grid to start a **Block Edit** session.

2 Select all vertical axes.

3 Start the NUMBER command and choose the appropriate number style.

4 Make sure to overwrite the existing numbers.

5 Repeat the previous steps for the horizontal axes.

6 Click the **Bclose and Save** icon ( ) on the **Block Edit** toolbar.

20.3.12.5 Editing a grid label

1 Double click the grid to start a **Block Edit** session.

2 Double click an axis.

3 The properties of the axis entity (line or arc) display in the **Properties** panel.

4 Select the **BIM/Axis number** property.

5 Type a value or name.

6 Click the **Bclose and Save** icon () on the **Block Edit** toolbar.

20.3.13 Inserting and editing windows and doors

20.3.13.1 Commands

BIMINSERT

20.3.13.2 About

In BricsCAD, the **Insert BIM Component** tool allows you to insert windows, doors and other components, such as furniture, sanitary equipment, plants, etc. in any face of a 3D solid and is typically used to insert windows and doors.

For more information about this command, visit the Command Reference article BIMINSERT.

20.3.13.3 Controlling the position of the window or door

To control the position of the window in the face of the solid (wall), you can use the following drawing aids:

- **Dynamic Dimensions**: During placement of a window or door on a 3D Solid face, dynamic dimensions are created, starting from the middle of each edge of the bounding rectangle of the opening to the nearest parallel edge on the face of the 3D Solid. Make sure **Dynamic Dimensions** are active by checking the DYN field in the status bar and **Dynamic UCS** is active by checking the DUCS field in the status bar.

  The **Dynamic UCS** aligns the window to the face of the solid. The edge of the face by which the cursor enters the face defines the orientation of the X-axis. Hit the Shift key to temporarily lock the UCS, which allows to use reference points outside the face of the solid. Hit the shift key again to unlock.
- **Temporary Tracking points**: Click the mouse wheel or press the **Temporary Tracking Points** tool button ( ) on the **Entity Snaps** toolbar to start specifying temporary points.
- **OSNAPZ**: If this system variable is ON, the Z-value of any entity snapping point is replaced by the current value of the ELEVATION system variable. If ELEVATION=0, entity snapping points are forced to lie in the face of the solid. OSNAPZ is toggled by the Ignore entity snap elevation tool button ( ) on the **Entity Snaps** toolbar. The state of the tool button (pressed or not) indicates the current value of **OSNAPZ**.

- **Entity snap to negative Z-values**: Press the **Entity snap to negative-z** tool button ( ) on the **Entity Snaps** toolbar to enable entity snaps on points that lie behind the selected face. If **OSNAPZ** is ON, the point is projected onto the selected face.

20.3.13.4 Procedure: inserting windows or doors

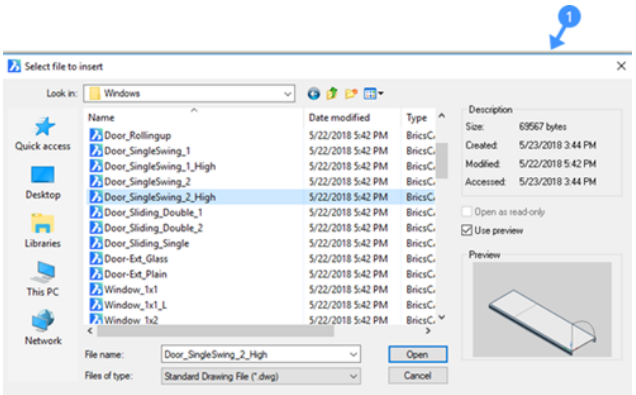
Nota: Make sure **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

Using the BIMINSERT command

1 Select **Insert BIM Component**from the **Model** tab of the No Selection Quad.

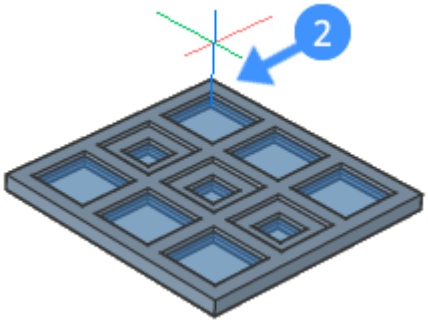
The file explorer opens, displaying the content of the most recently used folder to insert a BIM component. The default folders for BIM components sit in the following subfolder of the BricsCAD installation folder: ...\UserDataCache\Support\en_US\Bim\Components.

2 Select the **Windows** (1) or **Doors** folder.



3 Select a window or door you want to insert in your drawing, then double click or click the **Open** button.

The selected insert is attached to the cursor (2).



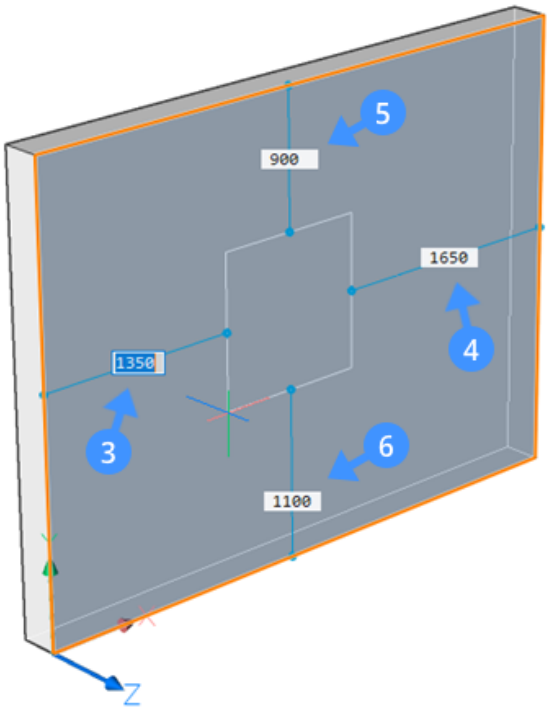
The parameters of the insert display in the **Properties** panel.

Prompts you: Select insertion point or [Edit inserted entity/Rotate component/set Base point/Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

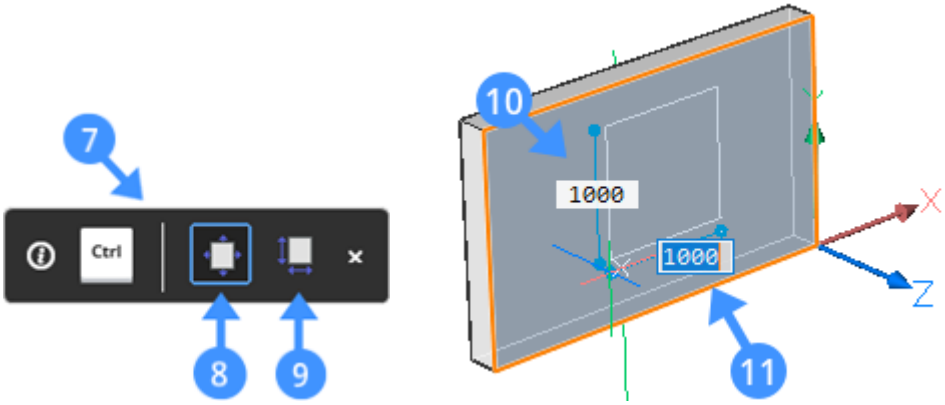
4 Optionally, edit the parameters in the **Properties** panel or choose an option (see Command Options of the BIMINSERT command).

5 When you move over a face of the solid, the insert aligns with the face under the cursor. The distances

from the start (3), end (4), top (5) and bottom (6) face of the wall to the placement rectangle of the insert display dynamically.



If the HKA (Hot Key Assistant)* is ON, the HKA Widget (7) displays at the bottom of the screen to show you the available options. Use 'pick position' (8) option to adjust the insertion point of the wall or door. Use the 'change parameters' option (9) by hitting the CTRL key to specify the width and height of the insert. The Width (10) and Height (11) field will indicate the current dimensions of the insert.

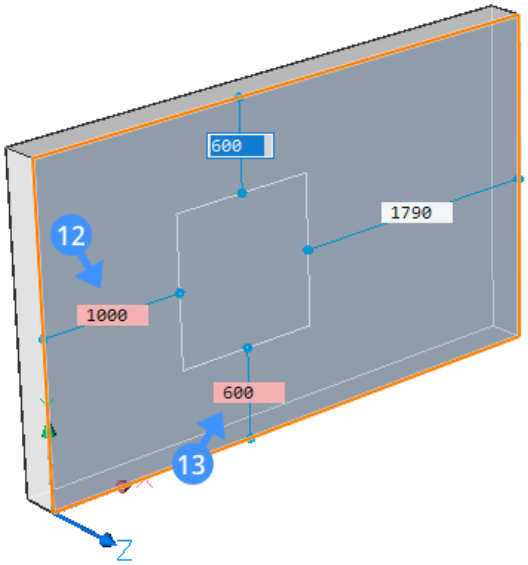


* Click the HKA field in the Status bar to toggle the display of the Hot Key Assistant.

6 Do one of the following to place the selected window or door using the dynamic dimension fields:
- Hit the TAB key to select one of the dynamic dimension fields, then type a value and hit the TAB key to lock the dimension.
- The locked dimension turns red (12).
- Hit the TAB key to select one of the other dynamic dimension fields, then type a value and hit

the TAB key to lock the dimension.

- The locked dimension turns red (13).



Using the components panel

See: Insert a window from the Components Panel into your drawing.

20.3.13.5 Procedure: editing the parameters of an insert

If parameters are defined in the window or door source drawing, these parameters can be used to edit each detail of a window/door separately.

1 Select any door or window in the drawing. The current values of the various parameters display in the **Parameters** section of the **Properties** panel. In the following illustration, window parameters are shown in the **Properties** panel.

Parameters	
Name	Window_3x2:1
File	
W	1000 mm
H	1000 mm
Rebate	0 mm
PlacementDepth	90 mm
FrameThickness	60 mm
FrameDepth	60 mm
PaneDepth	20 mm
PaneOffset	20 mm
MullionThickness	60 mm
TransomThicknes	60 mm

2 Select a parameter you want to change, then type a value in the settings field.

20.3.13.6 Moving an insert

In BricsCAD, there multiple methods to move an insert in the face of a solid:

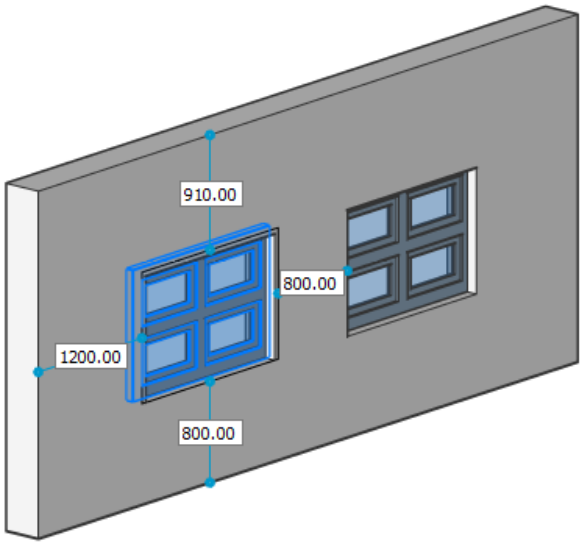
- Using Distance fields.

- Using **Push/Pull**
- Using the Manipulator

Procedure: using distance fields

1 Select the insert in the model.

1 Current distances from the insert to the nearest parallel faces of the parent solid display:

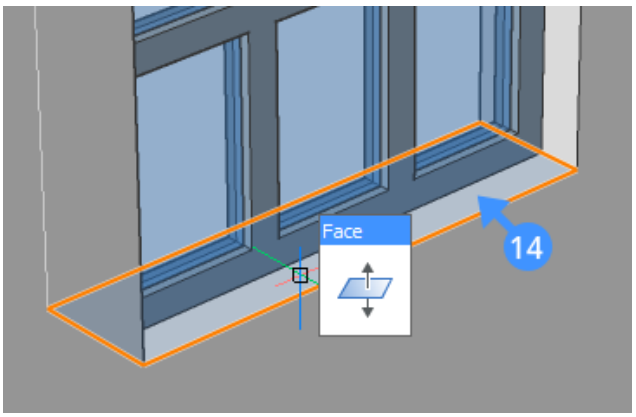


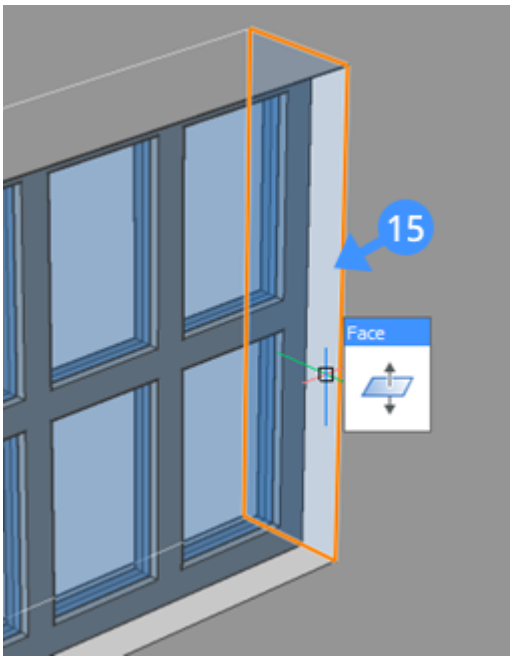
2 Double click a distance field, then type a value and press Enter.

Procedure: using Push/Pull to move an insert

1 Hover the cursor over the top or the bottom inner face of the opening (14) made by the insert to move the insert vertically. Or to move the insert horizontally, hover the cursor over the left or the right inner

face of the opening (15) made by the insert.





2 When the face highlights, select **DmPushPull** from the **Model** tab of the Quad.

The insert will move vertically if the top or bottom inner face is highlighted.

The insert will move horizontally if the left or right inner face is highlighted.

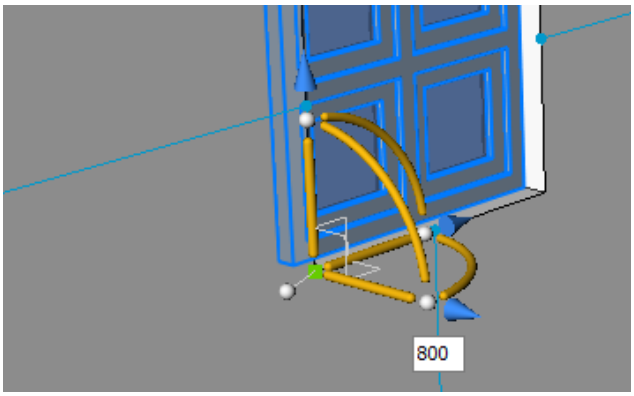
3 Do one of the following to complete the moving insert process.

- Specify a point.
- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

Procedure: using the Manipulator

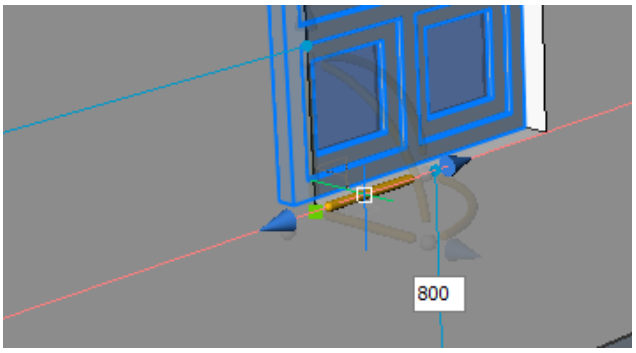
Select the insert by holding the left mouse button down a bit longer. Set the MANIPULATOR system variable to 2.

The Manipulator displays.



1 Hover over the vertical or horizontal axis trigger zone of the Manipulator.

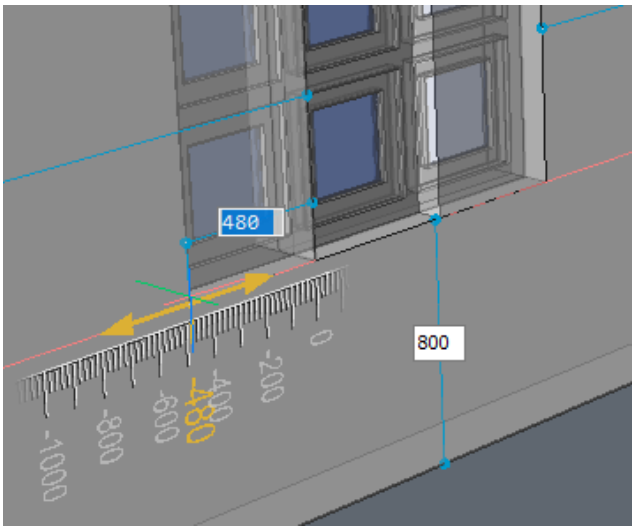
1 The axis highlights and the rest of the Manipulator fades.



2 Click to start the movement operation.

3 The window moves dynamically or if you hold down the Ctrl key creates a copy of the selected insert.

The Ruler displays.



4 Do one of the following:

- Type a distance in the dynamic dimension field.
- Use **Adaptive Grid Snap** of the Ruler.

20.3.13.7 Copying Inserts

Inserting a copy of an insert

1 Hover over the insert. When the insert highlights, choose **Insert BIM Component** in the **Model** command group in the Quad.

2 You are prompted: Select insertion point or [Edit inserted entity/Rotate component/set Base point/ Name/insertion Type/Flip/mUltiple/Change target 3d solids] <0, 0, 0>:

3 Optionally choose the **Multiple** option to insert multiple copies.

4 Proceed with the Inserting Windows and Doors procedure from step 2 on.

Copying an insert in a solid

The **Copy** tool can be used to place copies of an insert in the same solid. Using the repeat option on the **Copy** tool you can create a series of copies at the same interval.

Nota: The movement vector for the copy should be parallel to the face of the wall. If the orientation of the wall, in which the source window or door is placed, is not parallel to either the X- or Y axis of the current UCS, use the **Face** option of the UCS command to align the UCS.

20.3.13.8 Procedure: copying an insert on the wall face

Nota: Make sure that **Dynamic UCS** (DUCS) and **Dynamic Dimensions** (DYN) are active.

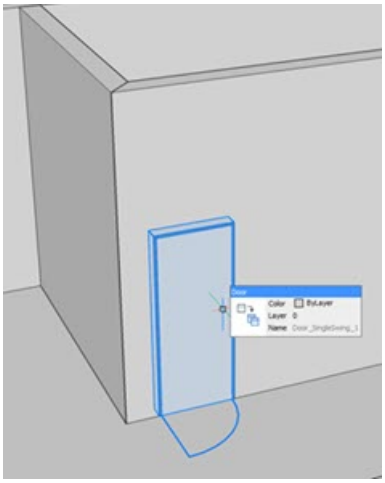
1 Select an insert using the **BIM Insert** tool.

You are prompted: Select insertion point:

2 Place the insert on the wall face.

3 The insert should align with the wall face.

4 Next, hover over the insert to create a copy of it. When the insert highlights, choose **Copy** in the **Modify** command group in the Quad.



You are prompted: Enter base point [Displacement/ mode] <Displacement>:

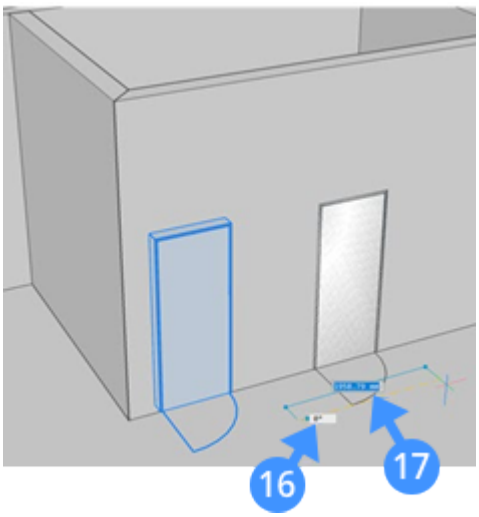
5 Pick a point in the drawing area.

6 You are prompted: Enter second point <Use base point as displacement>.

7 Type a displacement distance from the coordinates of the base point.

Nota: Displacement specifies the distance to position the copy.

You can use the dynamic dimension fields (16,17) to specify the displacement.



You are prompted: Enter second point [Undo/Repeat/Exit].

8 Optionally, use the **Repeat** option to create multiple copies of the insert keeping the displacement distance of each copy same.

9 Press Enter to accept the copying.

20.3.13.9 Replacing an insert

Use the BMREPLACE command to replace an insert.

20.3.14 Mass modeling with quickbuilding

20.3.14.1 Commands

BIMQUICKBUILDING, BIMSPATIALLOCATIONS

20.3.14.2 About

The **QuickBuilding** tool allows you to generate a BIM building just from one or more massing solids. BricsCAD creates a separate drawing and in this drawing walls, slabs and roofs, as well as spatial elements such as floors and spaces will be automatically created

It's useful in early design stages, volume studies and conceptual designs.

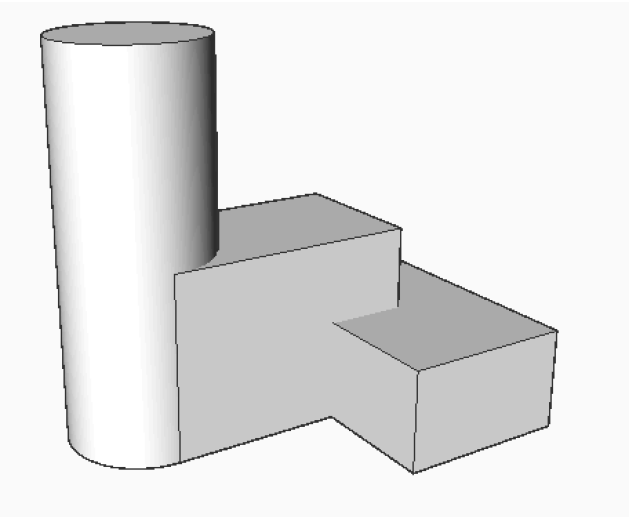
The tool will assign BIM properties and spatial information to each part of the new building.

For more information about this command, please visit the Command Reference article BIMQUICKBUILDING.

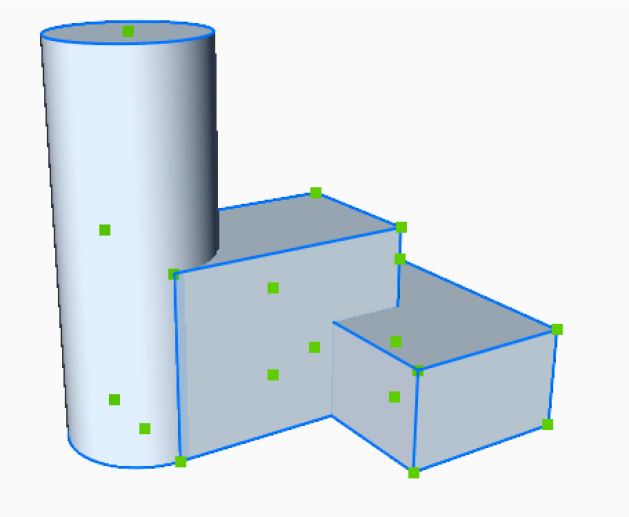
20.3.14.3 Procedure: top-down workflow

1 Draw one or more solids in your model space.

Nota: The solids don't need to be unioned, the command will do it automatically.



2 Type the command BIMQUICKBUILDING in the Command line or select from the Quad, select the solids and press Enter.

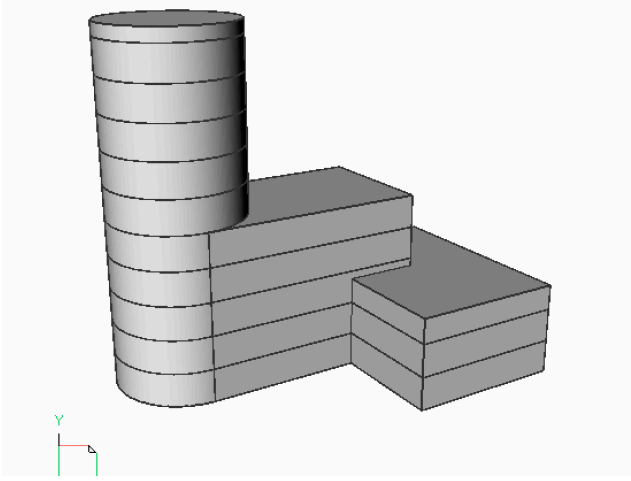


3 You are prompted: Do the selected solids represent [Buildings/Spaces]: Type B to continue with the top-down workflow and press Enter.

4 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatialallocations]:

5 Enter the desired story height.

Press Enter and the building will be generated.



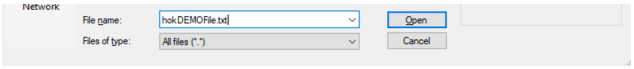
Nota: To generate different floor heights, type (2x4000,*3000,1x5000) which stands for two stories of 4000mm, the rest 3000mm and the top floor has a story height of 5000mm.

Or import the story height values from a .txt file.

Enter BIMQUICKBUILDING in the Command line,

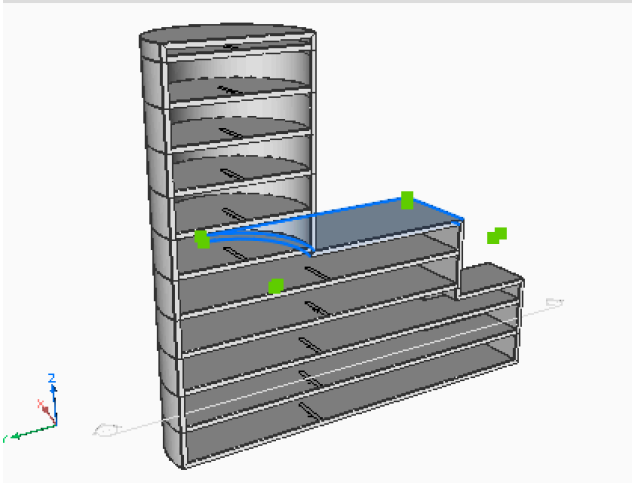
Type BUILDINGREQUIREMENTFILE and press Enter.

Then search for the text file in the file folder and click open.



Now the properties of the text file are applied to the building.
Example of a text file with parameters for your building:
Space:Name='Bricsys Tower',Area=15000
Space:Name='Level 0 - Reception',Area=6%,'Floor-to-Floor Height'=5000
Space:Name='Level 1-10',Area=5%,'Floor-to-Floor Height'=4000,Count=10
Space:Name='Level 11-17',Area=4%,'Floor-to-Floor Height'=3000,Count=7
Space:Name='Level 18-19',Area=4%,'Floor-to-Floor Height'=4000,Count=2
Space:Name='Level 20-21',Area=1.5%,'Floor-to-Floor Height'=4000,Count=2
The area and area ratios that are specified in the file are requirements. If the result of **Quickbuilding** does not satisfy the required areas this will be printed in the Command line, see also the section about Command Line Output later in this article.

6 If you make a section, you can see the newly formed slabs and walls which each have individual BIM properties.

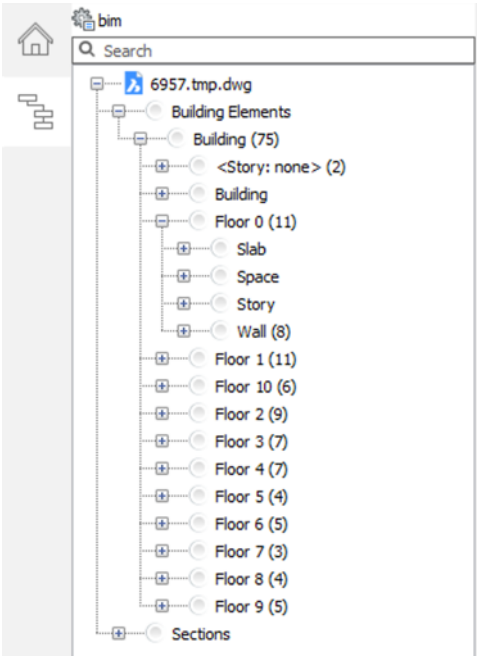


Select the roof and navigate to the **Properties** Panel - **BIM** tab. You will see that this entity has **Roof type** property, as well as some other properties such as spatial locations, quantities or a **GUID**.

BIM	
Type	Roof
Building Element	Roof
Name	
Description	
Building	Building
Story	Floor 5
Composition	
GUID	15iQzJUHr8rxP2hWT$pWj7
Roof type	Not defined
Space bounding	On
Number	
Entity property s	

Nota: All the elements are categorized, in the **Structure Browser** you will see a list of building elements, sorted by floors and building element.

The different buildings elements will be placed in different layers according to their building type (External wall/slab, Flat/pitched roof, Internal wall/slab).

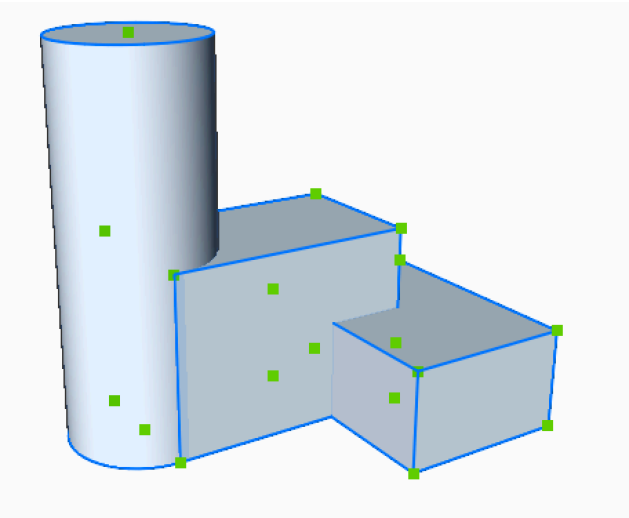


20.3.14.4 Procedure: bottom-up workflow

The bottom-up workflow creates a building out of solids that represent spaces. This approach allows you to first shape the spaces, then use **QuickBuilding** to create the shell and the new spaces.

1 Type QuickBuilding in the Command line.

2 Select the solids and press Enter.



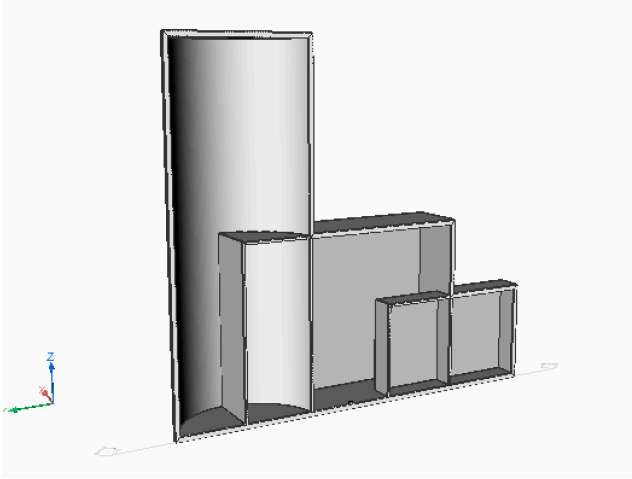
3 You are prompted: Do the selected solids represent [Buildings/Spaces]:

4 Type Spaces to create a building from the selected massing solids.

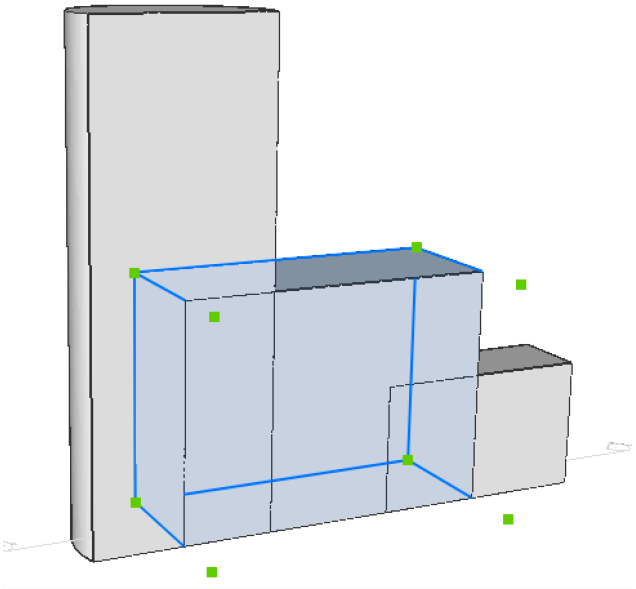
Nota: BricsCAD takes these defined solids and converts them to BIM Spaces with walls and slabs.

5 Press Enter to create the building.

6 (Optional) make a section to see the newly created spaces.



Nota: The original solids are kept, **Quickbuilding** just creates a Building from it with the information that these input solids represent spaces, so it knows that inner walls and slabs should be created based on those input spaces.

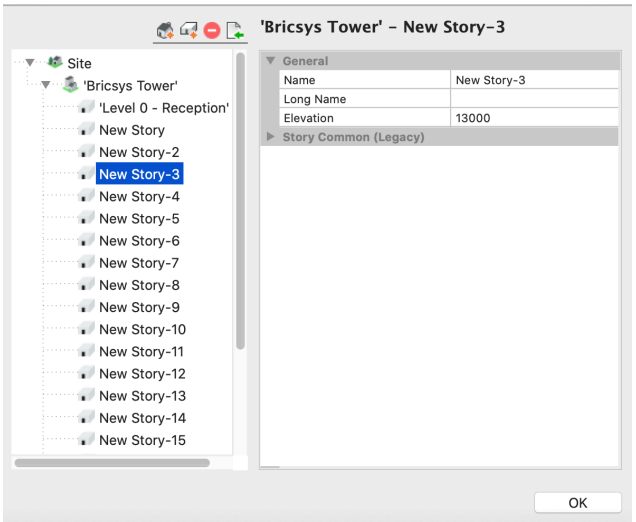


20.3.14.5 Procedure: SPATIALLOCATIONS as input

It is also possible to use a Spatial Structure as input for **Quickbuilding**. So, you can import a Spatial Hierarchy and use that data to subdivide the building.

1 Import or Edit the SPATIALLOCATIONS.

1 For more information about this command, please visit the help center article SPATIALLOCATIONS.



2 Make sure that the solid(s) you wish to use for **Quickbuilding** are:
   - classified as a Building (under **Spatial Structure Elements** in **BimClassify**), and
   - are assigned the correct 'building' in its BIM properties.

3 Type QUICKBUILDING in the Command line.

4 Select the building and press Enter.

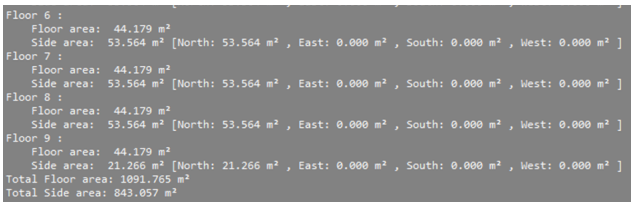
5 You are prompted: Enter desired story height (mm) or [BuildingRequirementFile/Spatiallocations]:

6 Type SPATIALLOCATIONS to use the Spatial Structure as input.

Nota: If you add more than one building in the Spatial Locations manager and then assign the building to the solids, **QuickBuilding** will use the different story heights on the different buildings.

20.3.14.6 Command line output

After you run the BIMQUICKBUILDING command, BricsCAD will create an overview with information on some basic building information. For instance, the total floor area of the building or other information for each floor.



```
Floor 6 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 7 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 8 :
    Floor area:  44.179 m²
    Side area:  53.564 m² [North: 53.564 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Floor 9 :
    Floor area:  44.179 m²
    Side area:  21.266 m² [North: 21.266 m² , East: 0.000 m² , South: 0.000 m² , West: 0.000 m² ]
Total Floor area: 1091.765 m²
Total Side area: 843.057 m²
```

Nota: If you used a Building Requirement File, and some requirements are not passed (e.g. not enough total area or floor area), you can see the problems in the output.

```
Total Floor area: 330.990 m² Requirement NOT passed, area should be >= 15000.000 m² m²
Total Side area: 1001.571 m²
Floor to Side ratio: 33.047 %
```

20.3.15 MEP modeling

20.3.15.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMFLOWCONNECT, DRAG, BIMSTRETCH

20.3.15.2 About BIMFLOWCONNECT

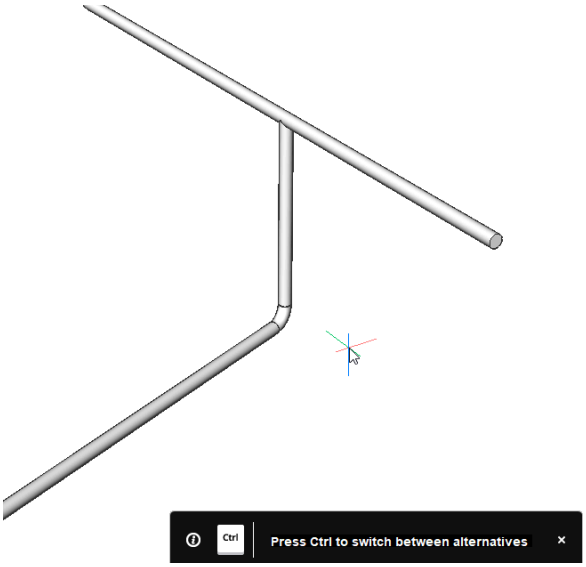
It's also possible to connect different segments using the BIMFLOWCONNECT command.

For more information about this command, visit the Command Reference article BIMFLOWCONNECT.

Connecting different segments to each other

Select the segments you want to connect and launch the BIMFLOWCONNECT command.

When the Hot Key Assistant is switched on (**HKA** on the status bar), the following widget appears on the screen press the CTRL-key to cycle through the different options.



End the command by pressing Enter, the segments are now connected.

20.3.15.3 Procedure: altering the position of segments

Before starting with the procedures: enable the selection of sides and ends:

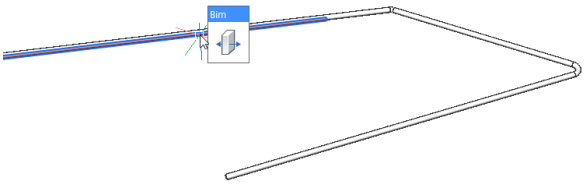
Click the icon on the **Selection Modes** toolbar or set the DISPLAYSIDESANDENDS system variable ON.

DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing its connection to the other segments.

For more information about this command, visit the Command Reference article DRAG.

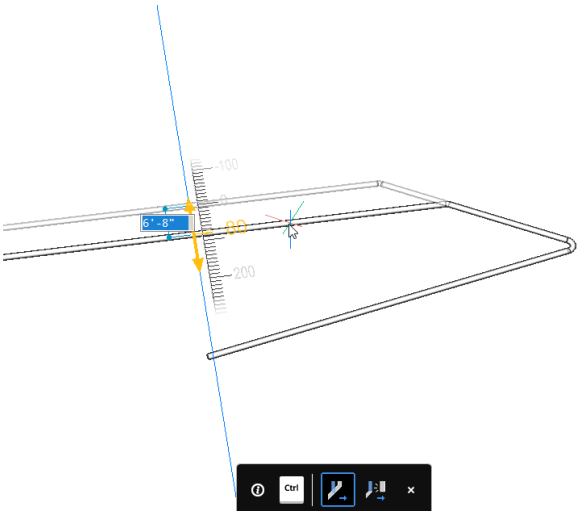
1 Hover over a segment, select a face perpendicular to the section of the profile.



2 Launch the DRAG command.

3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the displacement.

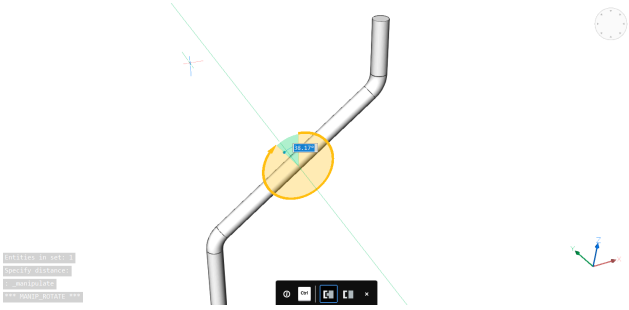
Nota: The software automatically regenerates the connection between the segments.



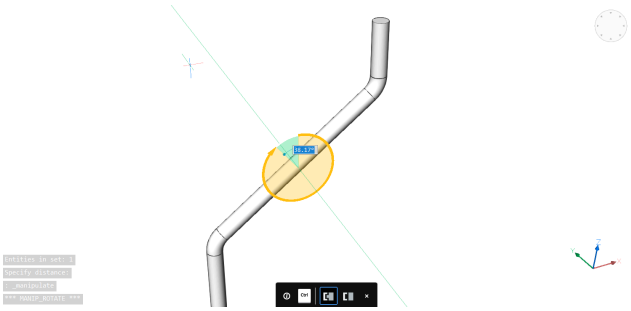
Manipulator

1 Hover over a segment, select a face perpendicular to the section of the profile.

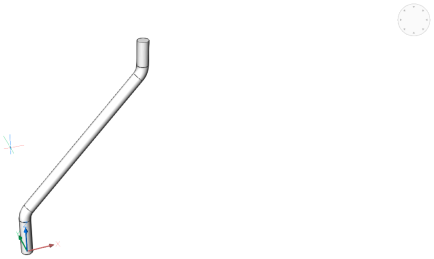
2 Select **Manipulate**.



3 Choose a new position for the segment or use the dynamic dimension field to enter a value for the rotation.

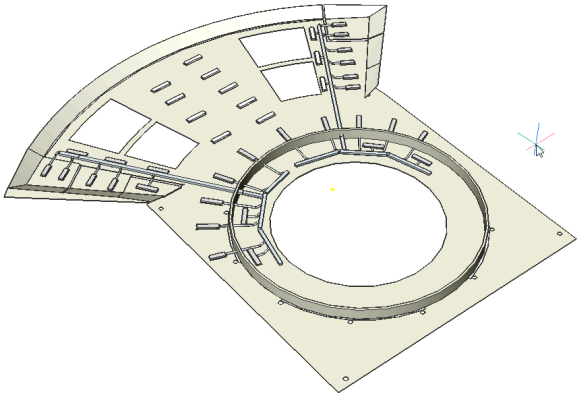


4 Click on the new position or enter the rotation angle and hit enter.



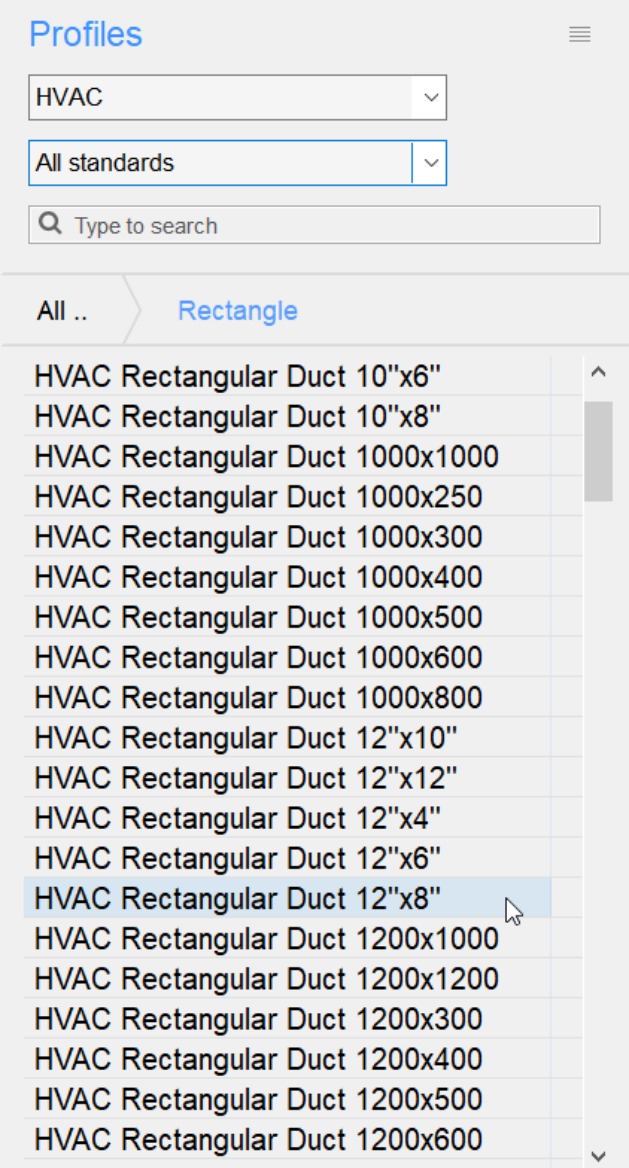
20.3.15.4 Procedure: using the MEP modeling tools to draw and connect profiles

1 Open the drawing that contains the MEP model. You can download this **MEP.zip**-file containing the drawings at the end of this article.



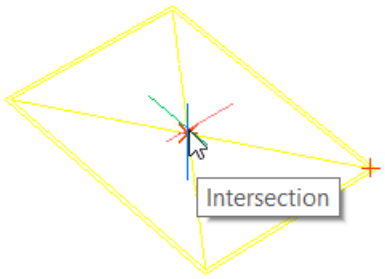
2 Open the BIM **Profiles** panel on the right-hand side of your screen (see Dockable Panels).

Filter on **HVAC** and select the Rectangular 12"x 8" profile, drag it to the model space.



3 Define the starting point of the profile.

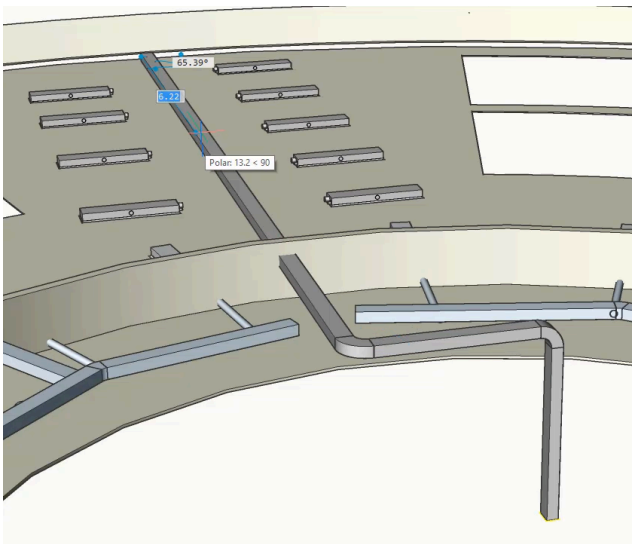
4 Snap to the geometric center of the yellow rectangle.



5 Turn **Ortho** on and define the next points of the profiles by using the dynamic dimension field.

Quarter turns the profile by entering Q in the Command line.

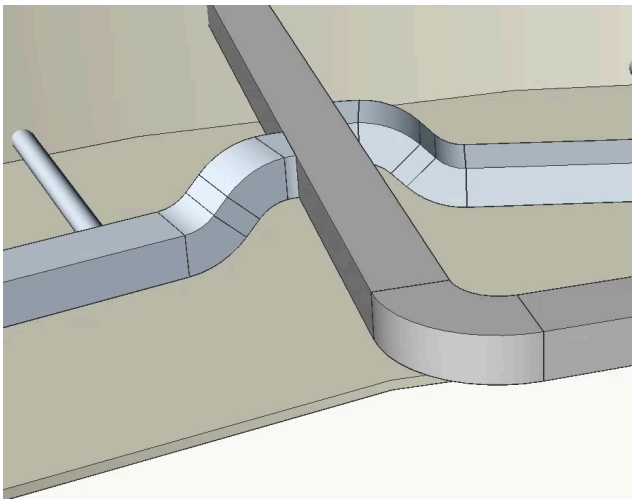
Go up 3.5, go 3.5 to the left and go 21 horizontally.



Nota: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as **Flow Segment**. You can manually classify the profiles through the BIMCLASSIFY command as **Building Service Elements** - **Flow Segments**.

6 Connect the other rectangular ducts on the side to the one we have just created.

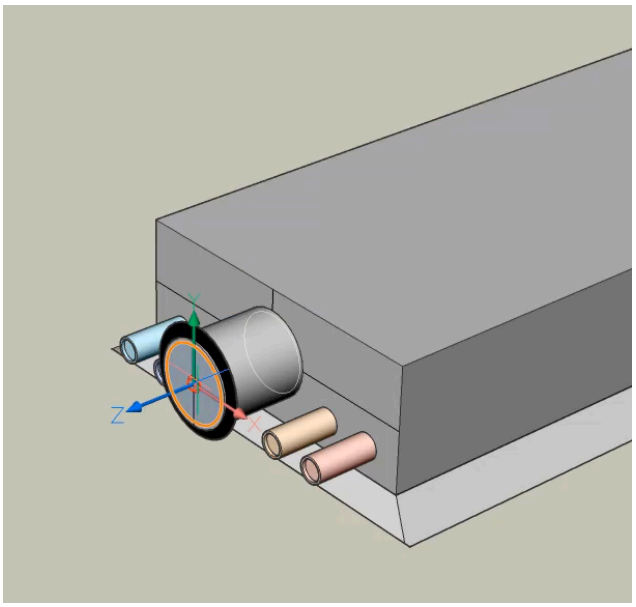
7 Select the ducts together with the main duct and launch the BIMFLOWCONNECT command. Your connection should look like the image below.



8 Create a duct from the flow terminal.

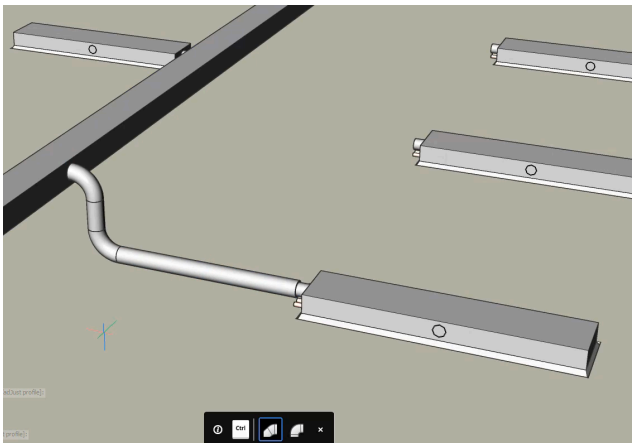
9 Launch the BIMLINEARSOLID command and select the flow connection point.

This connection point contains profile information, so the software automatically creates the predefined circular duct. Give the duct a length of 1.



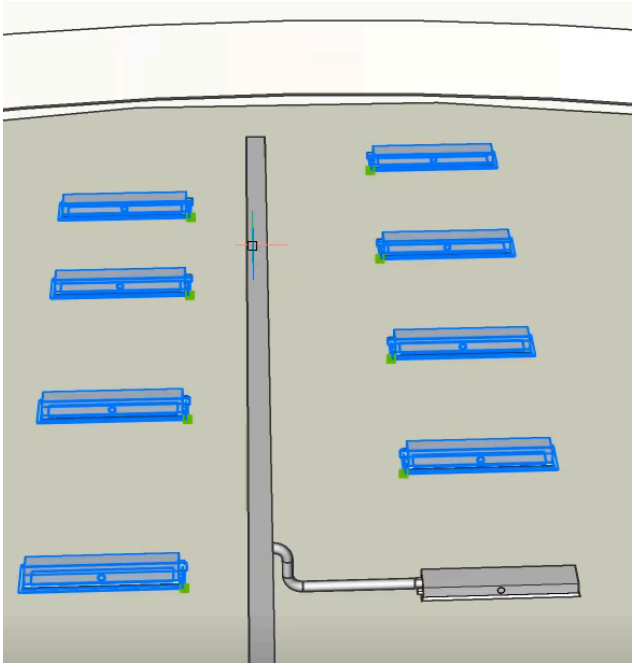
10 Connect the ducts from the flow terminal to the rectangular main duct.

11 Select the main duct and the duct from the flow terminal and launch the BIMFLOWCONNECT command.



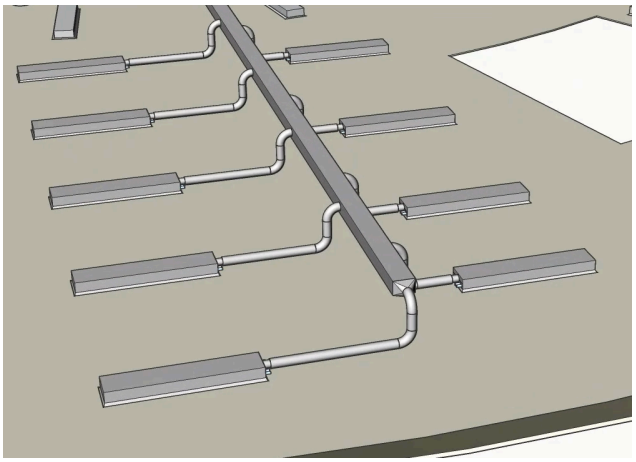
12 Connect all the other flow terminals to the rectangular duct at once.

13 Select the other flow terminals and the rectangular duct and launch the BIMFLOWCONNECT command.



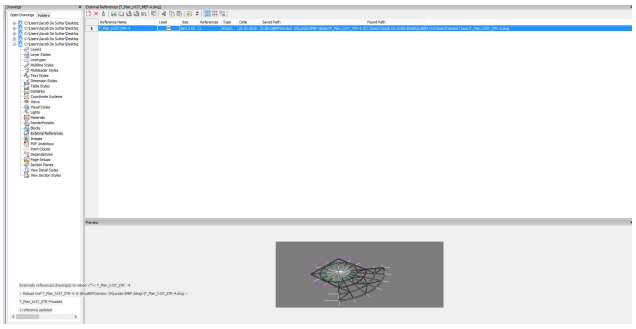
14 BricsCAD uses the information from the flow connection point to determine the type of duct used to connect with the main duct.

Nota: The flow terminal on the back is connected to the back of the rectangular duct.



20.3.15.5 Procedure: Using the MEP Modeling tools to alter profiles

1 Type XRef in the Command line and check the load box next to the structural model to load the structural drawing.



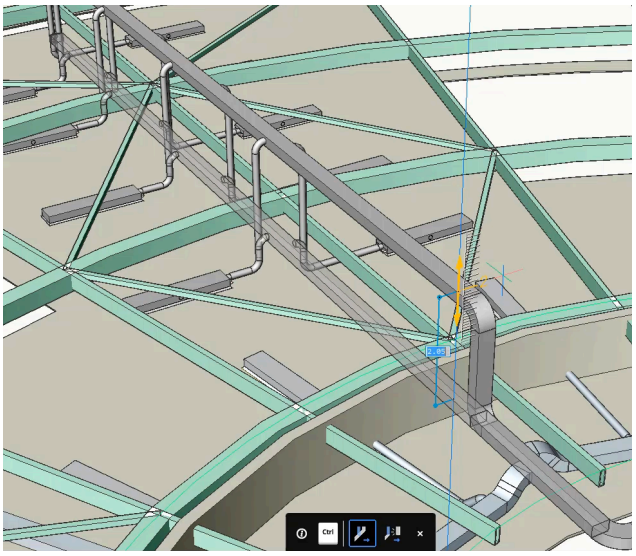
2 Use SLICE command and **Multislice** option to divide the rectangular duct.

Enter 3 in the dynamic dimensions field and press Enter to end the command.

3 Alter the positioning of the rectangular duct, so it doesn't interfere with the structural model.

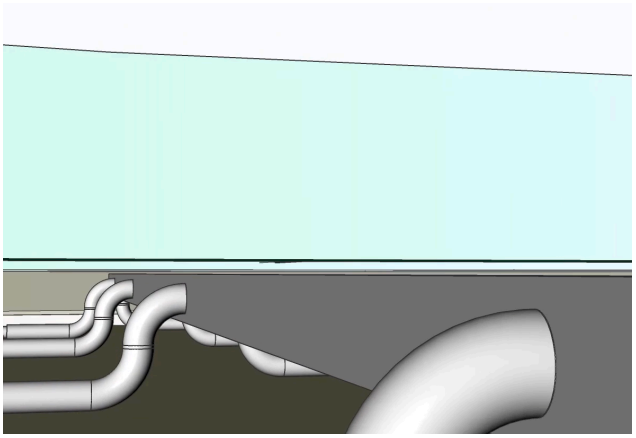
4 Select the upper face of the duct and launch the DRAG command.

Nota: BricsCAD preserves the connections with the circular ducts. You can change this by pressing the CTRL-key if you want to disconnect the ducts.



Move the duct down by 0.23.

5 The structural model no longer interferes with the MEP Model, while the connections between the ducts of the flow terminals and the main rectangular duct were preserved.



20.3.15.6 Procedure: using Bimstretch to modify flow segments

1 Open a New drawing.

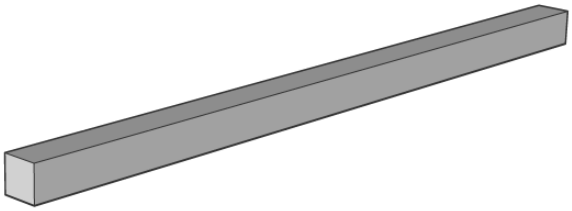
2 Open the BIM **Profiles** panel by clicking on it on the right-hand side of your screen.

Nota: If the BIM Profile panel icon isn't visible yet, right-click on the ribbon and check **Profiles** in the context menu under.

3 Filter on the **HVAC** standard. These profiles will be automatically classified as flow segments once they are dragged into the model space.

4 Search for the profile **HVAC Rectangular Duct 10"x10"** in the **Profiles** panel and drag the profile in the model space.

5 Select a starting point and draw a profile. Press Enter to stop drawing with this profile. Your model should look like this:

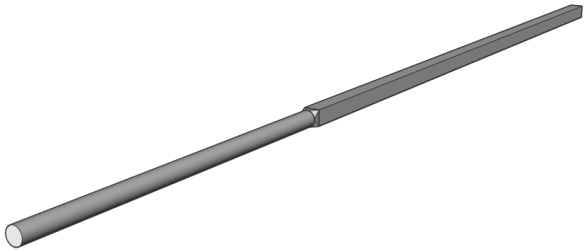


6 Search for the profile **HVAC Round Duct 8"** in the **Profiles** Panel and drag the profile in the model space.

7 Click on the end of the Rectangular profile, after hovering on its profile face.

Nota: To easily snap on the end of the rectangular profile, turn on the **Display Axes** setting or set the DISPLAYAXES system variable to 1.

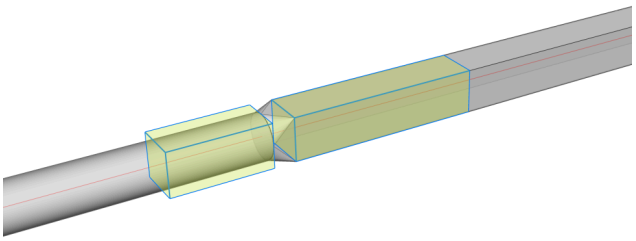
8 Make sure to draw the circular profile along the same axis as the rectangular one. After accepting the last point of the circular profile, it will be connected to the rectangular profile through an automatically generated reducer.



Nota: If you had chosen to draw with a non-HVAC profile, you will not get the automatic classification as Flow Segment and the connection will not be made immediately. You can still manually classify the profiles through the BIMCLASSIFY command and then use BIMFLOWCONNECT to connect the two through a reducer.

Nota: Make sure the **Display Sides and Ends** and **Display Axes** is enabled.

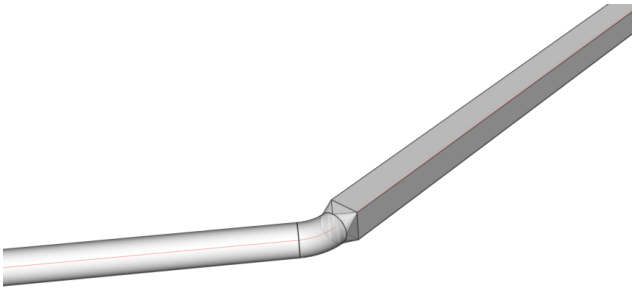
9 Select the connecting ends of both segments.



10 Launch the **BimStretch** tool.

11 Hover your cursor to one of the sides and click on the desired location. BricsCAD automatically stretches the 2 Flow Segments in their new position and adds an extra curved Flow Fitting in the

middle.



Attach **MEP example file.zip**
2 MB Download

20.3.16 MEP flow connection points

20.3.16.1 Commands

BIMLINEARSOLID, BIMFLOWCONNECT

20.3.16.2 About

When creating flow elements like Flow Terminals and Flow Segments, you can add a 2D symbol and classify it as a **Flow Connection Point**. A flow connection point contains BIM data where you can predefine the profile you want to use with this flow component. When you select the flow connection point and use the commands BIMLINEARSOLID or BIMFLOWCONNECT, BricsCAD will automatically use a BIM Profile based on the given information.

See the related links for more information about Linear Solids.

See the related links for more information about MEP modeling.

20.3.16.3 Procedure: creating a flow connection point

In this procedure, you will learn how to create your own HVAC components with flow connection points. As an example, we will create an air heat pump unit.

Creating a flow connection point in the Library panel

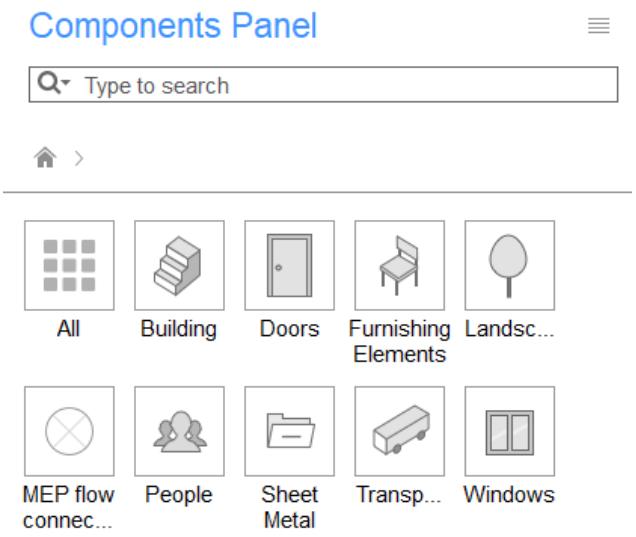
1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and launch BIMCLASSIFY with the <Other> option or hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

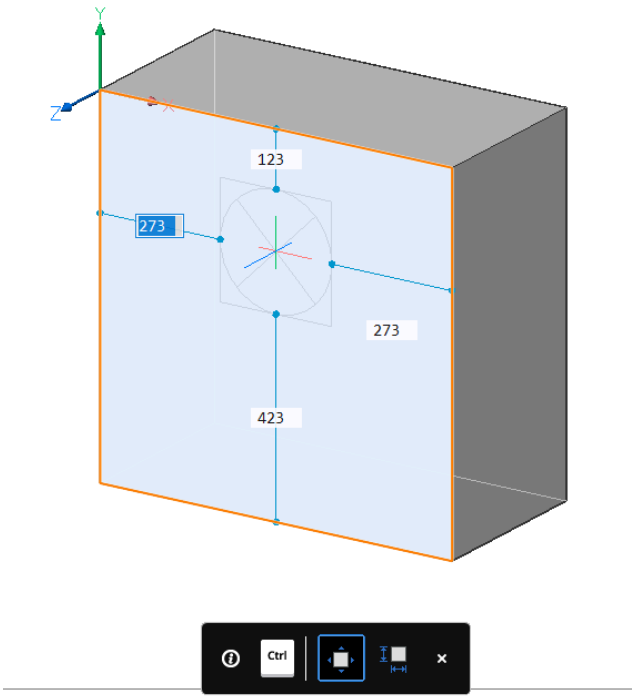
3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Open the **Library** panel.

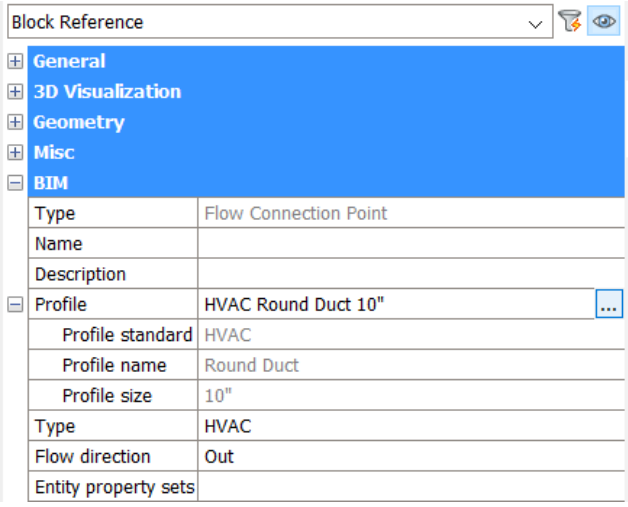
5 Go to the **MEP Flow Connection Points**.



6 Place HVAC Outlets Round on the heat pump box.



7 Select the Outlet and click the **BIM/Profile** property in the **Properties** panel.



8 Click the browse (...) button.

The **Profiles** dialog box displays.

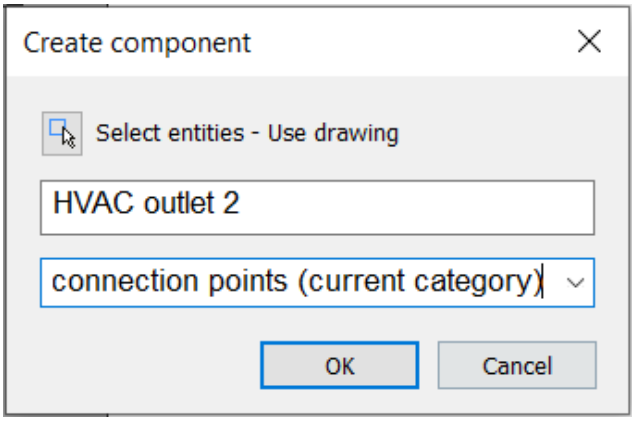
9 Set **Type** and **Standard** = **HVAC**.

10 Choose the desired profile and click the **Select** button.

11 Go back to the **Library** panel and click **Create Component**.

12 Choose a name and category and click the **OK** button.

See the **Components Panel** article for more information about adding a component to the library.



The drawing is now saved as a component and can be used in other drawings.

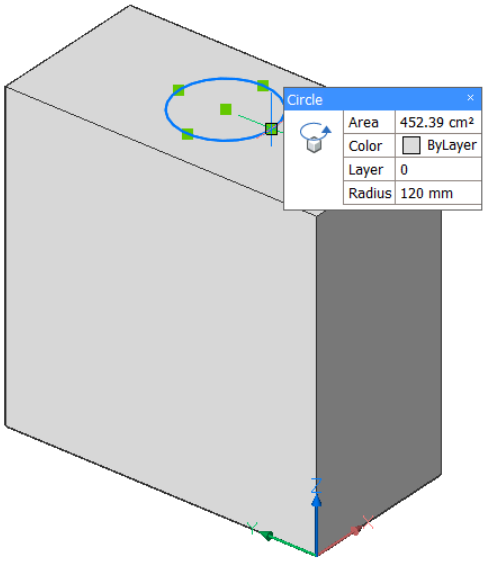
Creating a flow connection point from scratch

1 Draw a box with the desired dimensions (80x40x80 cm) representing the heat pump.

2 Select the box and do one of the following:
- Launch BIMCLASSIFY with the **Other** option.
- Hover over the box and select **Classify Manually** in the **BIM** tab of the Quad.

3 Classify the box as **Building Service Elements** > **Flow Terminal** and click **OK**.

4 Draw a circle on the top face of the box.

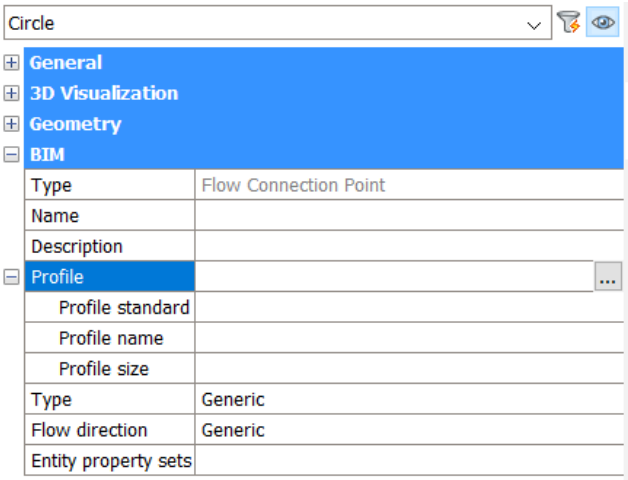


5 Select the circle and launch BIMCLASSIFY with the **Other** option.

6 Classify the circle as **Building Service Elements** > **Flow Terminal** and click **OK**.

7 Select the circle.

8 In the **Properties** panel, select the **BIM/Profile** property and click the Browse (...) button.



The **Profiles** dialog displays.

9 Select the desired profile and click **Select**.

10 On the **Library** panel and click **Create Component**.

11 Choose a name and category and click **OK**.

The drawing is now saved as a component and can be used in other drawings.

20.3.16.4 Procedure: Creating an HVAC System

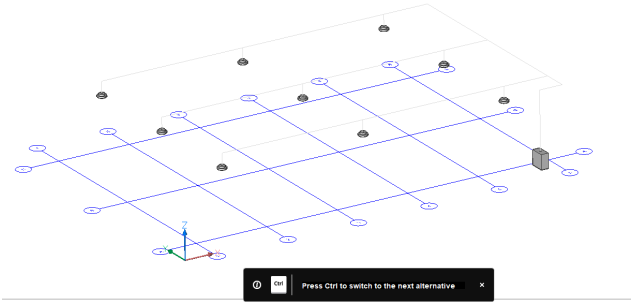
In this procedure, we will use flow connection points to create a ventilation system.

1 Let's start from a file with a series of flow terminals and a ventilation unit.

2 Select the flow terminals and the ventilation unit.

3 Select **FlowConnect** in the Quad or in the **Structural**/**MEP** ribbon tab.

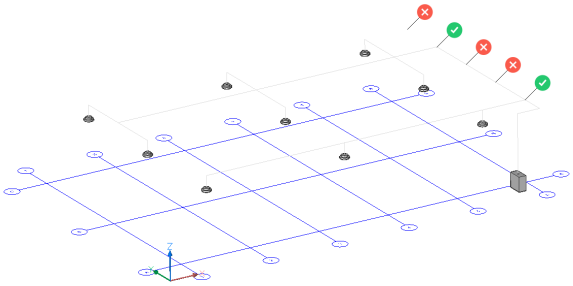
4 Press the CTRL-key to switch between the alternative setups and press Enter.



5 Click the **Tune** option or enter T in the Command line to modify the chosen setup.

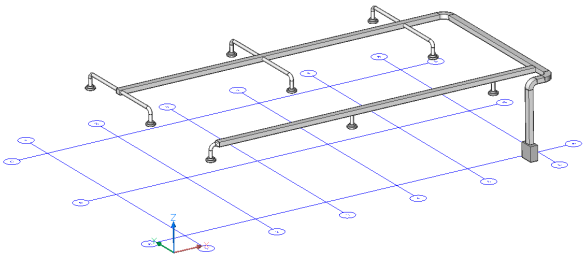
6 Select **Topology** in the toolbar or enter T and press Enter.

7 Select the check icons to modify the layout of the system.



8 Press Enter 3 times to exit the command.

9 ***Nota:*** BricsCAD automatically uses the profiles that were assigned to the ventilation units. This information is stored in the **Flow Connection Points**.



20.3.17 Offset

20.3.17.1 Commands

OFFSET

20.3.17.2 About

The **Offset** tool allows you to create copies of linear entities and align them parallel to the original entities at a specified distance.

Depending on the selected options in the **Selection Modes** toolbar, the **Offset** tool allows to create a parallel copy of:

- An edge of a 3D solid if the **Select Edges** option ( ) is enabled or on or set the SELECTIONMODES system variable to 1.
- The edges of a 3D solid face if the **Select Faces** option ( ) is enabled on or set the SELECTIONMODES system variable to 2.
- Detected boundaries if the **Select Detected Boundaries** option ( ) is enabled on or set the SELECTIONMODES system variable to 4.

20.3.17.3 Procedure: make a parallel copy of a 3D solid edge

Before starting the process, make sure that Quad is active and the **Select Edges** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor over the edge of the solid to highlight the edge.

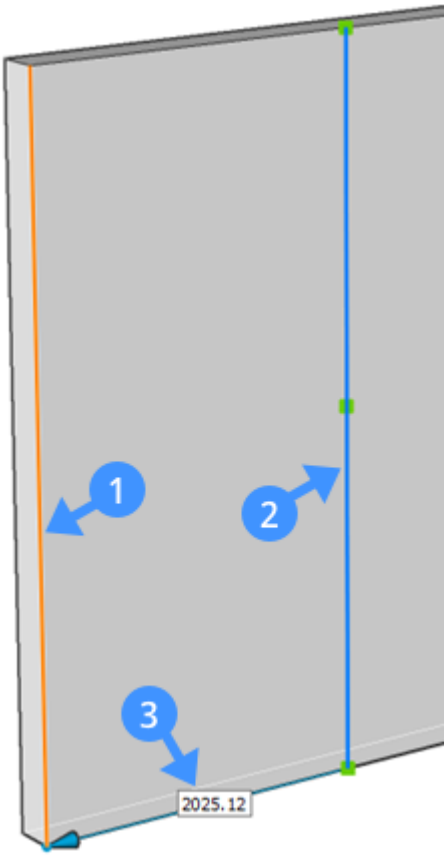
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by dragging the edge of the solid face.

Nota: The distance of the offset displays in the dynamic dimension field. The parallel copy of the edge is created dynamically.

4 While dragging the edge to set a precise offset distance, type a distance in the dynamic entry field and press Enter or left-click to accept the offset.

5 The following illustration shows the offset edge on the 3D solid. The edge highlights in orange is the original edge (1) of the 3D solid, whereas the blue highlighted line with grips (2) is the offset copy of the original one. The line (3) between the original edge and copied edge gives the offset distance.



20.3.17.4 Procedure: Make a parallel copy of a 3D solid face

Before starting the process, make sure that Quad is active and the **Select Faces** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor over the face of the solid to highlight the solid face.

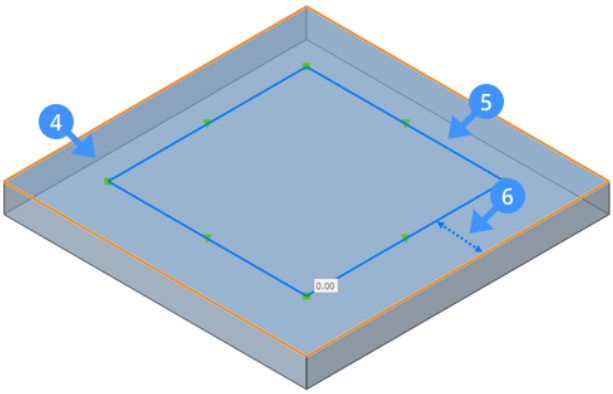
2 Choose the **Offset** tool from the Quad.

3 Specify the offset distance by moving the cursor outside or inside of the solid.

4 The parallel copy of the face is created dynamically.

5 Enter a distance in the dynamic entry field and press Enter, or left-click to finish the offset.

6 The following illustration shows the offset of the 3D solid face. The highlighted face is the original 3D solid face (4), whereas the created polyline (5) is the offset from the original one. The line (6) between the original 3D solid face and copied polyline gives the offset distance.



20.3.17.5 Procedure: make a parallel copy of a detected boundary

Before starting the process, make sure that Quad is active and the **Select Detected Boundaries** option in the **Selection Modes** toolbar is on. Next, follow these steps:

1 Move the cursor inside the boundary to highlight the detected boundary.

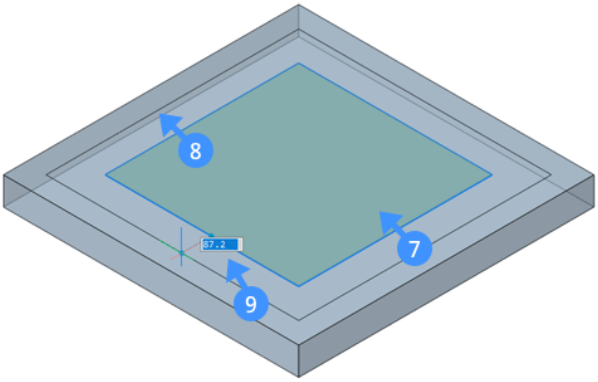
2 Choose the **Offset** tool from the Quad.

3 Move the cursor to define the offset distance and left-click.

4 The parallel copy of a detected boundary is created dynamically.

5 Enter a distance in the dynamic entry field to set a precise offset distance and press Enter or left-click to accept the offset.

6 The following illustration shows the offset boundary on the face of a 3D solid. The highlighted face is the detected boundary (7), whereas the created polyline (8) is the copy from the original one. The line (9) between the detected boundary and copied polyline gives the offset distance.



20.3.18 Splitting solids

20.3.18.1 Commands

BIMSPLIT

20.3.18.2 About

The BIMSPLIT command either splits segmented solids into separated solids automatically or allows to split a solid using a selection of cutting faces.

20.3.18.3 To split a solid automatically

Make sure that the value of the SELECTIONPREVIEW system variable is 2 or 3.

1 a Do one of the following:
   - Click the BimSplit tool button ( ) on the **Model** tab of the ribbon.
   - Choose **Split** in the BIM menu.
   - Type BIMSPLIT in the Command line, then press Enter.

   You are prompted: Select solid for auto-splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 The face highlights.

4 Without moving the mouse, hit the TAB key until the complete solid highlights, then click.

5 You are prompted:

   Entities in set: 1

   Select solid for auto-splitting or cutting faces for manual splitting:

6 Right click.

7 The selected solid is split.

   The number of parts is reported in the command window.

Using the Quad

Make sure the **Select Faces** option ( ) of the SELECTIONMODES system variable is switched off.

Place the cursor over a face of the solid.

When the solid highlights, choose Split ( ) in the BIM command group of the Quad.

The solid is split.

The number of parts is reported in the command window.

20.3.18.4 To split a solid using cutting faces

Make sure that the value of SELECTIONPREVIEW system variable is 2 or 3.

1 Do one of the following:
   - Click the **BimSplit** tool button on **Model** tab of the ribbon.
   - Choose **Split** in the **BIM** menu.
   - Type BIMSPLIT in the Command line, then press Enter.

   You are prompted: Select solid for auto splitting or cutting faces for manual splitting:

2 Place the cursor over a face of the solid.

3 Hit the TAB key to highlight an obscured face.

When the face highlights, click to select the face.

You are prompted:

Entities in set: 1

Select solid for auto splitting or cutting faces for manual splitting:

4 (Optional) Repeat step 4 to select more faces.

5 Right-click to stop selecting faces.

6 The solid is split using the selected faces as cutting planes.

The number of parts is reported in the command window.

Using the Quad

Make sure the **Select Faces** option of the SELECTIONMODES system variable is set.

1 Place the cursor over a face you want to use as a cutting plane.

1 When the face highlights, do one of the following:

2 Go to step 3.

3 Go to step 2 to select more faces.

4 (Optional) Place the cursor over a face you want to use as a cutting plane.

5 When the face highlights, do one of the following:

6 Go to step 3.

7 Repeat this step to select more faces.

8 Choose **Split** in the **BIM** command group of the Quad.

The solid is split using the selected faces as cutting planes.

The number of parts is reported in the command window.

20.3.19 Stair tool

20.3.19.1 Commands

BIMSTAIR

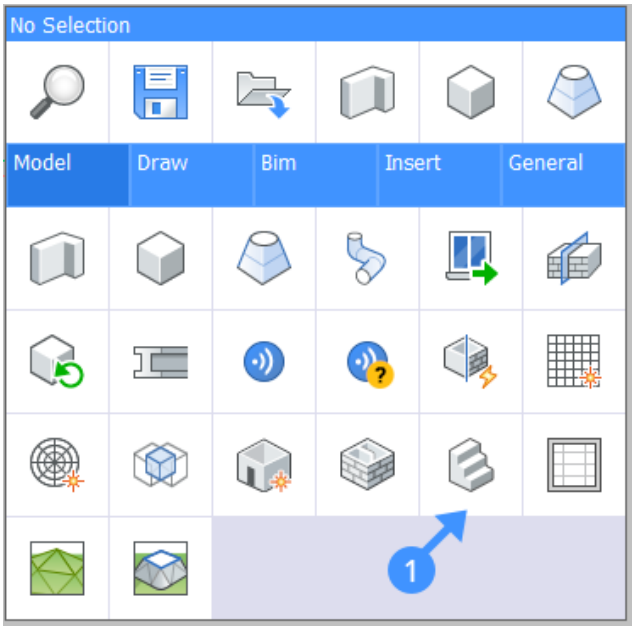
20.3.19.2 About

The **Stair** tool is a powerful tool that allows you to add a custom flight of stairs for your project in just two clicks. You can change the default stair settings as required. The **Stair** tool will automatically try to fit these requirements. It is possible to create single-flight straight, double-flight L-shaped, and double or triple-flight U-shaped stairs.

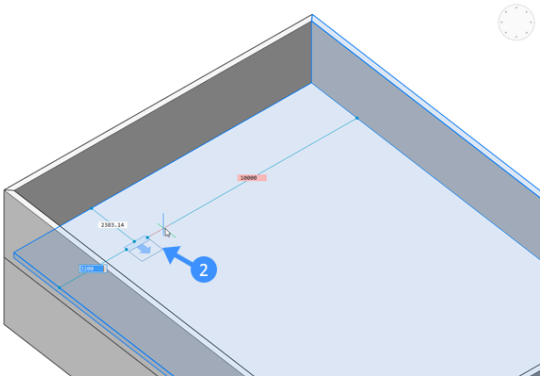
For more information about this command, visit the Command Reference article BIMSTAIR.

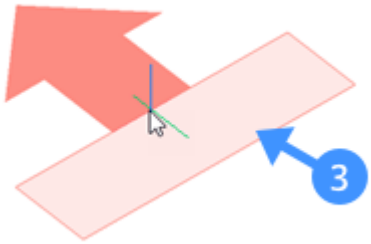
20.3.19.3 Create a straight flight of stairs

1 Select the **Stair** tool from the Quad (1).



2 Hover over a slab. The Stair Cursor will appear (2) and the slab on which the stair cursor hovers, is highlighted. This indicates the position of the first step. If the position of the cursor is invalid, the cursor will turn red (3).



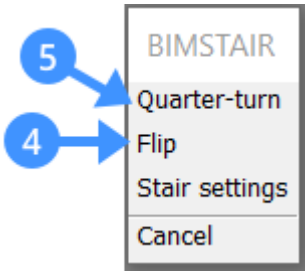


3 To flip the direction of the first step, click the **Flip** button (4) or type F.

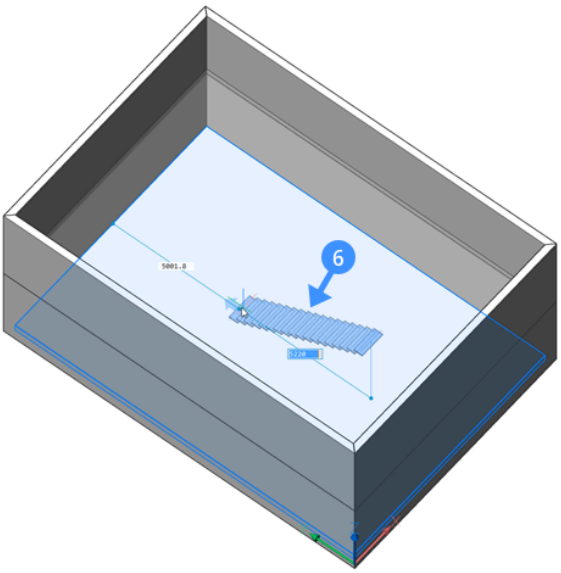
To turn the direction of the first step 90°, click the **Quarter-turn** button (5) or type Q.

To cycle through the different stair variants, the CTRL-key can be used when the Hotkey Assistant is active.

Nota: The cursor will automatically flip or quarter-turn when it is positioned close to a wall or the edge of a slab. This auto-quarter-turning/flipping behavior is only active when the relevant Stair Setting is turned on.

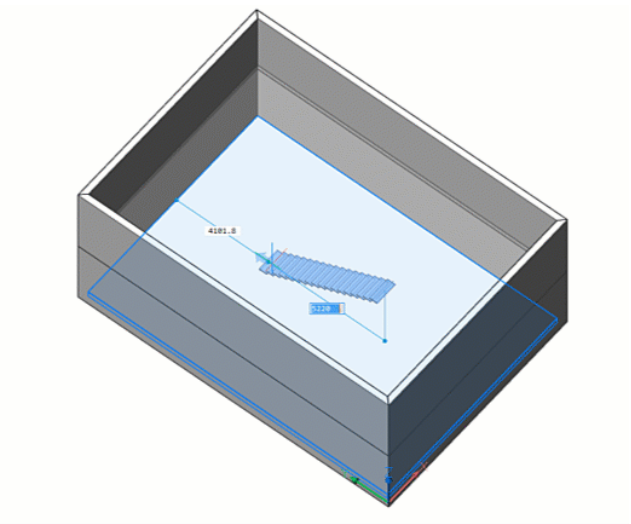


4 Once the cursor is in position, press Left-click. If the upper slab is blocking the view, the slab will become transparent.

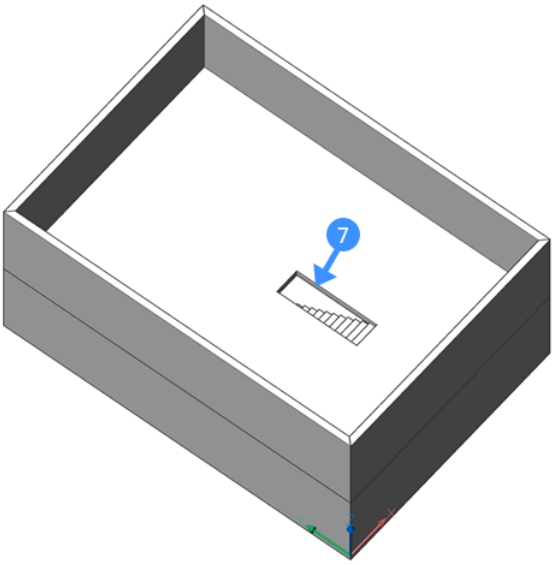


5 To create a straight flight of stairs, align the mouse perpendicular to the first step.

Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (6). The **Stair** tool will automatically snap to the configurations that correspond to "ideal step length" setting, as can be seen in the animation below. When the optimal configuration is satisfied, this preview will display in a darker shade of blue. The other settings are taken into account as well of course, but not to this specific snapping feature (where the stair preview turns darker blue).



6 Once you have the cursor in the correct position, left-click. A flight of stairs will be created. A hole is automatically generated in the upper slab, corresponding to the headroom setting (7).

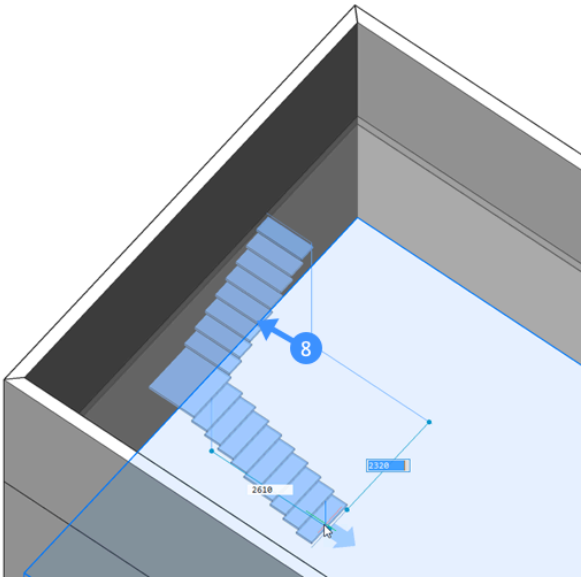


20.3.19.4 Create an L-shaped staircase

1 Select the **Stair** tool from the Quad.

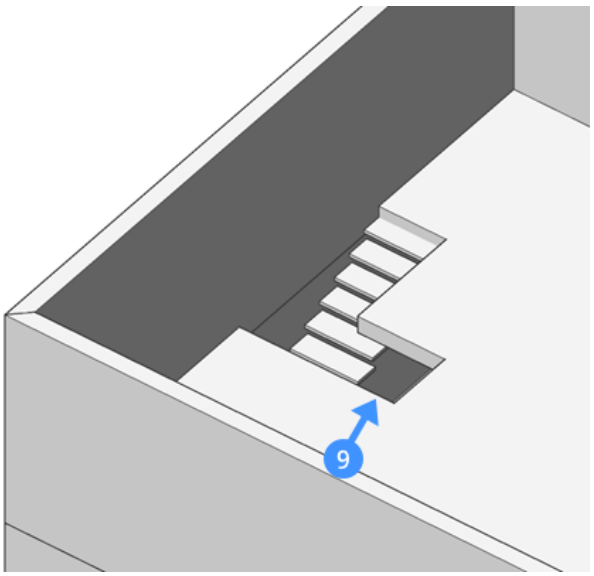
2 Hover over a slab. The Stair Cursor will appear. Left-click once the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to create an L-shape flight of stairs (8).



Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (8). The **Stair** tool will automatically try to find optimal configurations of the stair that correspond with the dimensions given in the settings dialog. It will snap to the optimal configuration and display this preview in a slightly darker color.

4 When you are satisfied with the preview, left-click. A flight of stairs will be created and a hole will be automatically generated (9). This time, the hole may be L-shaped, depending on the length of the upper flight and the "Headroom" stair setting. If the headroom is so large it cannot be covered with only the topmost stair flight, it will span also a part of the second stair flight and thus be L-shaped.



20.3.19.5 Create a U-shaped staircase

1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct

position.

3 Move the cursor away. You can type Q and Enter so that the direction of the second cursor is rotated 180° with respect to the first cursor.

Or use the CTRL-key when the Hot Key Assistant (**HKA** field in the **Status** bar) is ON.

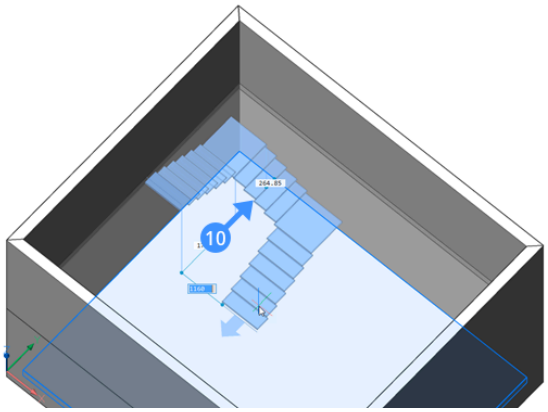
1 Select the **Stair** tool from the Quad.

2 Hover over a slab. The Stair Cursor will appear. Left-click when the stair cursor is in the correct position.

3 Move the cursor away from the perpendicular to get an L-shaped stair.

4 Press the CTRL-key for switching between an L-shaped stair and a U-shaped stair.

5 **Or**align the last step with the first fixed step. When entering a strip parallel to the first step, the last step will automatically quarter turn.

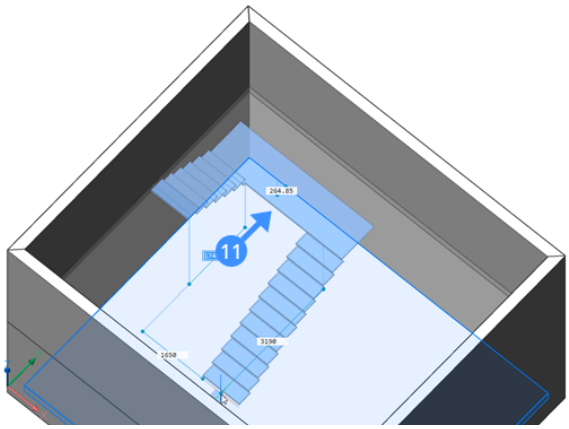


Nota: A live preview of the flight of stairs to be created (blue) will follow the mouse dynamically (10). The **Stair** tool will automatically snap to the configurations that correspond to the current settings. When the optimal configuration is satisfied, this preview will display in a darker shade of blue.

6 There are two possible types of U-shaped stairs you can switch between:
   - The middle section includes steps (10).
   - The middle section is a landing in between the two opposite stair flights (11).

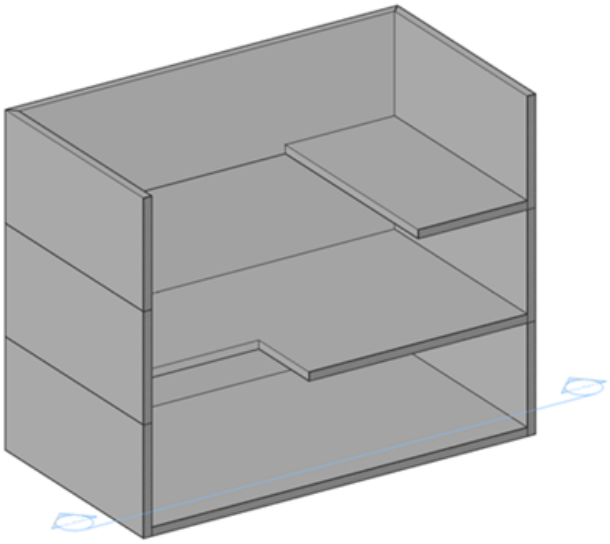
   It is possible to switch between these two modes by typing U and hitting Enter.

   Orpress CTRL to cycle between different stair variants.



20.3.19.6 Create a flight of stairs in an existing opening

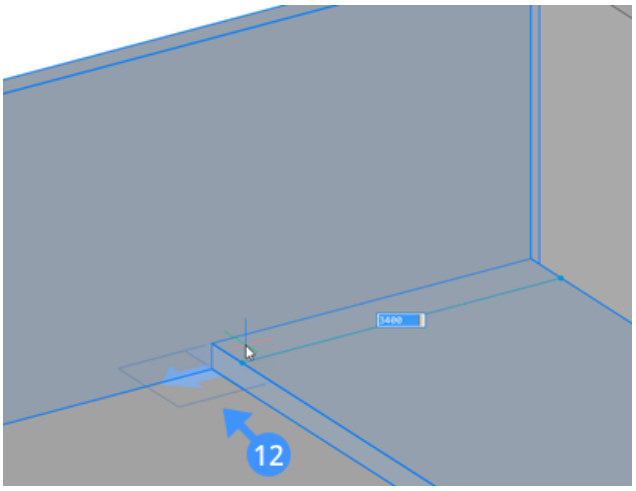
Sometimes it is necessary to create a flight of stairs starting from the edge of a slab, where an opening already exists.



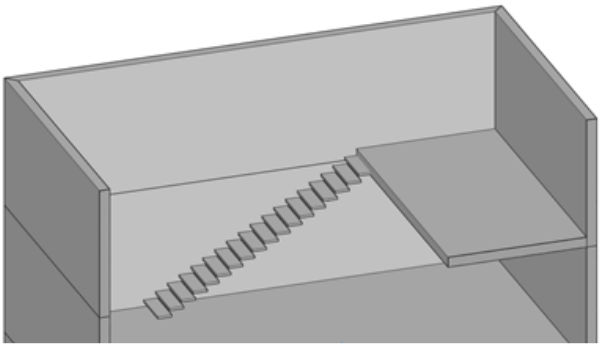
In the example above, different floors need to be connected with flights of stairs. However, an opening does not need to be created.

1 Select the **Stair** tool from the Quad.

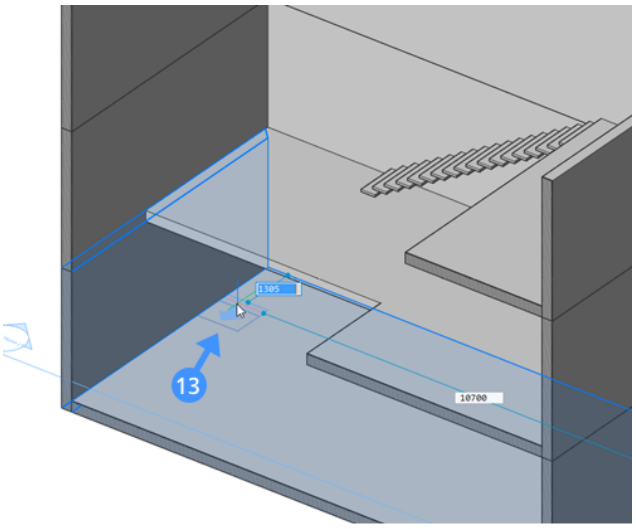
2 Hover the Stair Cursor near the edge of the slab (12). The cursor automatically snaps to the outer side of the edge.



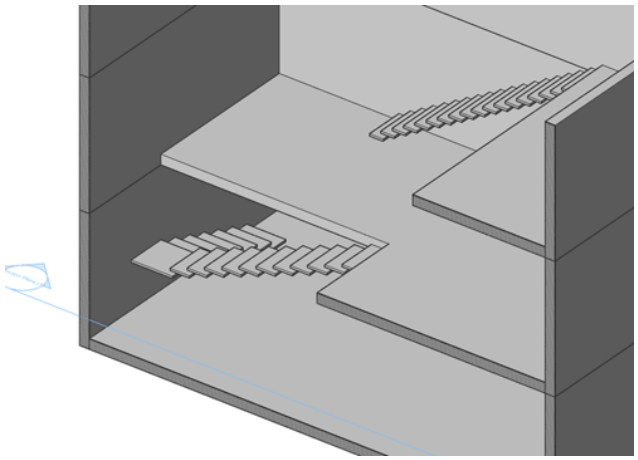
3 You can now create a flight of stairs as described above.



4 You can also make a flight of stairs by starting at the lowest point. This time select a starting point on the first floor (13).



5 Now hover over the edge of the slab on the second floor. The Stair Cursor automatically snaps to the edge of this slab. Left-click to create the flight of stairs.



20.3.19.7 Change the Stair Tool Settings

You can change the stair settings by clicking the **Stair settings** button in the **Stair** tool menu after the **Stair** tool command is launched or simply open the **Settings** dialog and search 'Stair'.

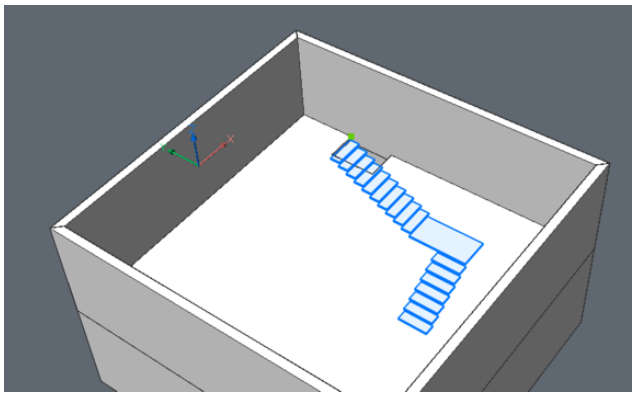
Edit a flight of stairs

You can adjust multiple parameters of a flight of stairs created with the **Stair** tool in the **Properties** panel. These parameters include:

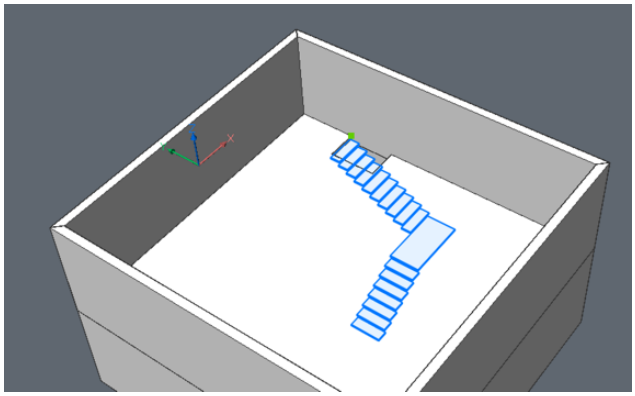
Parameters	
Height_Staircase	325 cm
Stair_Width	1000 mm
Ideal_Riser_Height	170 mm
Step_Thickness	50 mm
Nosing	50 mm
Length_StairFlight_0	145 cm
Length_StairFlight_1	174 cm
Length_StairFlight_2	145 cm

Height_Staircase	The overall height of the flight of stairs. The height is fixed at the top and will adjust from the base.
Stair_Width	The width of the flight of stairs. When a flight of stairs is selected a green dot appears on the upper slab. This dot indicates the insertion point of the flight of stairs. When this point is in the middle of the flight of stairs, the flight of stairs will widen from the center. When it is located at the side, for example when a stair is attached to a wall, the stair will only widen the side away from the dot.

Ideal_Riser_Height	The ideal riser height. When creating a flight of stairs, the real riser height will be calculated based on the ideal riser height and the height of the steps, so that the risers are equally divided as close to the ideal riser height as possible.
Thickness_Step	The step thickness. The upper face of the steps is fixed so that the thickness increases downwards.
Nosing	The horizontal distance of overlap between subsequent treads.
Length_StairFlight_0	The length of the first flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs.
Length_StairFlight_1	The length of an optional, second flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for L-stairs and U-stairs.
Length_StairFlight_2	The length of an optional third flight of stairs. Decreasing this value will shorten the length. The length of the treads and the number of treads for each flight will be updated according to the total length of all the flights of stairs. This option is only available for U-stairs.
Landing_Extension_Up	The length of the landing going down the first flight of stairs. The first flight is fixed so the second flight adjusts to the given parameter value.
Landing_Extension_Down	The length of the landing going down the second flight of stairs. The first flight is fixed so the second flight adjusts to the given value.



Setting: **Landing_Extension _Up**.



Setting: **Landing_Extension_Down**.

You can re-position a flight of stairs created with the **Stair** tool using the MOVE command.

For complex edits to a flight of stairs created with the **Stair** tool, use the BEDIT command to edit the block definition. When in the BEdit-mode navigate to the **Mechanical Browser** and here you can see and edit all the constraints that apply to a flight of stairs.

20.3.20 Structural steel

20.3.20.1 Commands

BIMAPPLYPROFILE, BIMLINEARSOLID, BIMSTRUCTURALCONNECT, BIMADDECCENTRICITY, BIMRECALCULATEAXIS, DRAG, BIMSTRETCH

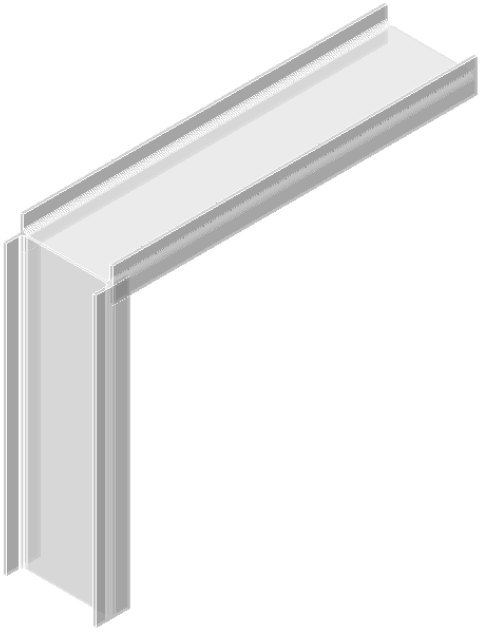
20.3.20.2 How to use the BIM Profiles panel

1 Drag and drop the desired profile from the BIM **Profiles** panel to the model space.

Nota: This command works in the same way as the LINE command.

2 To define the starting point and the following point(s) click in the model space or use the dynamic dimension fields.

3 Press Enter to end the command.



Nota: BricsCAD doesn't automatically generate the connections between the different segments because the chosen profiles are structural steel.

20.3.20.3 About BIMSTRUCTURALCONNECT

The BIMSTRUCTURALCONNECT command automatically generates connections for profiles.

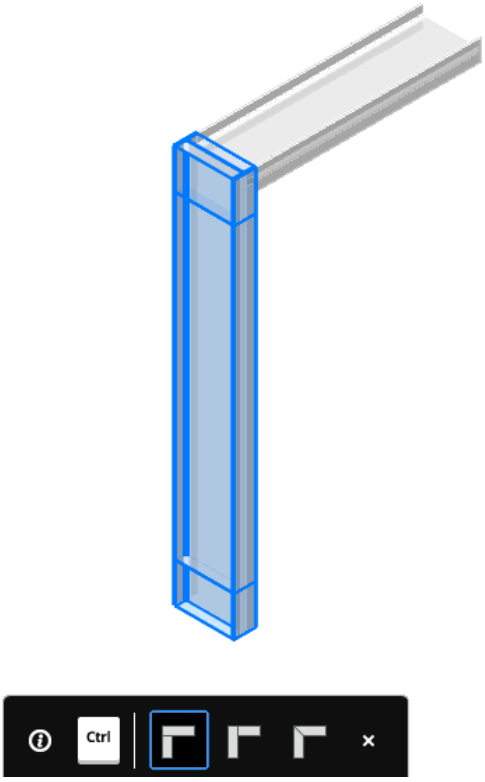
For more information about this command, visit the Command Reference article BIMSTRUCTURALCONNECT.

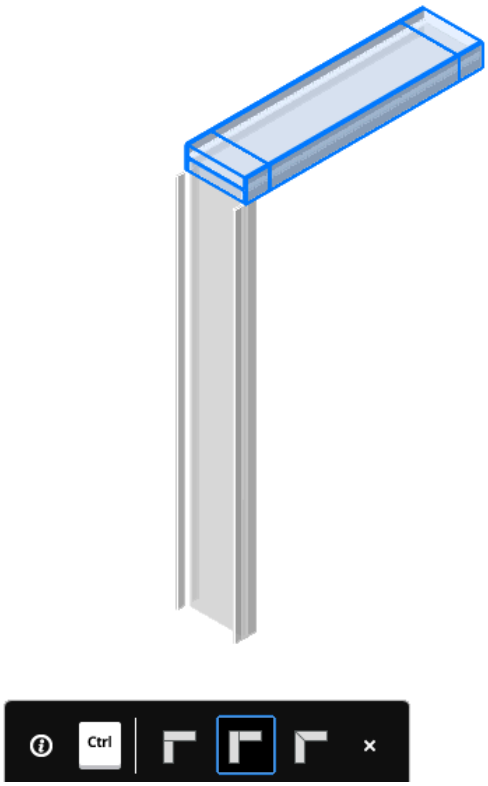
Making a structural connection to linear solids

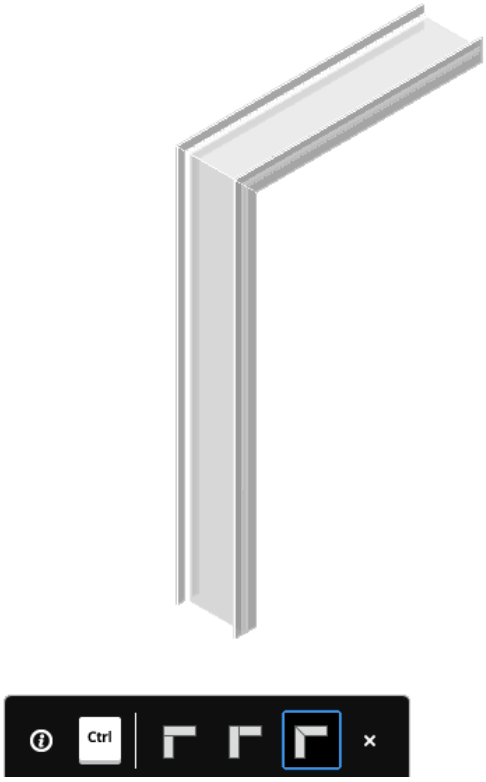
1 Select the profiles you want to connect with.

2 Launch the BIMSTRUCTURALCONNECT command. by typing it in the Command line or select **StructuralConnect** in the **Model** tab in the Quad.

3 When the Hot Key Assistant is on (**HKA** on the **status** bar), a widget (as displayed below) appears on the screen. Tap the CTRL-key to cycle through the different options.


Ctrl


Ctrl



4 Press Enter to accept the selected option and end the command.

20.3.20.4 About DRAG

The DRAG command makes it possible to alter a segment of a drawn profile without losing the connection to other segments.

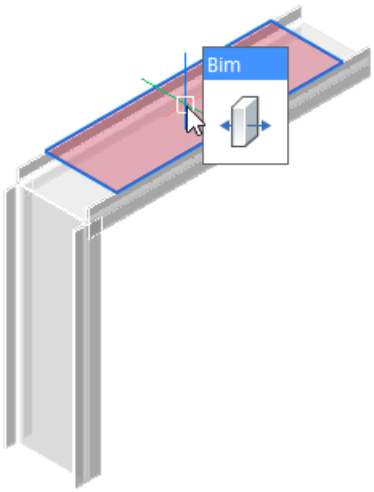
For more information about this command, visit the Command Reference article DRAG.

Altering the positioning of segments

Nota: This only works if the connections were created with the BIMSTRUCTURALCONNECT command.

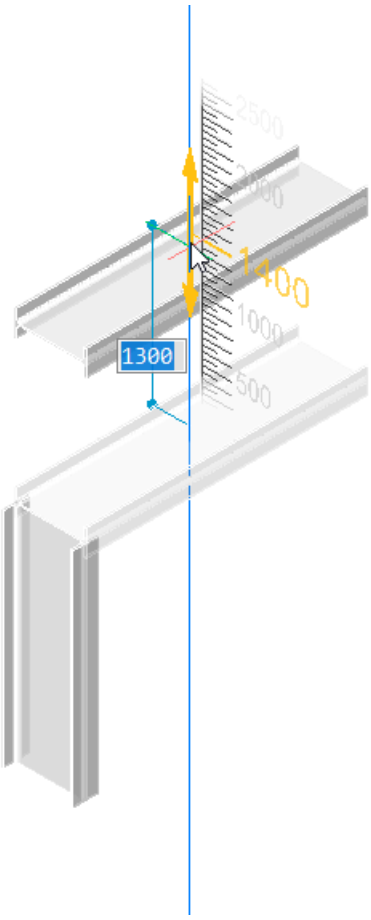
1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Select a face perpendicular to the section of the profile.



3 Launch the DRAG command.

4 Click the mouse to set a new position for the segment. Or use the dynamic dimension field to enter a value for the displacement.



20.3.20.5 Procedure: use modeling tools to create a structural steel model

1 Draw a BimGrid as a base.

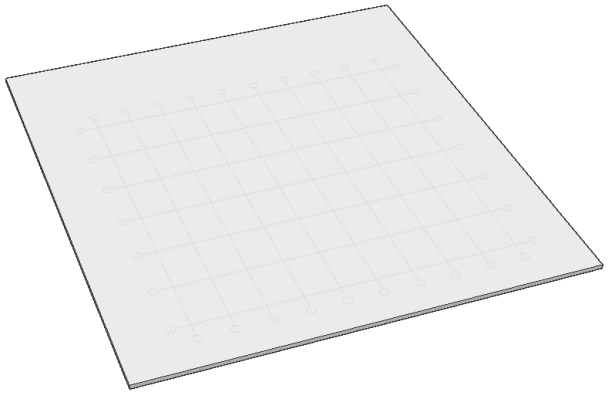
2 Launch the BIMGRID command.

3 Click a starting point and enter values into the dynamic dimension fields.

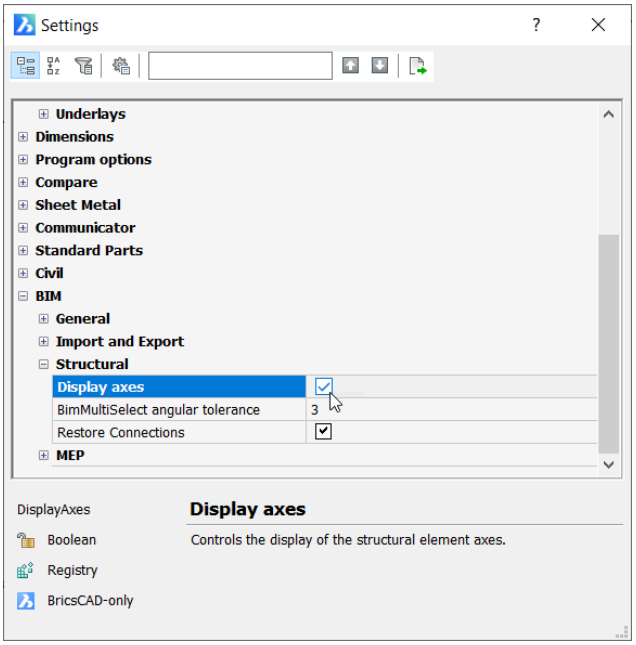
4 Draw a base slab underneath the grid.

5 Create a slab using the BOX command.

6 Ensure that the grid is entirely on top of the slab and that the axes of your profiles are displayed.

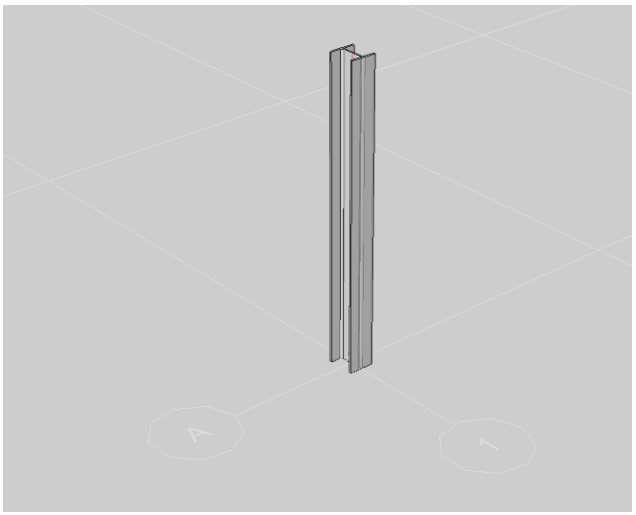


7 Go to the **Settings** dialog and under the **BIM** > **Structural** section. Ensure that **Display Axes** is checked.



8 Open the **Profiles** panel and drag a structural steel profile into your model. Start drawing at the intersection point of two grid axes and draw vertically upward.

Nota: To snap specific points, make sure you have toggled on the different types of points you want to snap in the context menu when right-clicking on ESNAP.



9 Launch the BIMPROPAGATE command.

10 Select the slab as base solid and press Enter.

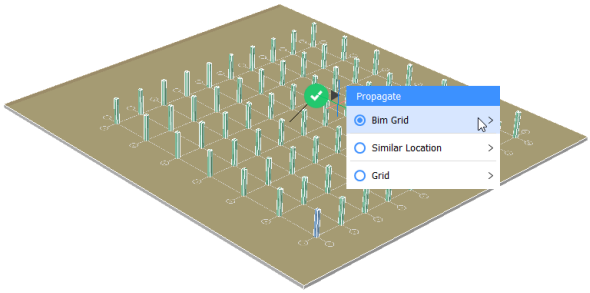
11 Select the column as your detail element and press Enter. BricsCAD will zoom into the 3D detail. Press C to copy the column as a solid, not as a block.

Nota: The default setting for BIMPROPAGATE is to copy the object as a block.

12 Hit Enter.

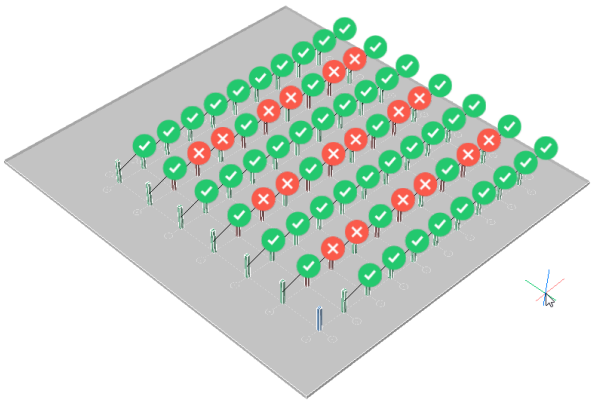
Nota: BricsCAD automatically copies the beam to every grid intersection. This is because you chose the floor slab with a BimGrid as your base solid.

13 Hover over the green checkmark and click **Bim Grid** in the dialog box.



14 Click **Explode**. You can now manually toggle individual suggestions to create the pattern of your choice.

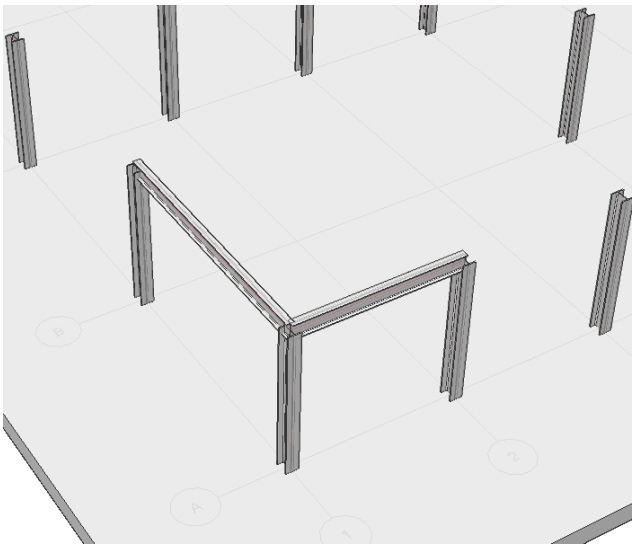
15 Press Enter to accept and end the command.



Create beams between the columns

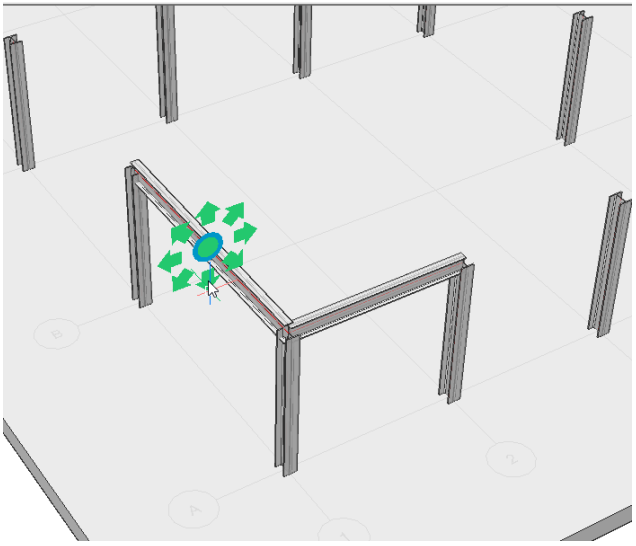
1 Open the **Profiles** panel and drag a structural steel profile into your model. To set the start and endpoints, click the top of the column and then the next. Do this in both directions.

Nota: If you want to turn your profiles to make the flanges are facing upwards, press Q (from Quarter turn). The profile will turn 90 degrees around its axis.



2 Drag and drop a beam profile from the **Profiles** panel onto this beam. Give the second beam a large profile, as it has a longer span.

3 Select a beam and launch the BIMADDECCENTRICITY command to alter the relative position of the axes of the beams.

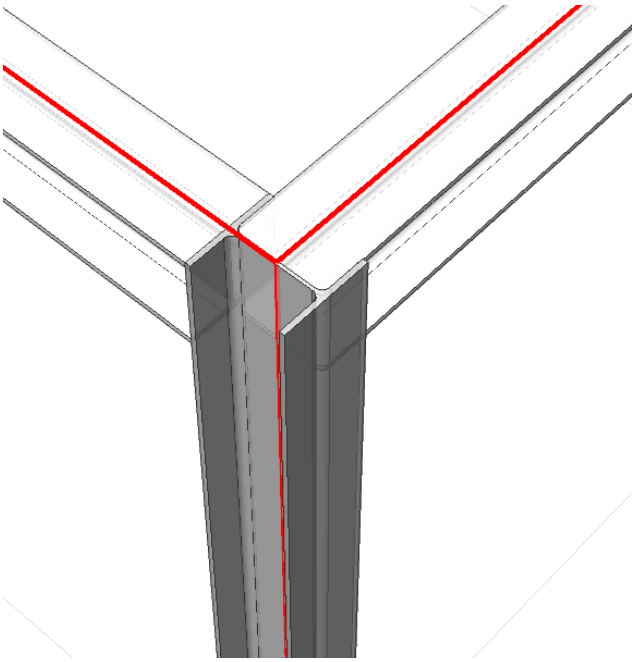


4 Click the down arrow. The beam moves down in relation to its axis.

5 Press Enter to end the command.

6 Repeat steps 3-5 for the other beam.

7 Select the column and the two beams then launch the BIMSTRUCTURALCONNECT command to make a simple connection. Press Enter to accept the default option.



8 Repeat step 7 for the other 2 junctions.

9 Select the two columns connected to the beam in the Y-direction and launch the BIMPROPAGATE command.

10 Select the beam in the Y-direction as the detail solid.

11 Press C to copy it as a solid, not as a block.

12 Press Enter.

13 Choose which suggestions to accept. Press Enter to accept and finish the command.

Nota: Suggestions are made at the bottom of every column because the situation is symmetrical. You can toggle the suggestions off.

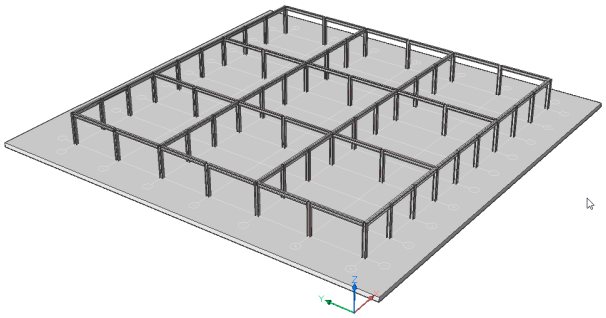
14 Repeat steps 9 - 13 to create beams on the X-direction.

Nota: You can use box selections to select all the lower beams at once.

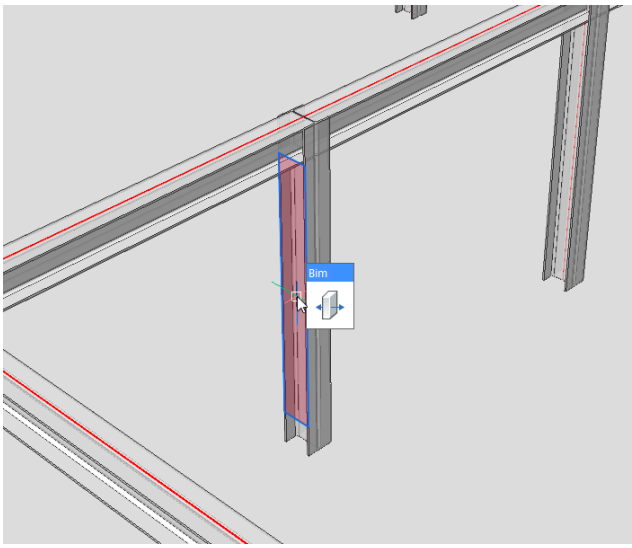
Move a Beam

1 Set the value of DISPLAYSIDESANDENDS system variable to 1.

2 Highlight the side face of the desired column and launch the DRAG command.



3 Move the column to the desired position.



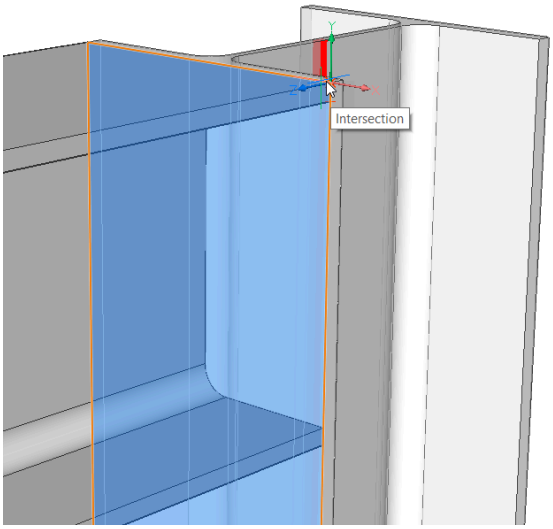
Nota: The connections with the adjoining beams are maintained.

20.3.20.6 Procedure: use the modeling tools to create a more detailed connection

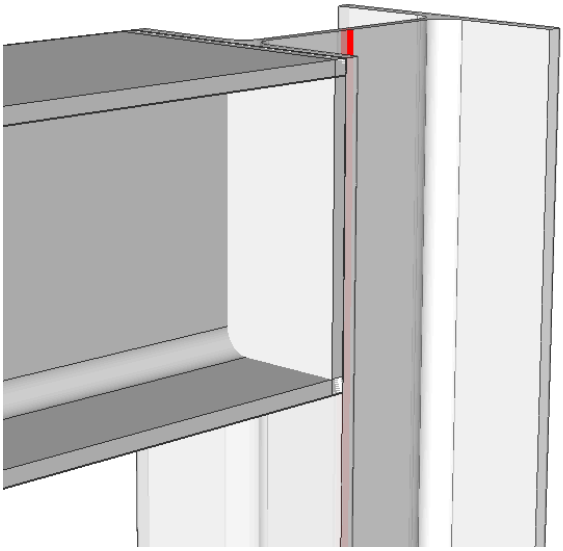
1 Set the value of DISPLAYSIDESANDENDS system variable to 0.

2 Launch the BOX command and hover over the side face of the column.

3 Press Shift once so that the face is highlighted in blue.

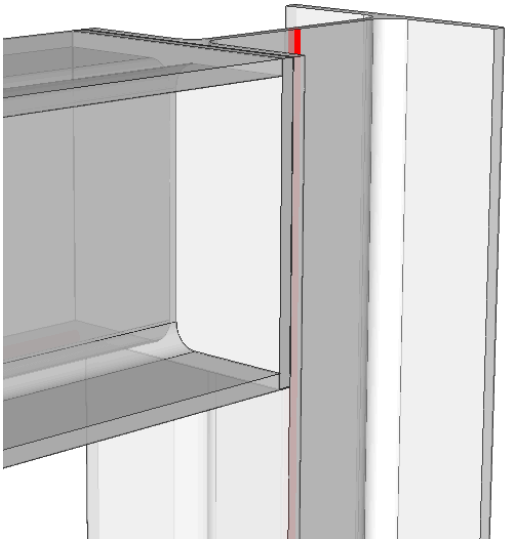


4 Now draw a plate on the flange of the column.



5 Select the beam and launch the SUBTRACT command.

6 Select the endplate and press Enter.

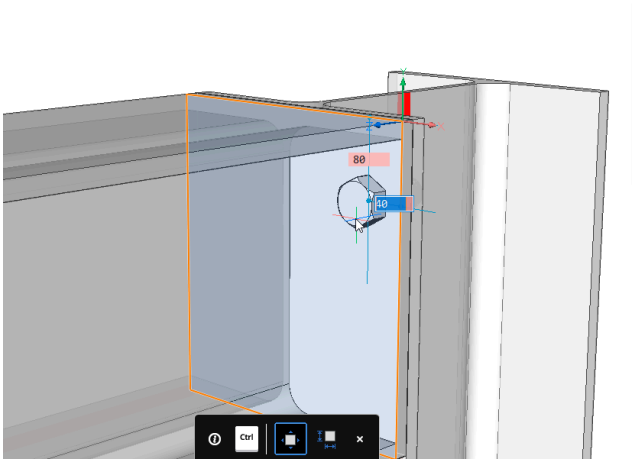


7 Select the face and launch the BIMIFY command from the Quad.

Nota: You can also Classify the plate manually by Quad selecting **Classify Manually** in the **BIM** tab.

8 Open the **Library** panel and drag a bolt onto the plate.

Nota: You can position it precisely by entering values in the dynamic dimension fields.

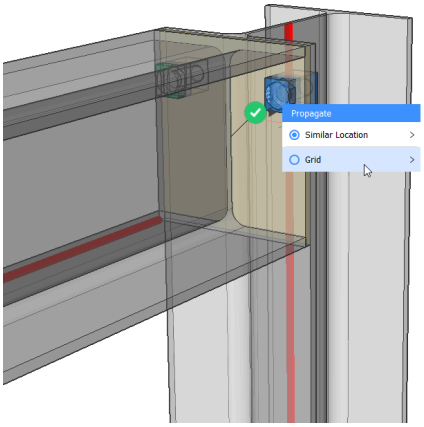


9 Select the plate and launch the BIMPROPAGATE command.

10 Select the bolt as the detail and press Enter.

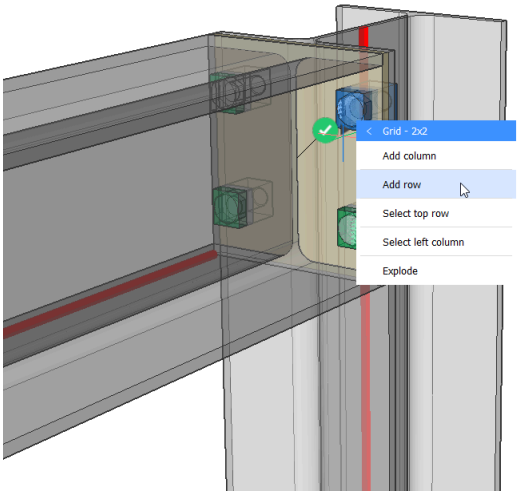
11 Press Enter again and click on the blue question mark. It will turn into a green checkmark.

12 Hover over the checkmark and select **Grid**.

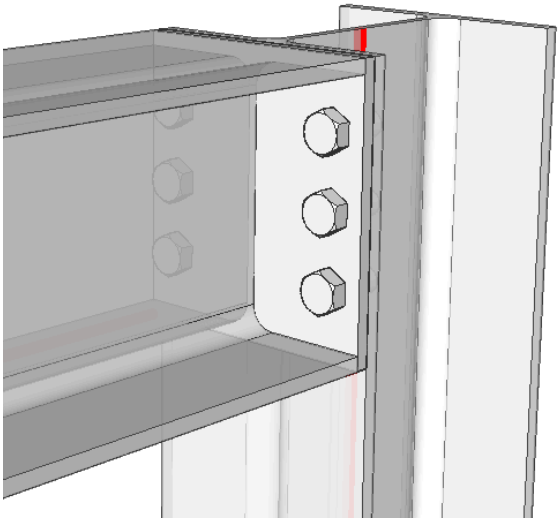


13 Add one row.

14 Press Enter to end the command.



The detailed connection has now been created.



20.3.20.7 Procedure: using BIMSTRETCH to modify structural steel

1 Open a New drawing.

2 Open the **Profiles** panel by clicking on it on the right-hand side of your screen.

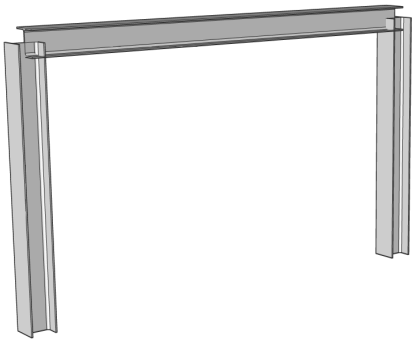
Nota: If the **Profiles** panel icon isn't visible yet, right-click on the ribbon and check **BIM Profiles** in the context menu under **Panels**.

3 Drag the profile you want to draw a structure with within the model space.

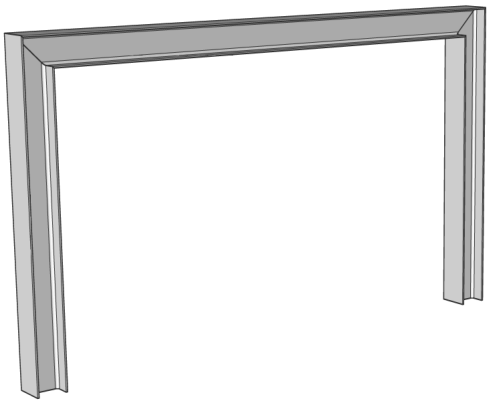
4 Draw a column with a height of 3000 mm. Don't exit the command and draw a connecting beam with

a length of 5000 mm horizontally. Now, add another column at the end of the beam. The height of this column is also 3000 mm.

Nota: Make sure you turned the profiles in the right way.



5 Select the 3 beams and select **Connect Structural** in the ribbon or select the command from the Quad. You will be able to choose which connection you prefer. Choose the L-connection option and press enter. Your entities are now connected.



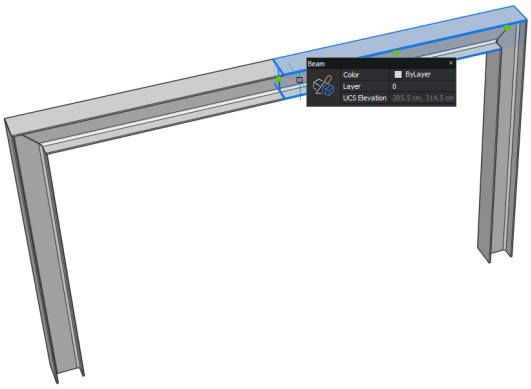
6 In the ribbon, make sure you enable the **Display sides and ends** and **Prioritize selection of Faces**. You can find them in the **Structural/MEP – View** tab or set the value of DISPLAYSIDESANDENDS system variable to 1 and SELECTIONMODES system variable to 2.

7 Check if the profiles are correctly classified as **Beam** and **Columns**.

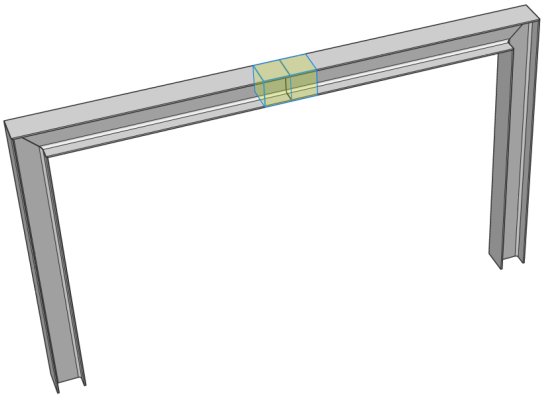
8 Launch the SLICE command and slice the beam in the middle.

9 You will be prompted: Slice along axis? Press Enter to accept the default option (Yes).

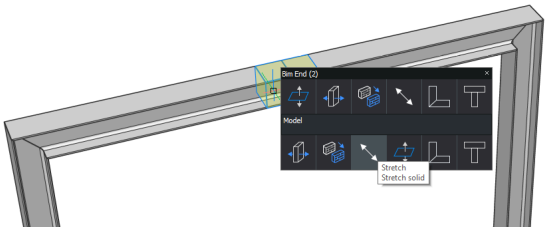
10 Select the midpoint of the beam and press Enter. Your beam is now sliced into two beams.



11 Select the 2 ends of the beams in the middle of the structure. They will be highlighted in yellow.

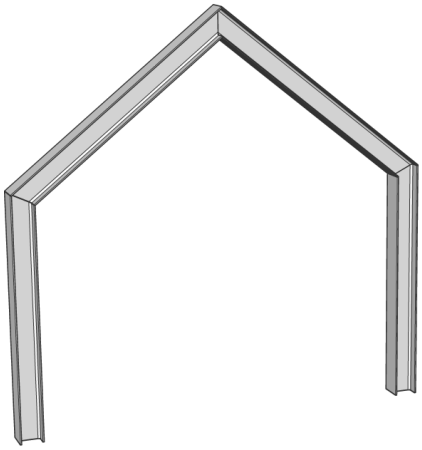


12 Quad select the BIMSTRETCH command in the **Model** tab.



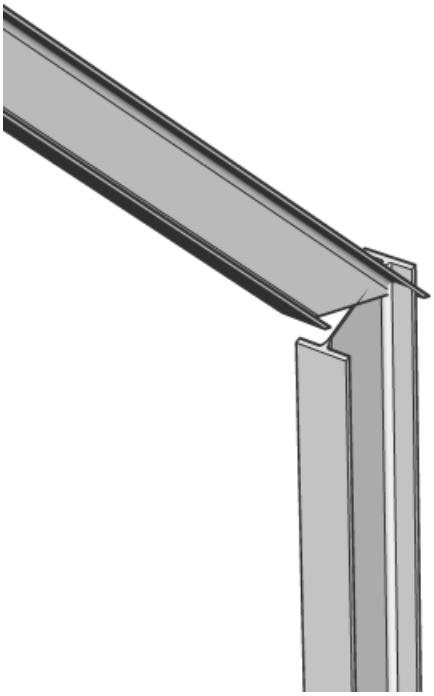
13 Hover orthogonally upwards, enter a height of 2000 mm and press Enter to accept.

14 The result should look like this.



As you can see, the connections between the profiles are automatically adapted to the new angle between the entities.

Nota:If you would only have selected one of the two side-ends in the middle of the beam, the connections on the side that was not selected won't be included in the stretch and will look like this:



20.3.21 Using BIM quickdraw

20.3.21.1 Commands

BIMQUICKDRAW

20.3.21.2 About

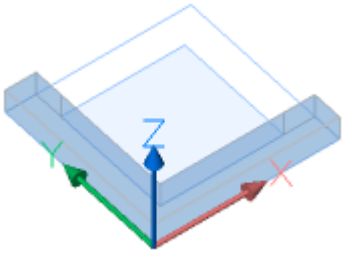
The **Quickdraw** tool allows you to get started with conceptual modeling easily. It also automatically adds BIM data. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, **Quickdraw** will remain active until you explicitly exit the command or until another command is executed.

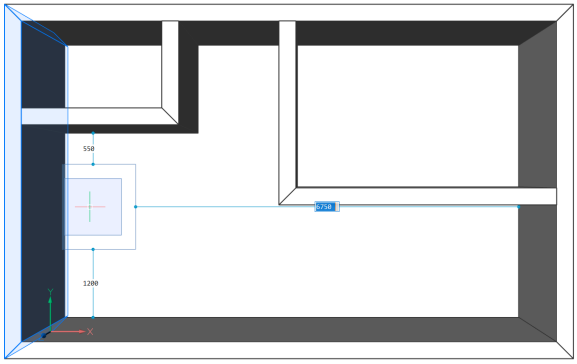
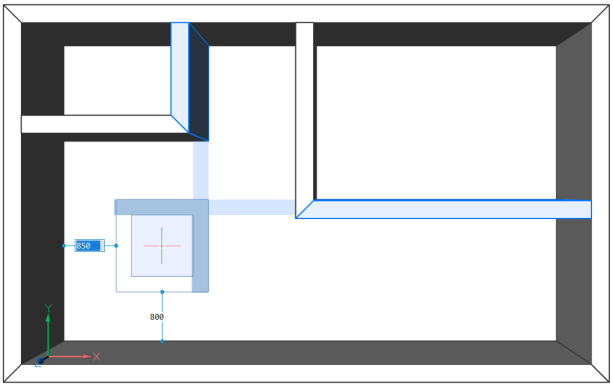
For more information about this command, visit the Command Reference article BIMQUICKDRAW.

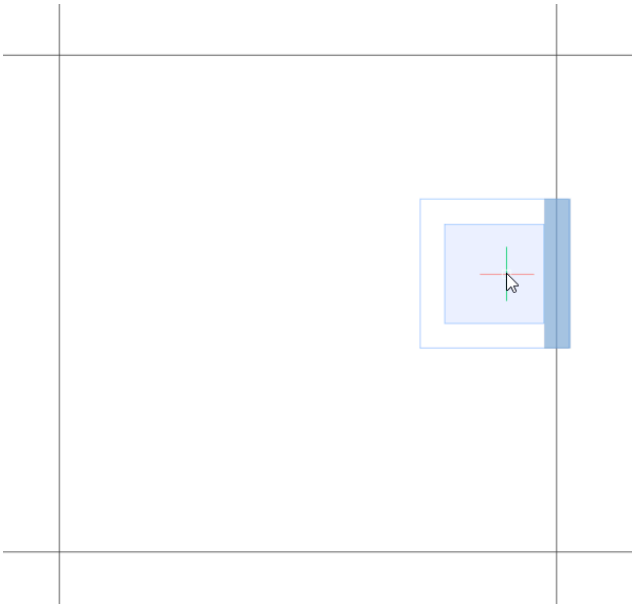
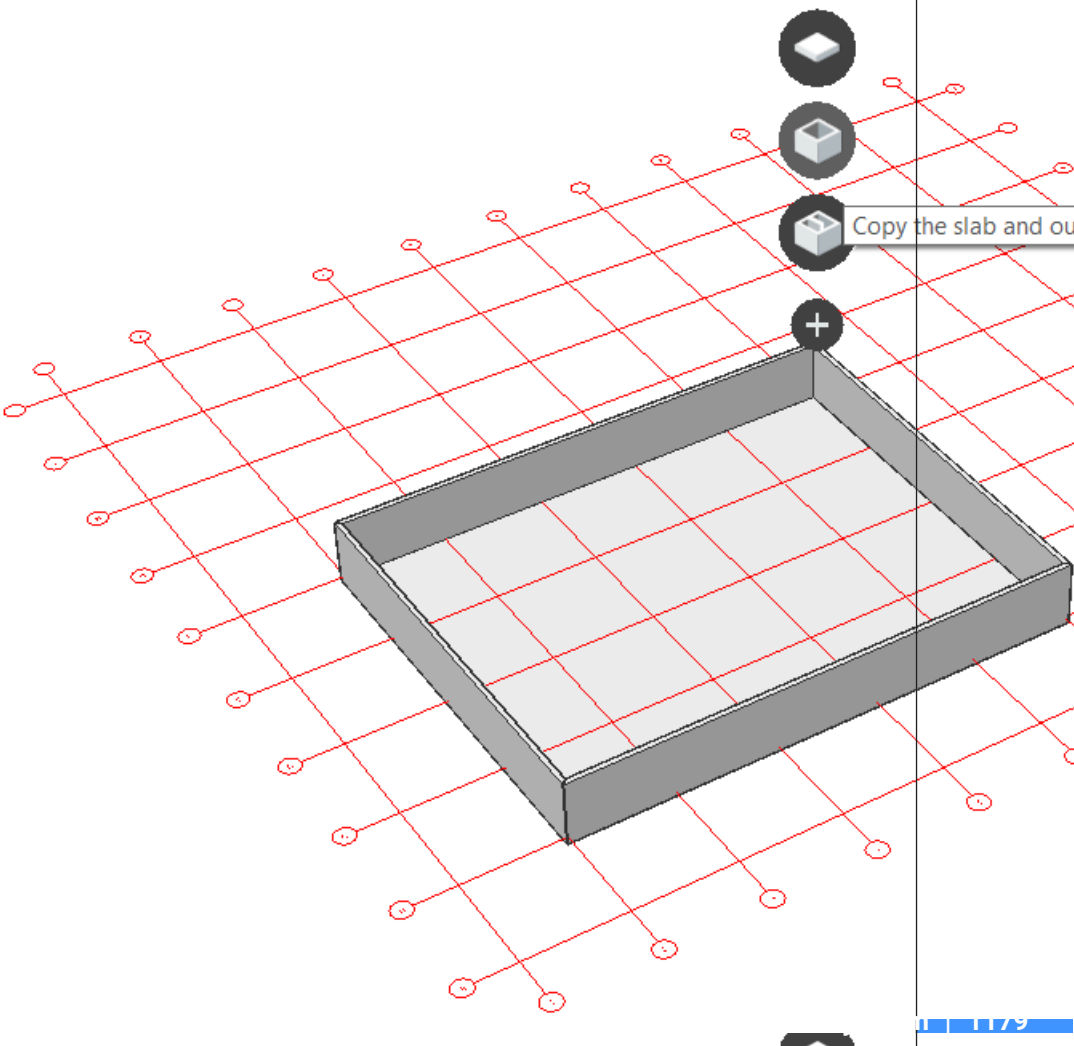
20.3.21.3 The magnetic snapping feature

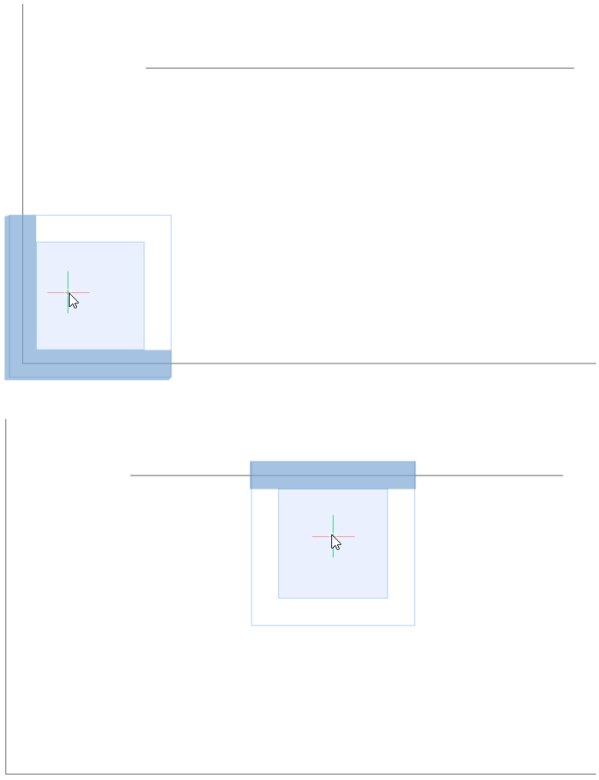
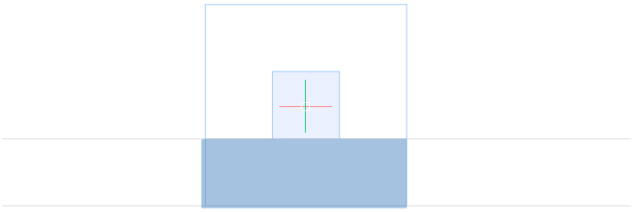
With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object.

You have the following options:

1. Align to the origin	

2. Align to existing walls	To the wall itself 
	To the extents of the wall 

3. Align to lines	To grid lines  ***Nota:*** If you use **Quickdraw** to copy a story, the gridlines will only be copied if the grid is assigned to the same spatial location of the ground floor slab and walls. If this is not the case, the gridlines that lie in the boundary of the ground slab will be projected on the slab of the top floor. This is only a visual help. The gridlines on the slab of the top floor will disappear when you exit the QUICKDRAW command. 

	To X- and Y- aligned lines and line segments 
	To parallel lines 

20.3.21.4 Setting the Quickdraw system variables

You can use the **Quickdraw Settings** to set the Quickdraw system variables:

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the

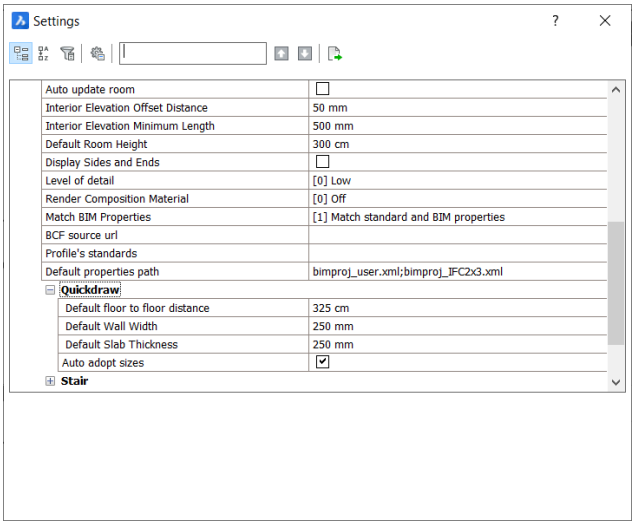
Command line.

2 You are prompted: Select first point or [quickdraw Settings]:

3 Enter S for the **Quickdraw Settings**.

4 Specify a new value to the system variables of **Quickdraw** to change their default values (e.g. **Default Wall Width**, **Default Slab Thickness**, and Default floor to floor distance).

When the **auto adapt sizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to.



20.3.21.5 Changing system variables while running Quickdraw

It is possible to edit the dimensions of the walls you will draw while the QUICKDRAW command is launched. Change these settings in the **Quickdraw Settings** dialog box, select the settings in the prompts menu or enter the specific letters in the Command line and press Enter to change the dimensions.

- H: Change the dimension of the floor height.
- W: Change the dimension of the wall width.
- T: Change the dimension of the slab thickness.
- A: Adapt the dimensions of the room you are drawing in/against automatically.

Nota: Make sure that you enable the hidden options in the prompt menu. Set the value for PROMPTMENUFLAGS system variable to 1.

20.3.21.6 Procedure: creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

You are prompted: Select the first point or [quickdraw Settings]:

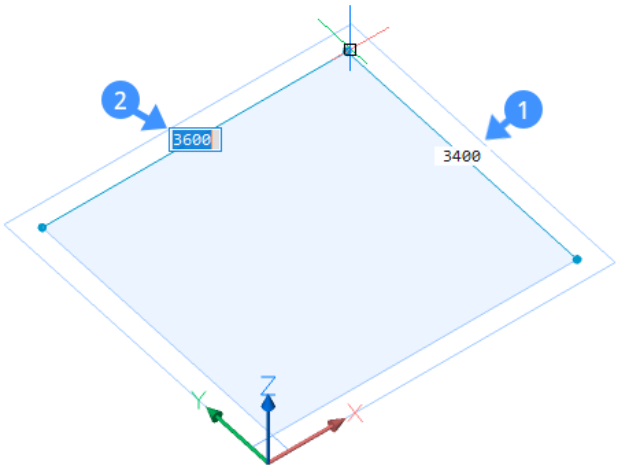
Nota: Use the **Quickdraw Settings** to change the wall width, wall height and slab thickness of the room before you click and then specify the first corner of a building.

2 Select any point in the drawing area to set the first corner.

Nota: To start from the origin, use the magnetic snapping feature of **Quickdraw**.

3 Drag the mouse cursor to define the footprint of the building.

Nota: If the **Dynamic dimension** (DYN) option is on, the current values of the room width (1) and length (2) are displayed in a dynamic dimension field.

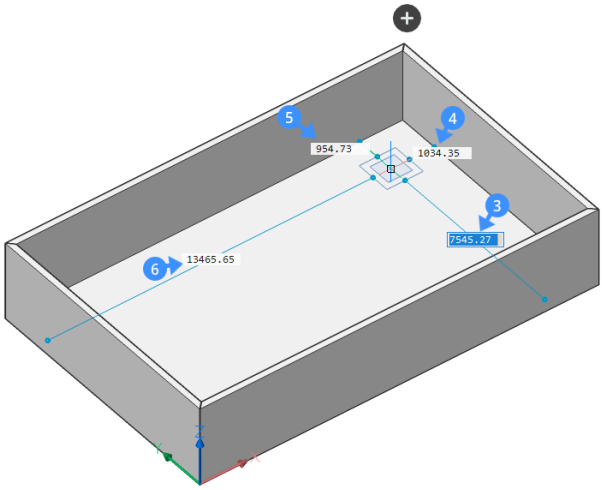


Specify the values of the room width and length in the Dynamic dimension field. Press the Tab key to switch between width and length and press Enter or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

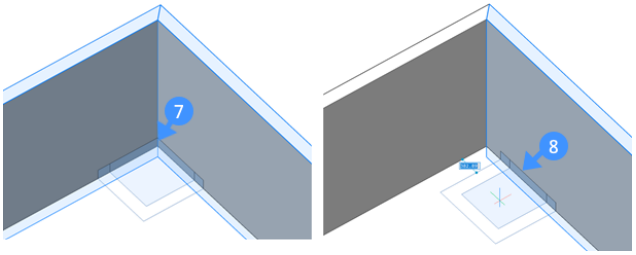
4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).

When the "auto adapt sizes" option is turned on, the dimensions of the walls of the new room will be the same as the wall you snap to.



If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



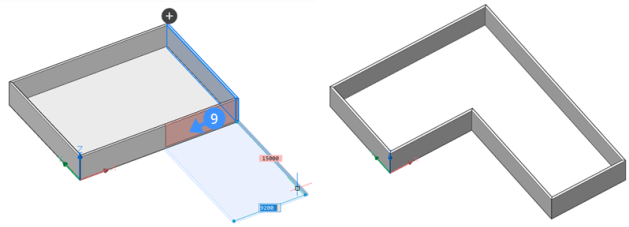
To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

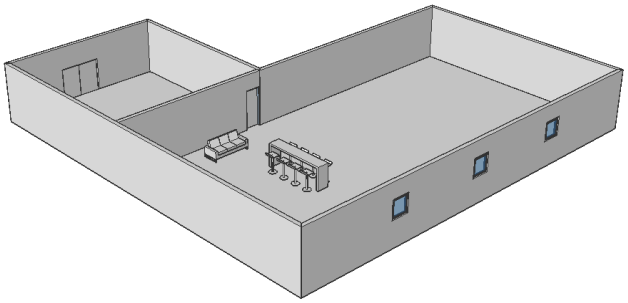
Nota: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.



20.3.21.7 Procedure: creating a story and a roof

Start with a similar configuration:



Go through the following steps to create two extra stories, a floating part and a flat roof.

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab or type BIMQUICKDRAW in the Command line.

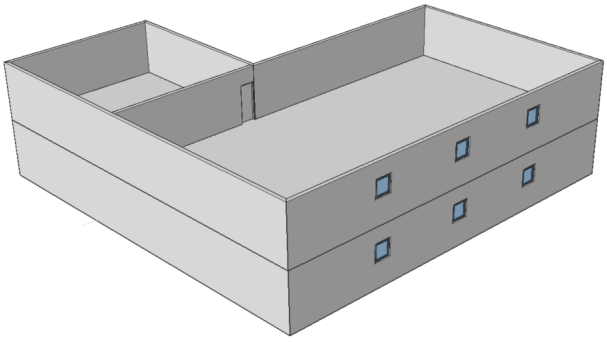
2 Click on the icon ( ) to see the options:

- Fully copy the top floor. All features will be copied (windows, doors, interior walls, and

furniture, excluding doors on the outer walls of the ground floor).

- Copy the slab and the outer walls of the top floor.
- Finish the building with a flat roof.

3 For the first story, choose the first option (Full copy).

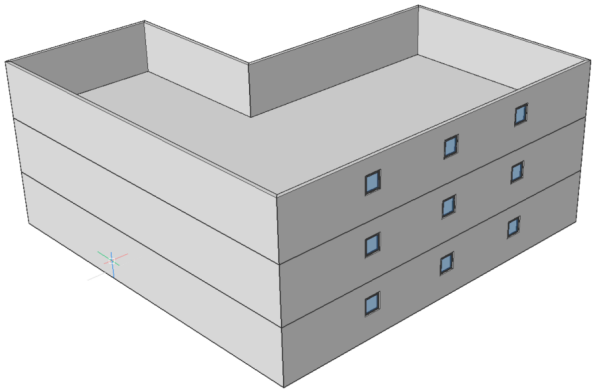


Nota: The interior wall and interior door and the windows in the exterior wall are copied. The door in the exterior wall on the ground floor is not copied.

Nota: The furniture is not copied in this case, because it is not assigned to the ground floor (see the **Properties** panel). If you want to copy the furniture, use BIMIFY to assign the furniture to a spatial location. The new furniture will be automatically assigned to the new spatial location.

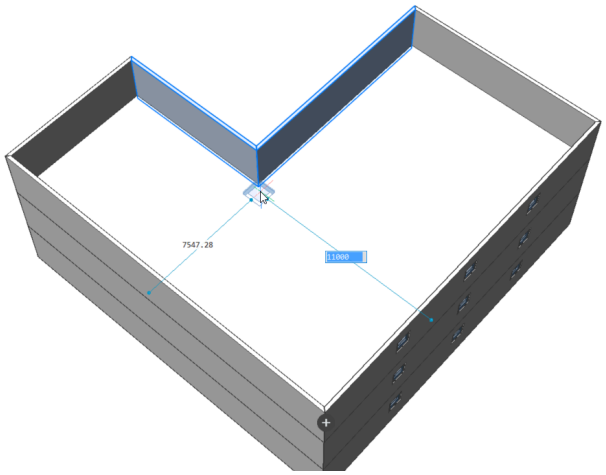
Nota: Walls inherit the spatial location of the slab on which they are created.

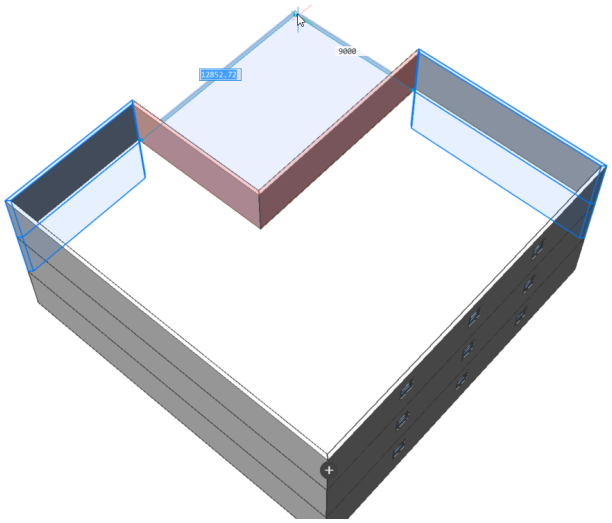
4 Repeat step 2. Choose the second option (Copy the slab and the outer walls) to create the top story.



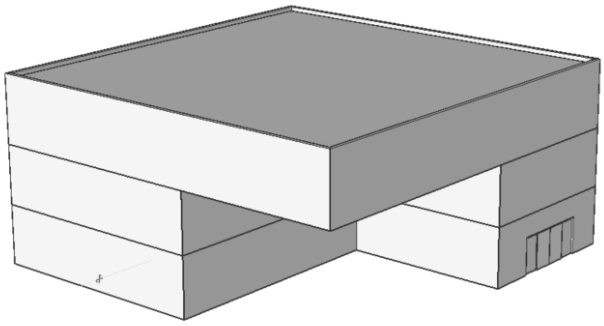
5 To make the overhang you should remove parts of the outer walls of the second floor. First, snap the Quickdraw cursor in the corner of the two outer walls, inside the building. Then click and drag the cursor outside.

6 The outer walls will turn red to indicate that the walls will be removed.



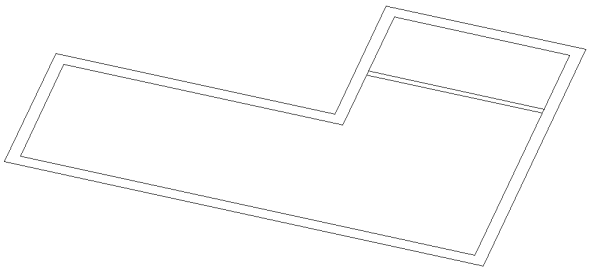


7 Use the same method as in step 2, selecting option 3 (Finish the building with a flat roof).



20.3.21.8 Procedure: using Quickdraw starting from a 2D layout

1 Open a 2D floorplan of a rectangular or L-shaped building.

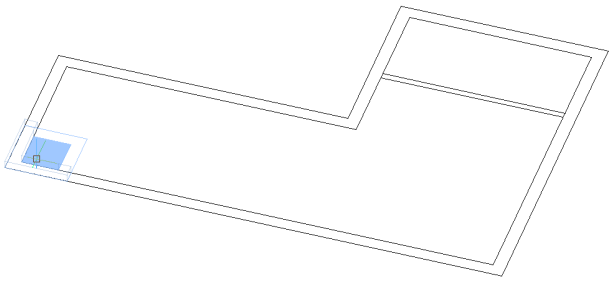


Nota: It is also possible to draw a simple 2D floor plan with the (POLY)LINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

Nota: The Quickdraw cursor now supports snapping to (double) lines inside XREFs.

2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

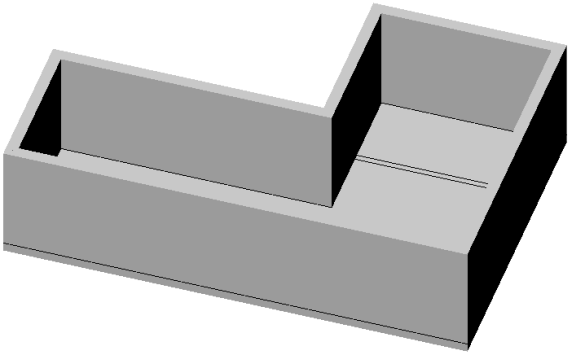
3 Hover over one of the outer walls with the cursor.



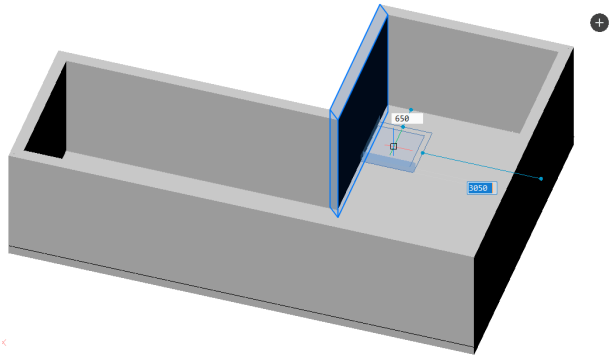
Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

Nota: If the lines in the (imported) 2D floorplan are not optimal (e.g. not exactly X- or Y-aligned), the QuickDraw-cursor will not snap to it (as it is perfectly X-Y-aligned). In that case, use the Optimize command to make sure all 2D lines exactly X- or Y-aligned.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

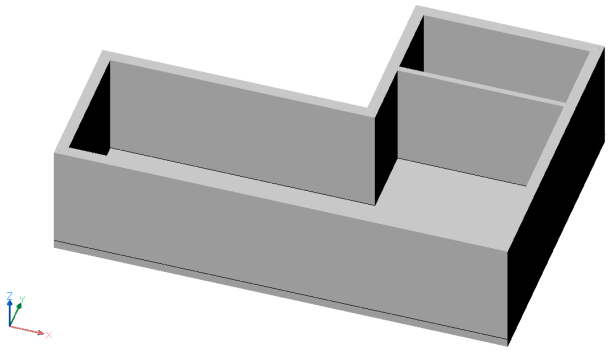


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



Nota: you can use the same method as in step 2 and 6 of the previous procedure to add floors or finish the building with a flat roof.

20.3.22 Using drag

20.3.22.1 Commands

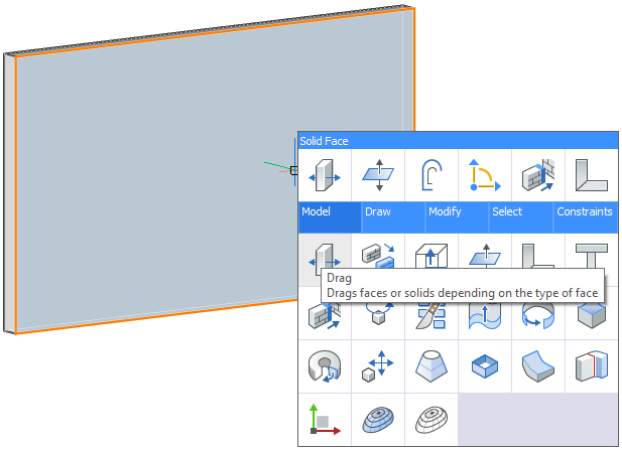
DRAG

20.3.22.2 About

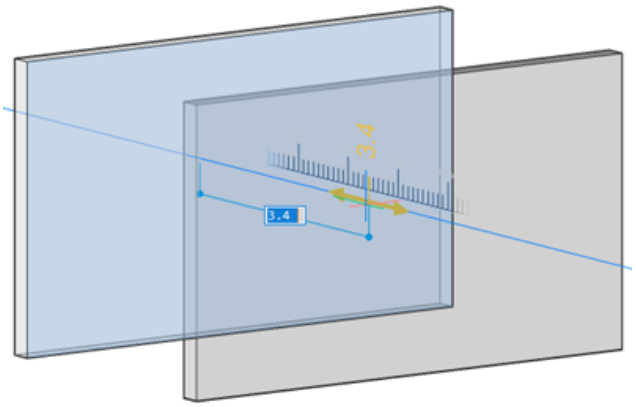
The **Drag** tool allows you to move walls, slabs or linear elements on the highlighted face of a solid. For instance, the **Drag** tool can be used to move linear solids by highlighting/selecting the end or side face of a linear solid. Depending on the modeling operation, the connection between solids can be either preserved while dragging one or more solids at a time or solid (s) can be moved independently by disjoining the connectivity to other solid(s).

20.3.22.3 Procedure: using drag on a 3D solid

1 Highlight/select any faces of the solid and select **Drag** from the **Quad** > **Model** tab.

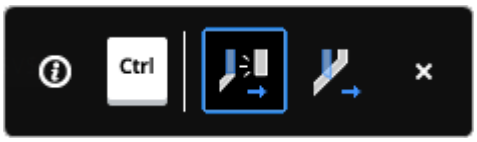


2 Move the cursor to adjust the position of the moving solid in the dynamic dimension field, then click or enter a precise value in the dynamic dimension field and press Enter.



Nota: A ruler appears to visually represent the distance over which you will move the entity.

3 The Hotkey Assistant appears at the bottom of the screen showing the available options.

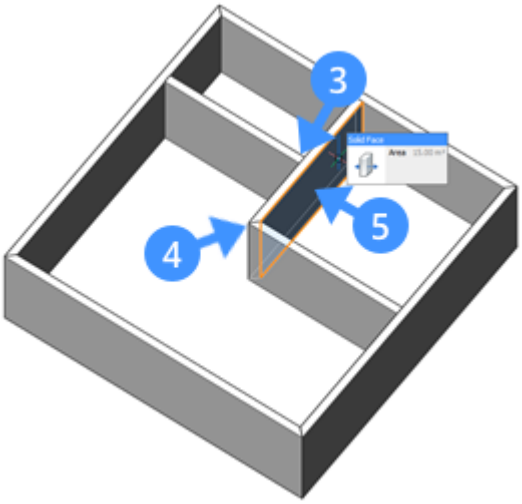


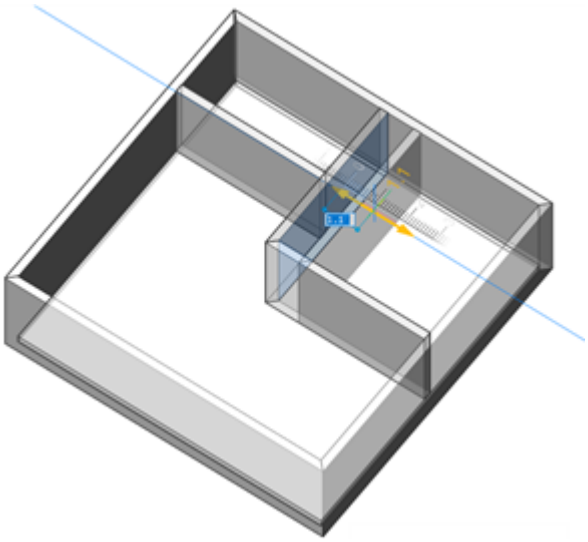
Nota: The current mode is indicated by a blue frame.

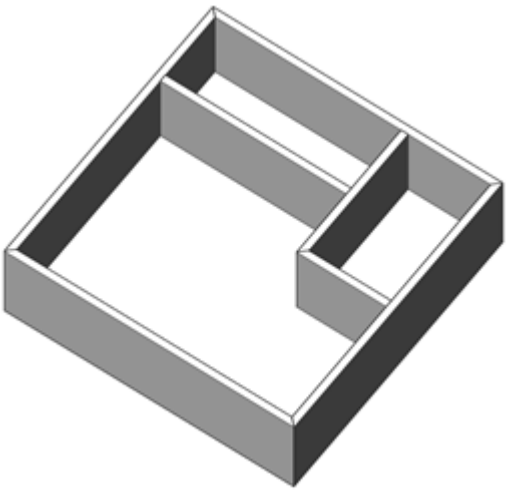
Hit the CTRL-key to toggle the connectivity mode. Alternatively, type D+Enter to disable connectivity or type E+Enter to enable connectivity.

- Select the **Enable connectivity mode** option, to move the solid while preserving the connections between solids.

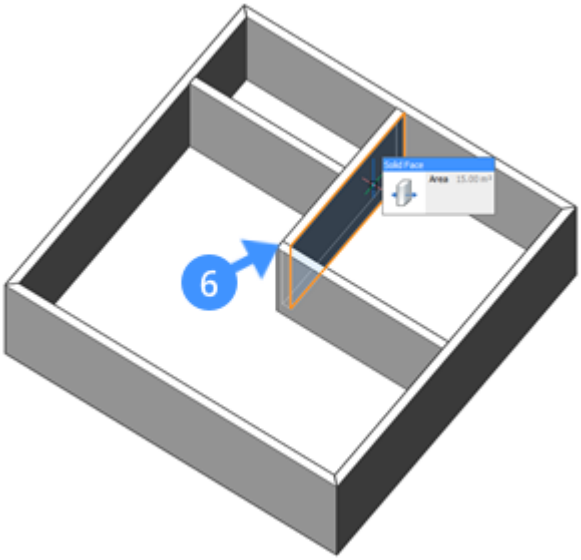
In the following illustration, the solid behavior is shown before and after applying the DRAG in **Enable connectivity mode**. The connectivity between walls, i.e. T- (3) and mitered (4) connections are preserved after dragging an inner wall surface (5).

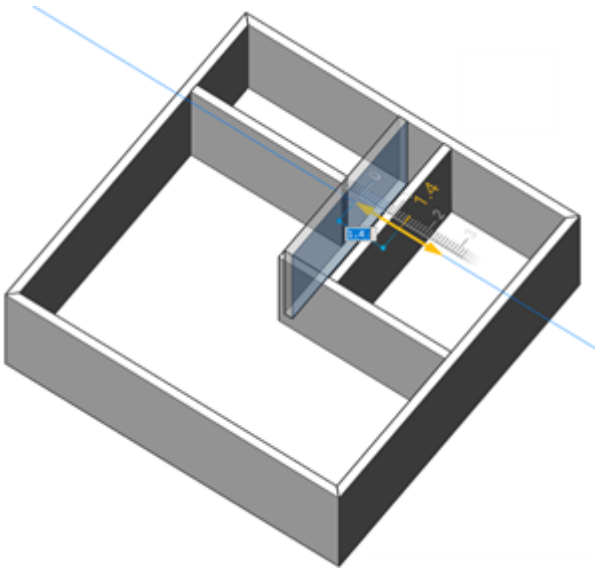


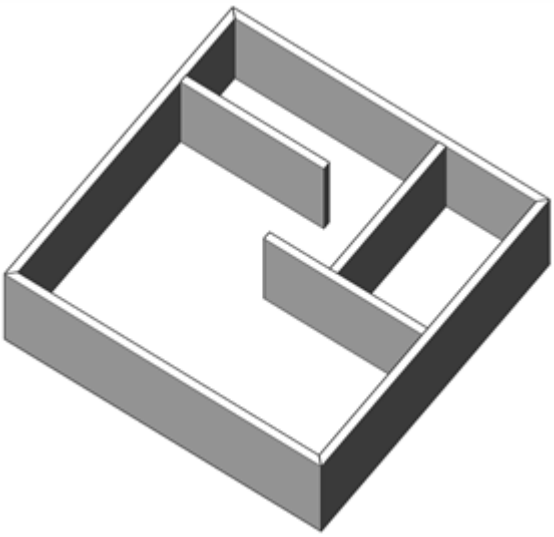




- Select the Disable connectivity mode (default) option to move the solid while breaking the connections between solids.
- In the following illustration, the solid behavior is shown before and after applying the Drag in "Disable connectivity mode". The connectivity between walls, i.e. T- (3) and L- (6) connections are disjointed after dragging an inner wall surface (5).







20.3.23 Using parametrize

20.3.23.1 Commands

PARAMETRIZE, PARAMETERSPANELOPEN

20.3.23.2 About

Parametrizing is a process that automatically adds constraints and parameters to a set of 3D solids, without any manual work. For each unique value, a specific parameter is created. The constraints and parameters are represented by their name and expression in the **Mechanical Browser**. Each parameter can be animated to show what the parameters correspond to in the model. The parametrized geometry becomes controllable in terms of shape and size. When one of the parameters is changed, the geometric model will be updated automatically.

20.3.23.3 Procedure: parametrizing a component automatically

1 Open the geometric model that represents your component.

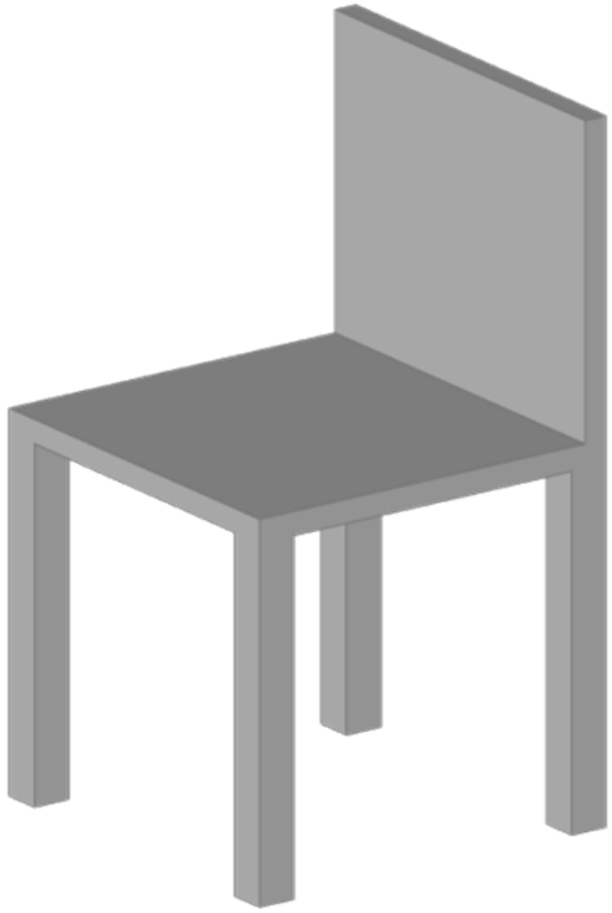
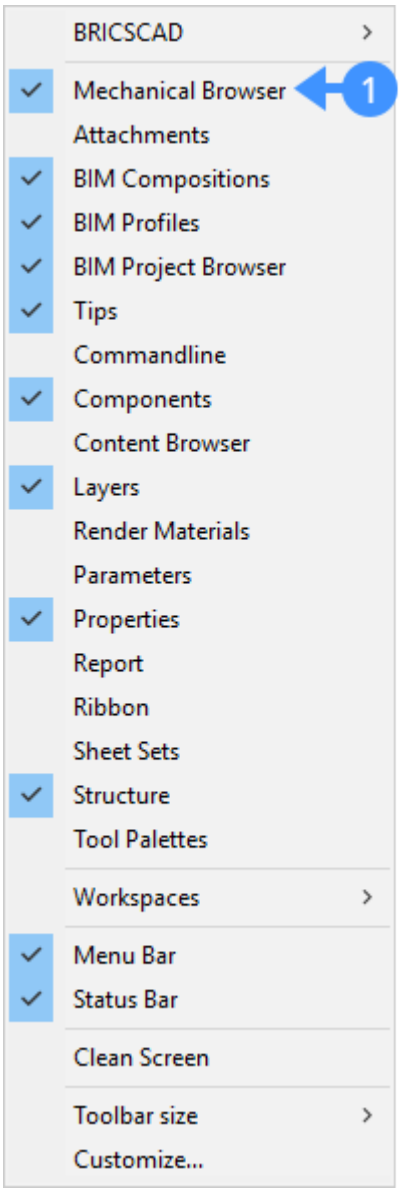


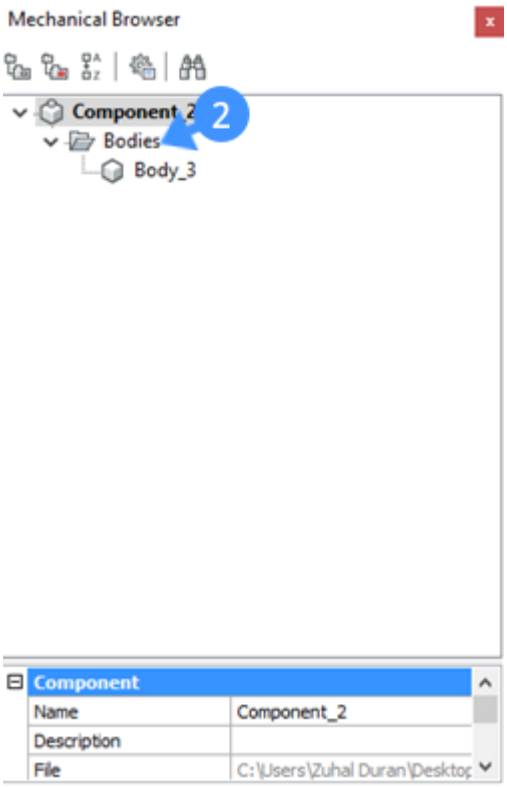
Figure 1, a geometric model of the sample chair.

2 Open the **Mechanical Browser** by clicking its icon in the **Toolbar** panel or type the MECHANICALBROWSEROPEN in the Command line.

Nota: If the Browser or Parameters icons are not displayed in the **Toolbar** panel, place the cursor on a toolbar, then right-click to open **Panels** and choose **Mechanical Browser** (1) in the context menu.



If the model does not contain any parameters and constraint it will display only the 3D solids under the **Bodies** (2) section. If the model contains any constraint or parameter, the parameterized tool asks whether you want to reuse the existing ones.



3 Select the component geometry you want to parametrize.

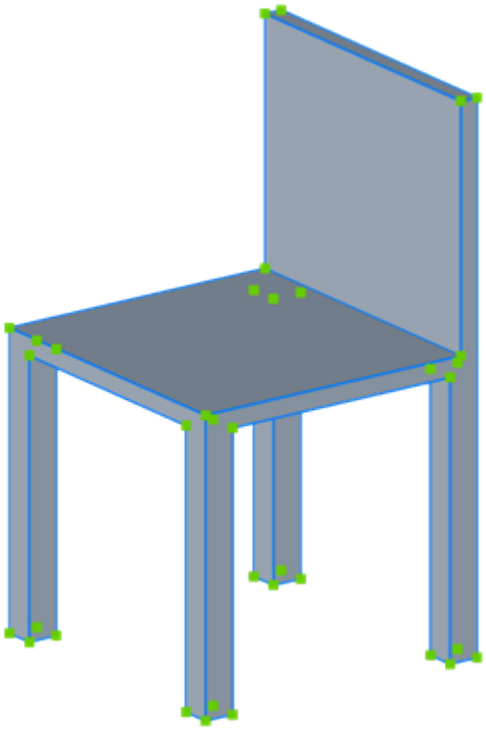


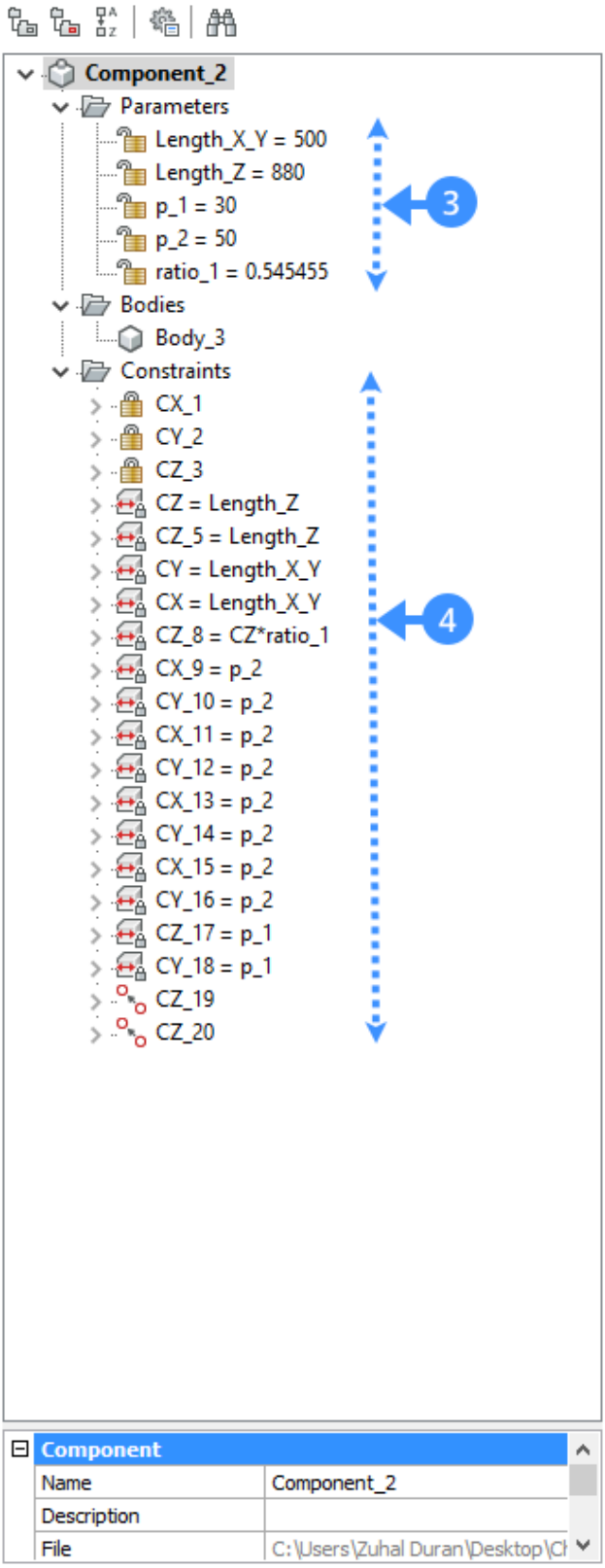
Figure 2, selected object.

4 Launch the **Parametrize** tool in the **Constraints** tab from the Quad.

Or type PARAMETRIZE in the Command line.

A set of parameters and constraints are defined automatically.

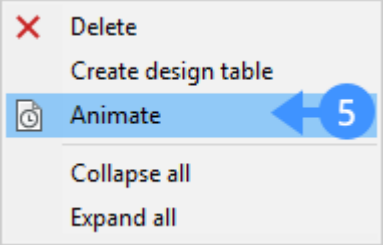
These parameters and constraints are shown in the **Mechanical Browser** (3 and 4).


Component_2
Parameters
Length_X_Y = 500
Length_Z = 880
p_1 = 30
p_2 = 50
ratio_1 = 0.545455
3
Bodies
Body_3
Constraints
CX_1
CY_2
CZ_3
CZ = Length_Z
CZ_5 = Length_Z
CY = Length_X_Y
CX = Length_X_Y
CZ_8 = CZ*ratio_1
CX_9 = p_2
CY_10 = p_2
CX_11 = p_2
CY_12 = p_2
CX_13 = p_2
CY_14 = p_2
CX_15 = p_2
CY_16 = p_2
CZ_17 = p_1
CY_18 = p_1
CZ_19
CZ_20
4
Component
Name
Component_2
Description
File
C:\Users\Zuhal Duran\Desktop\Cl

Nota: **Parametrize** creates Distance, Radius, Rigid Sets, Fix, Coincidence, Concentric constraints depending on the geometric model.

Nota: Perpendicularity and tangency can be set (if they are applicable) by changing the DMRECOGNIZE system variable in the **Settings** dialog box.

5 To check easily which parameter corresponds to what, right-click one of the parameters, and choose **Animate** (5) in the context menu.



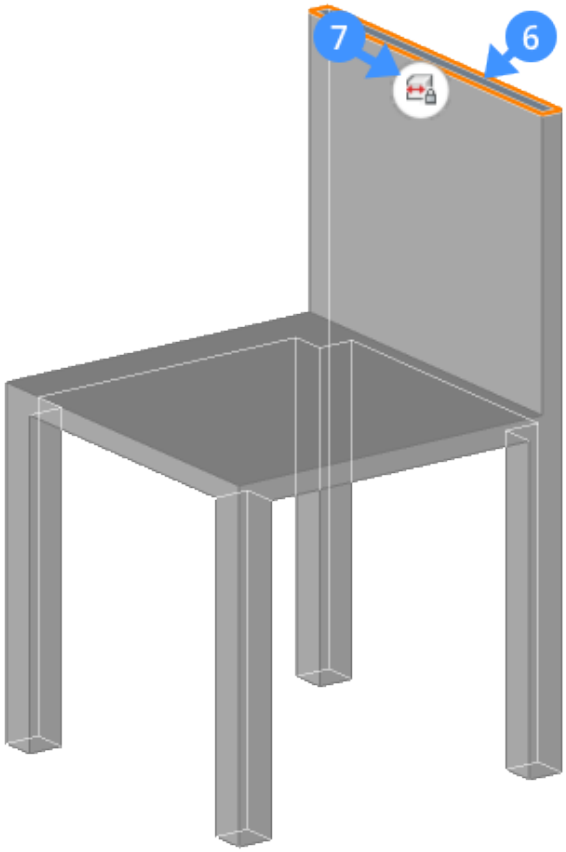
Nota: In the attached .dwg file after parametrizing the chair geometry, parameters and constraints will be assigned and shown in the **Mechanical Browser**. Each parameter corresponds to one measurement in the chair geometry. For instance, **Length_X_Y** controls the width and depth, **Length_Z** controls the overall height, **p_1** controls the seat thickness **p_2** controls the leg thickness **ratio_1** controls the ratio between seat height and overall height.

20.3.23.4 Procedure: changing the parameters and constraints in the parameters manager

Nota: After the auto parametrizing, it's possible to change what you do not find clear about the parameters and constraints using the parameters manager.

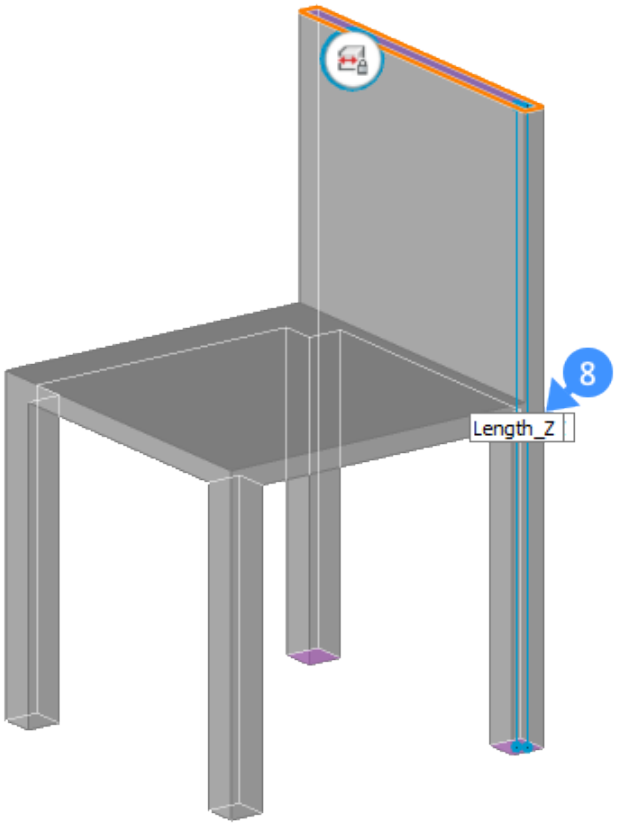
1 Select one of the faces (6) of your component. The constraint type (e.g. distance, fix) regarding the

face is displayed.

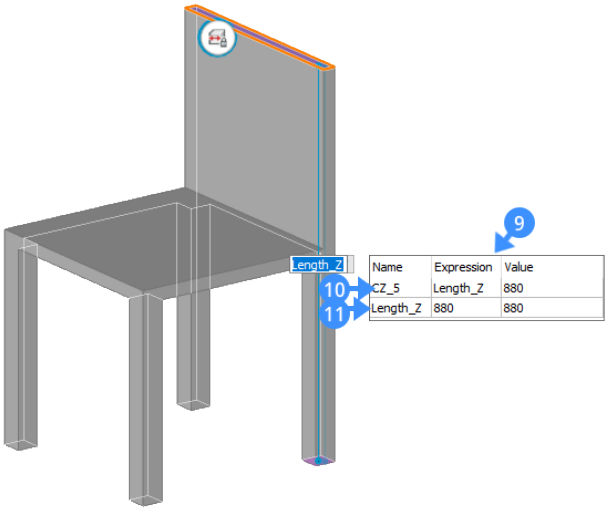


2 Click the distance constraint (7) icon on the element. The editable dimension(s) is shown inside the box.

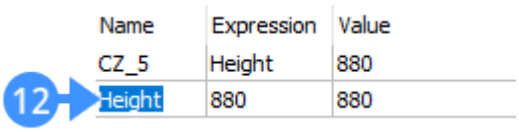
3 Click the editable dimension box (8).



A parameter manager displays (9). The parameter manager shows the expressions of the distance constraint (10) and the parameter (11) depending on the selected face.



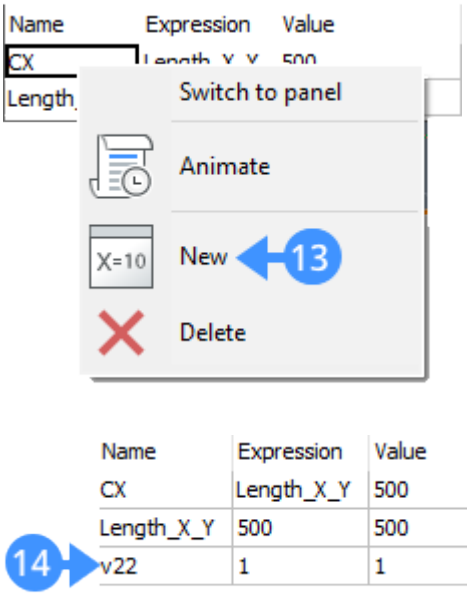
4 Change the parameter name by clicking its field (12) in the parameters manager.



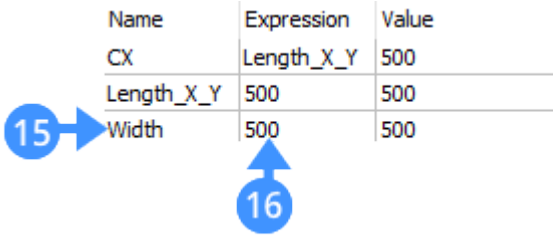
5 (**Optional**) A new parameter can be defined using the **New** option in the parameter manager's context menu.

6 To do so, right-click the parameters manager and select **New** (13).

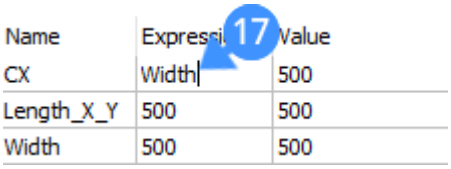
7 A new parameter is added to the list (14).



8 Specify a **Name** (15) for the newly created parameter and enter an expression using the **Expression** field (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).



Nota: The above procedure can also be done using the **Mechanical Browser**.

20.3.24 Working with parameters and constraints

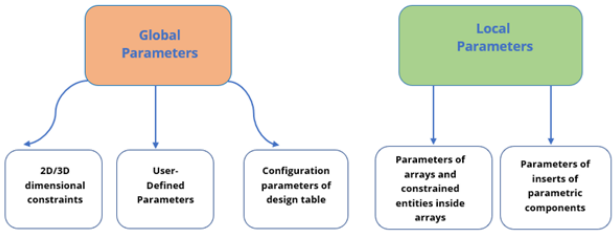
20.3.24.1 Parametric modeling

In BricsCAD, parametric modeling is done through the use of geometrical and dimensional constraints. A rich toolset controls the 2D and 3D geometric model with a set of parameters. Each parameter will be

displayed with a value in the **Mechanical Browser**. The value from the different parameters can be linked together via expressions. When one of the parameter values is changed, the geometric model is updated automatically. The toolset allows BricsCAD users to add parametric behavior to any geometry and easily explore design intent.

Nota: In BricsCAD, there are no parent-child dependencies between geometric elements. For instance, if you change a sketch used to create an extruded 3D solid, the solid is not changed accordingly. Any kind of dependencies can be created using the parameters and expressions.

There are two types of parameters: Local parameters are attached to a particular entity. Global parameters are not attached to a particular entity.



20.3.24.2 Working with constraints

If you create some components (e.g. windows, doors,...) in your BIM model, you can parametrize them by applying 3D constraints. Defining constraints allows you to control the shape and size of the elements. Together with constraints, parameters determine the positions of entities through an expression.

There are two types of 3D constraints in BricsCAD: one specifies the size of the entities, the other locate their positions.

20.3.24.3 Geometrical constraints

Geometric constraints allow you to control the position of 3D entities with respect to each other.

Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

The following table shows the 3D geometrical constraints.

Fix	Keeps solids, edges or faces of solids in-place in the drawing.
Coincident	Applies a coincident constraint between two edges, two faces or an edge and a face of two different solids.
Concentric	Keeps two cylindrical, spherical or conical surfaces centered.

Parallel	Keeps the two faces of a solid or of different solids parallel.
Perpendicular	Keeps the two faces of a solid or of different solids perpendicular.
Tangent	Keeps a face and a curved surface of different solids tangent.
Rigid set	Makes a set of entities or sub-entities a rigid body.

20.3.24.4 Dimensional constraints

Dimensional constraints allow you to control the sizes of 3D entities in drawings and the distances between them.

Toolbar: **Parametric** > **3D constraints**

Quad: **Constraints**

Ribbon: **Parametric Tab** > **3D constraints**

The following table shows the dimensional constraints.

Distance	Creates a distance between two sub-entities.
Radius	Creates a radius to cylindrical surfaces or circular edges.
Angle	Creates an angle between two faces of a solid or of different solids.

20.3.24.5 Setting an expression to a parameter

In BricsCAD, you can set an expression to any parameter, be it global or local. As an example, you can create a simple expression that only contains a number or a name of a global parameter and apply it to the related geometry in the drawing. You can also use more complex formulas that include the use of standard operators and functions.

NOTES	• Local parameters cannot be referenced by names in expressions. • The constants Pi=3.14... and e=2.72... can be used in expressions. The constants names are not allowed to be used as parameter name and name of 3D constraint.

The following table shows the operators that can be used in expressions.

Addition (+)	Produces the sum of numeric values.
Subtraction or Negative (-)	Subtracts the two numeric values.
Multiplication (*)	Multiplies the numeric values.
Division (/)	Divides two numeric values.
Exponentiation (^)	Calculates the exponential value of the given numbers.
Modulo or Remainder operator (%)	Gives the remainder after the division of one numeric value to another e.g. The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1.

The following table shows the functions and the syntax which can be used in expressions.

Cosine	cos(expression)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)

Square root	sqrt(expression)
Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2) *
Smallest element in array	min(expression1;expression2) *
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base	10 log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow
Random decimal, 0-1	Random(expression1;expression2) *

*Use the list separator character as defined on your system: , (comma) or ; (semicolon).

20.3.24.6 Parametric blocks

Parametric Blocks refer to blocks that have parameters that determine the size and shape of the 3D geometry. Once the parametric block is created, it can be inserted as a block reference in the drawing. The parameters of your block can be changed after you insert it in your model. Thus, no need to modify the parametric block file individually. The parametric blocks also enable you to use the same block in different sizes and in different shapes in your model.

20.3.24.7 Procedure: parametrizing a custom component

The window that was created in the previous lesson is used to demonstrate how the parametric design works. This window consists of three solids: subtractor, fixed frame, and glass pane.

Step 1: Before starting to add constraints:

1 Open the **Mechanical Browser** with the MECHANICALBROWSEROPEN command, to control and manage the values of 3D constraints and parameters.

Nota: The **Mechanical Browser** allows to navigate through all the constraints and parameters in the drawing and to edit dimensional constraints.

2 Make sure that the **Enable selection of 3D solid faces** ( ) is toggled on in the Selection Modes To easily follow the steps, make sure Boundary Detection is toggled off.

Step 2: Applying fix constraints

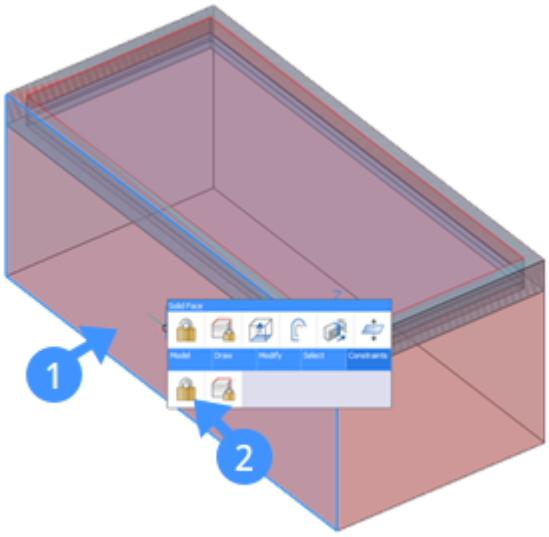
Command: DMFIX3D

1 Turn on the **BIM_Subtract** layer.

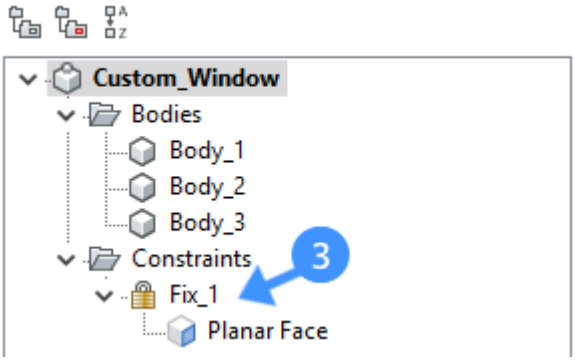
2 Select the entities (1) you want to add a fixation to.

3 (**Optional**) Press the TAB key to select obscured geometry.

4 Choose **Add Fixation** (2) in the 3D constraints command group in the Quad.



The fixation appears in **Mechanical Browser** as **Fix_1**.



Step 3: Applying rigid constraints

Command: DMRIGIDSET3D

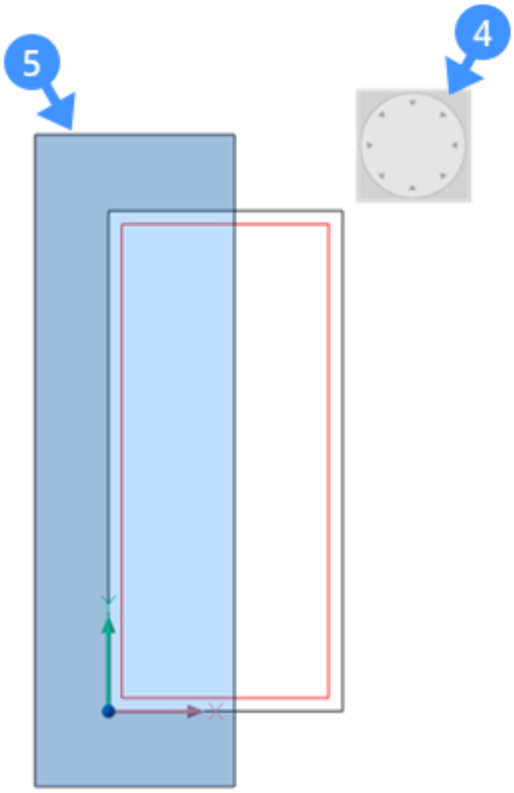
1 Select the entities to apply the **Rigid Set** constraints.

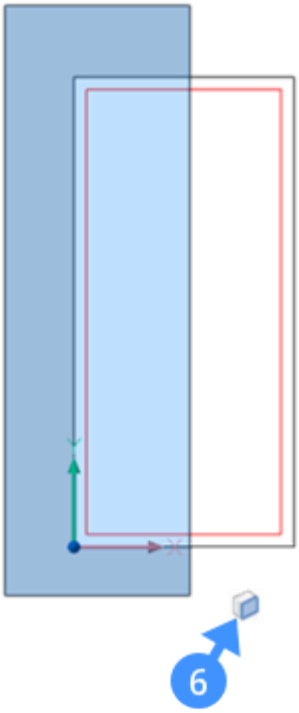
The entities remain in the same position with respect to each other when they are in a Rigid Set.

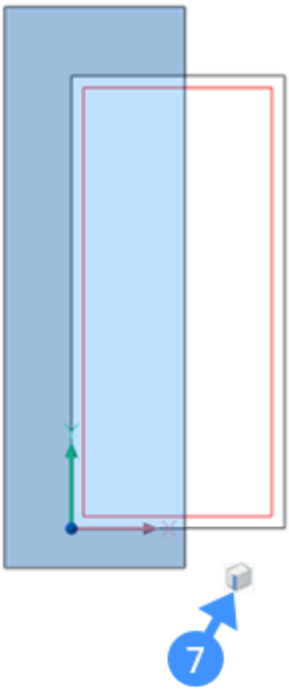
To select more than one sub-entity of the same type at a time, take the following steps:

- Use **Look From Widget** (4), by default it is at the top right of your screen, to change the view from 3D to TOP.
- Select the entities using the selection box (5). While drawing a selection box, press CTRL key to change the type of sub-entity selection. Every time you press CTRL key the type of the sub-entity selection will change. The type of the sub-entity can be seen under your mouse cursor, for instance, faces (6) or edges (7).

Nota: By default, the selection boxes see and select solids.


5
4


6



2 Choose **Add Rigid Set** ( ) in the 3D constraints command group in the Quad.

3 Repeat the same process for the other sides: top, bottom, and right side.

Step 4: Applying distance constraints

Command: DMDISTANCE3D

Dimensions of the window and the thickness of the glass pane.

1 Turn off the **BIM_Subtract** layer.

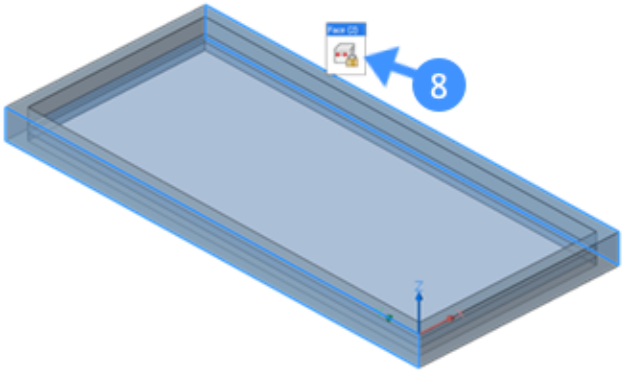
2 Select the first face.

3 The face highlights.

4 Hover over the second face.

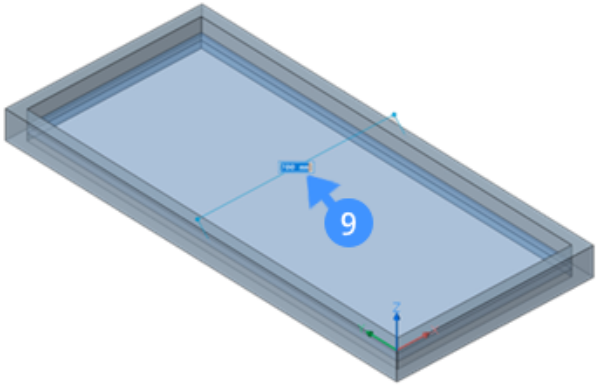
5 Hit the TAB key to highlight an obscured face.

6 When the second face highlights, choose **Add Distance** (8) in the 3D Constraints command group in the Quad.



You are prompted: Specify distance value or [Geometry-driven] <xxx.xx>:

7 Do one of the following:
   - Press Enter key or right-click to accept the current value.
   - The current value displays in the dynamic dimension field (9) when the **Dynamic Dimensions** (DYN) is active.



   - Type a value in the Command line.

8 The distance value between the two surfaces appears in the **Mechanical Browser**.

Nota: To remove any constraint in your model, select the constraint in the **Mechanical Browser**, e.g. fix, rigid set..., and press the Delete key or right-click and select the **Delete** option.

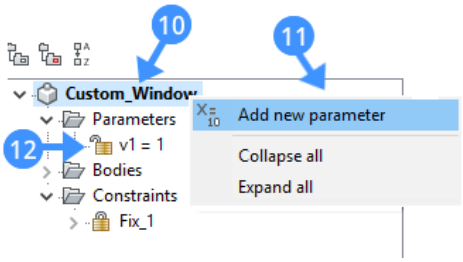
Step 5: Applying parameters

We will create a parameter that controls the width of the window.

1 Right-click the file name (10) in the **Mechanical Browser**.

2 Select **Add New Parameter** (11) in the context menu.

The parameter settings grid is created and the parameter is added in **Mechanical Browser** as **v1 = 1** (12).



3 Select the parameter, then edit its properties in the settings grid.

4 The following illustration shows the properties of a **Width** parameter of the custom window. The table below demonstrates each property.

Parameter	
Name	W
Expression	700
Value	700
Geometry-driven	No
Description	Width
Exposed	Off
Units	mm

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 Dati dell'Edificio

20.4.1 Analytical model

20.4.1.1 Commands

BIMANALYTICALMODEL

20.4.1.2 About the BIM analytical model

An Analytical Model is in essence a simplified wireframe model that only contains the load-carrying building elements. The linear building elements like columns, beams and members are represented by lines. The planar building elements like walls and slabs are not yet supported in the analytical model.

In BricsCAD, the BIMANALYTICALMODEL command allows you to generate an analytical model from a full 3D model of a structure. The command starts with an automatically generated proposal but allows you to further adapt the model to your desire. The automatic proposal will try to minimize rigid links (eccentricities) as much as possible by shifting, extending and even rotating some axis lines. The number of such changes can be controlled by the deviation settings.

You can modify the generated proposal afterward by using the options when you select an element (nodes or axes). To propagate the effect of a local change further you can use recalculate. This will launch a global solution that again tries to minimize rigid links but will respect the changes you made.

After editing the model, you can export it to an IFC file and a CIS/2 file. This analytical model can then be used for further structural analysis after import in a dedicated analysis software package.

It's important to emphasize that the input model has to be fully classified since the BIMANALYTICALMODEL command assumes that all the load-carrying elements are indicated in the building element properties.

In the procedures, the analytical model is explained with an example model. You can find this model in the appendix.

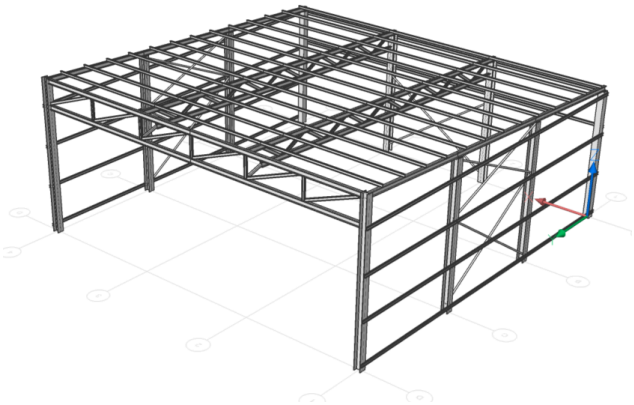
20.4.1.3 Procedure: creating an analytical model with strict deviation settings

1 First, you have to classify your BIM model. This is necessary for using the BIMANALYTICALMODEL command.

1 Use the BIMIFY command on your model or classify elements manually.

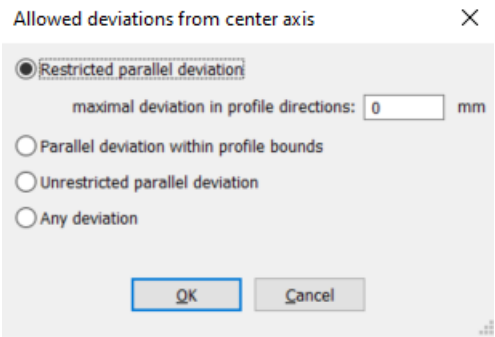
You are prompted: *run Bimify on entire model or [Advance] <model>*

2 Press Enter if you want to classify everything in your model or type **A** if you want a selection of elements.



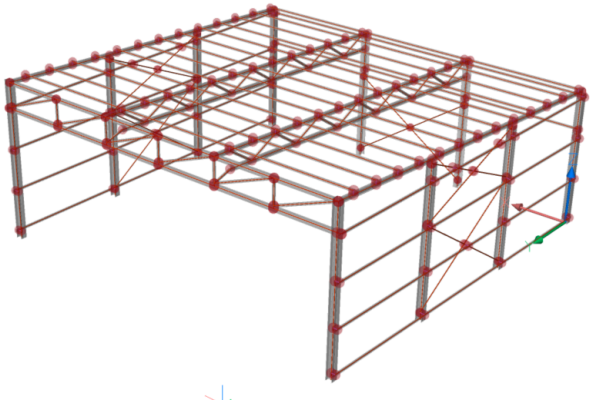
3 Launch the BIMANALYTICALMODEL command in the Command line.

4 A pop-up window opens.

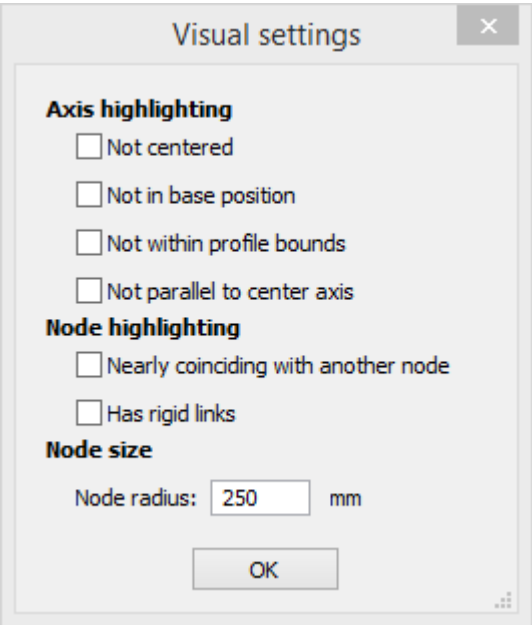


This window allows you to set the level of freedom used when generating an analytical model. When hovering over one of the options, a tooltip shows you what type of restrictions are imposed for that option. In this case, choose the **Restricted parallel deviation** with a maximal deviation in profile directions of 0 mm. No axis is allowed to deviate from its central position in the profile.

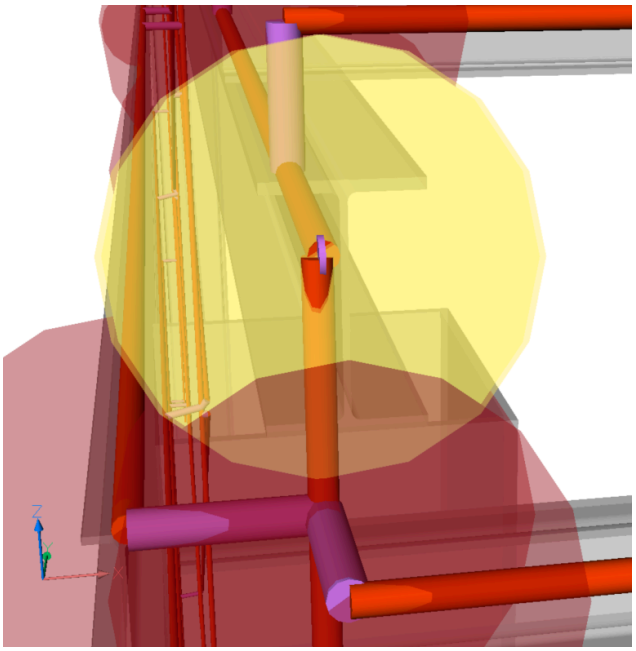
5 An interactive preview of the analytical model is constructed based on the chosen global deviation setting.



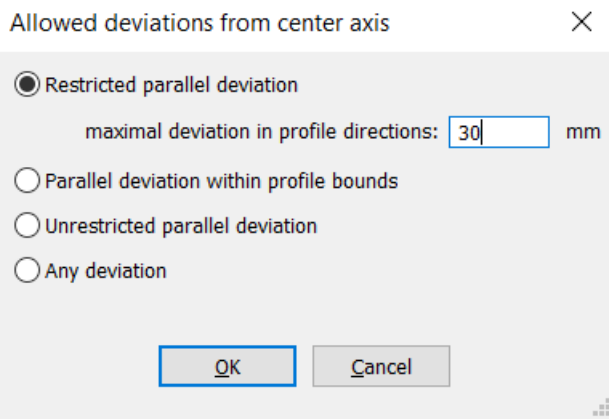
Also, the visual settings window appears. In this window, you can check and uncheck different features of axes and nodes to highlight (in yellow) the axes and nodes which correspond to those features. This allows you to explore the current properties of the axis and nodes of the analytical model. Based on what you find you can determine if any adjustments need to be made. You can also change the size of the disks representing the nodes. This is useful in combination with the node highlighting, which can be used for example to find nearly coinciding nodes.



6 Select the checkbox **Nearly coinciding with another node** in the **visual settings** window. You will see that the center axes at these nodes do not align perfectly. As a result, small rigid links (eccentricities) had to be introduced. This is because of the previous setting in step 4. These small rigid links are displayed in purple.

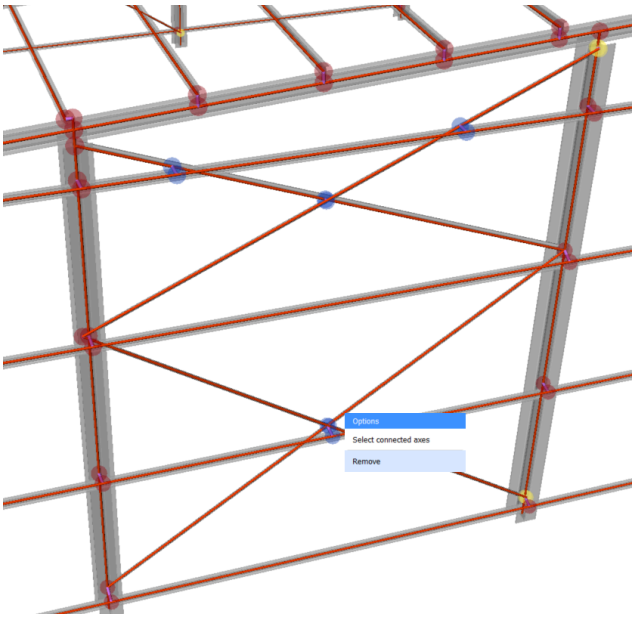


To avoid this, restart the command with a slightly more flexible setting. In the **Allowed deviations from center** window, choose again for **restricted parallel deviation** but this time with a maximal deviation in profile directions of 30 mm.

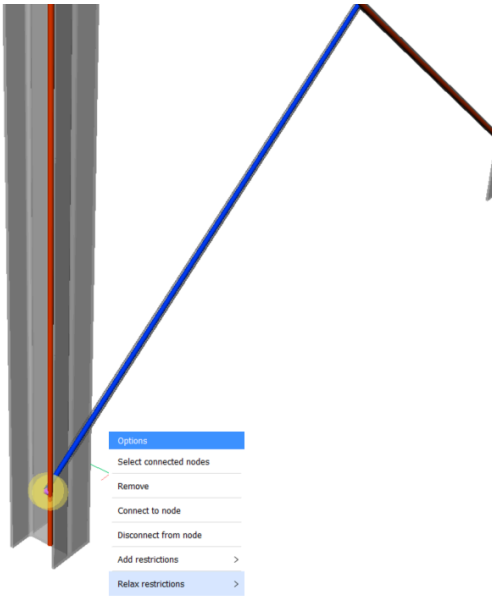


7 For the detection of nodes in the preview of the analytical model, the proximity of structural elements is used. Because of this, it is possible that there arise nodes in the model where tie rods cross. You can easily remove these nodes by clicking on the node and select **Remove**.

Nota: You can select several nodes at once using box selection.

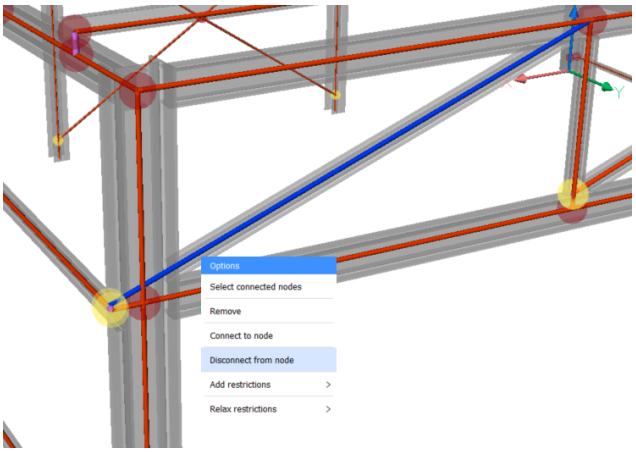


8 If you take a look at the remaining coinciding nodes, you see that the tie rods are not aligned perfectly with the center lines of the columns and small rigid links are formed. You can remove these rigid links by relaxing the position of the axes of the tie rods to a non-parallel position. You can do this by clicking on the tie rod, click **Relax restriction** and click **Non-parallel**. The extra freedom will be used to remove the rigid links by moving the axes of the tie rods a bit.



9 The only coinciding nodes that are left are at the front truss of the hangar structure. If you take a look at the side of the front truss, you see that a tilted member is connected to the horizontal beam rather than the column.

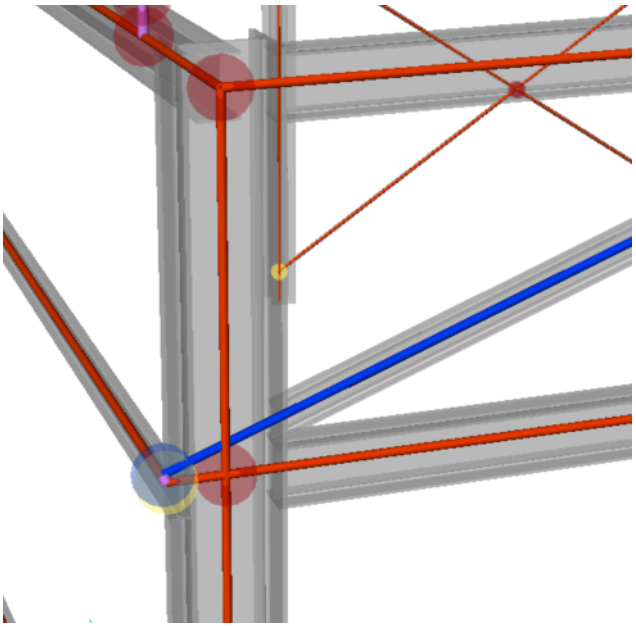
10 You can disconnect the tilted member by clicking on it and choose **Disconnect from node**.



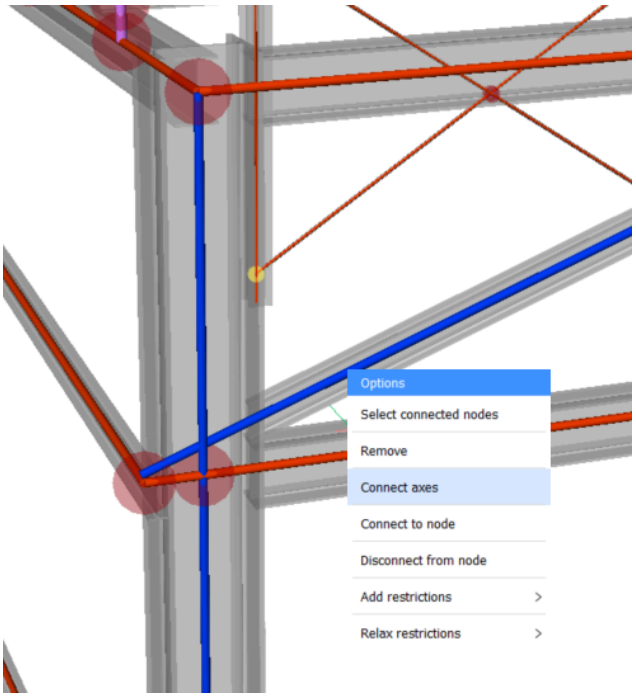
You are prompted:*Select node disconnect from axes [Ok]*

Select the node you want to disconnect from.

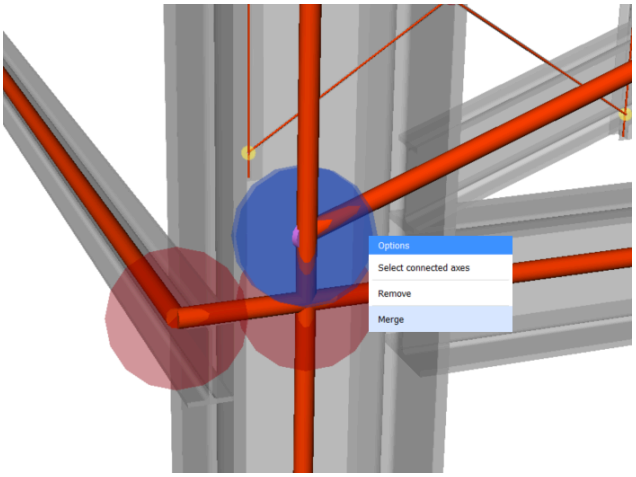
Type **O** to choose Ok.



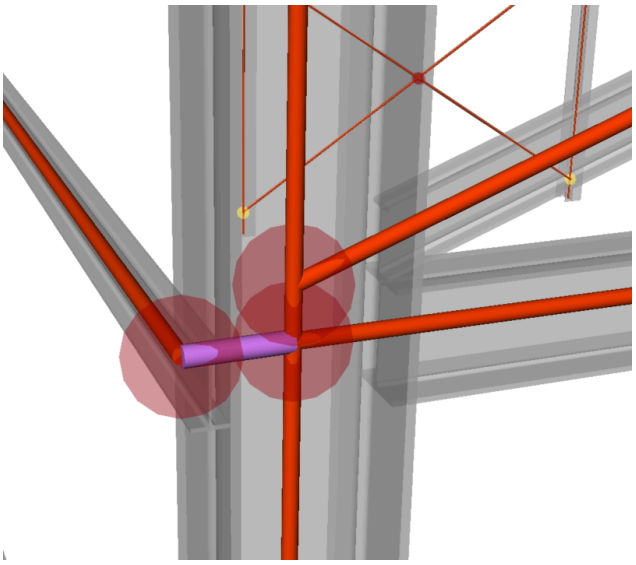
Now select the two axes that you want to connect and click **Connect axes**.



With the global deviation setting, you chose this can only be done by introducing a small rigid link again. You can remove this by merging the two nodes. Select both nodes and click **Merge**.

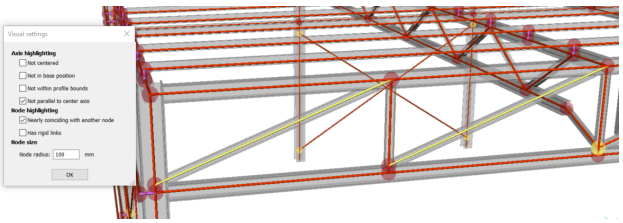


11 You see that the side member also is attached to the front beam instead of the column. Repeat step 9 for the side member.



12 You can remove the last coinciding nodes in the front truss by merging the nodes. This will shift the axes of the tilted members and introduces rigid links on the other end.

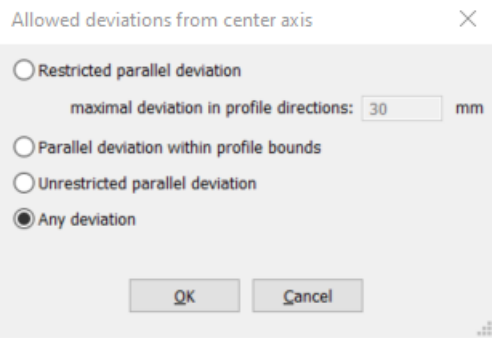
Nota: If you take a look at the axis **Not parallel to center axis** by using the **Visual settings** window, you will see that the tilted member in the front truss has been allowed to be non-parallel to the center axis. So, you could also remove the small rigid links by relaxing the tilted member to a non-parallel position like as we did for the tie rods.



20.4.1.4 Procedure: creating an analytical model with the least restrictive deviation settings

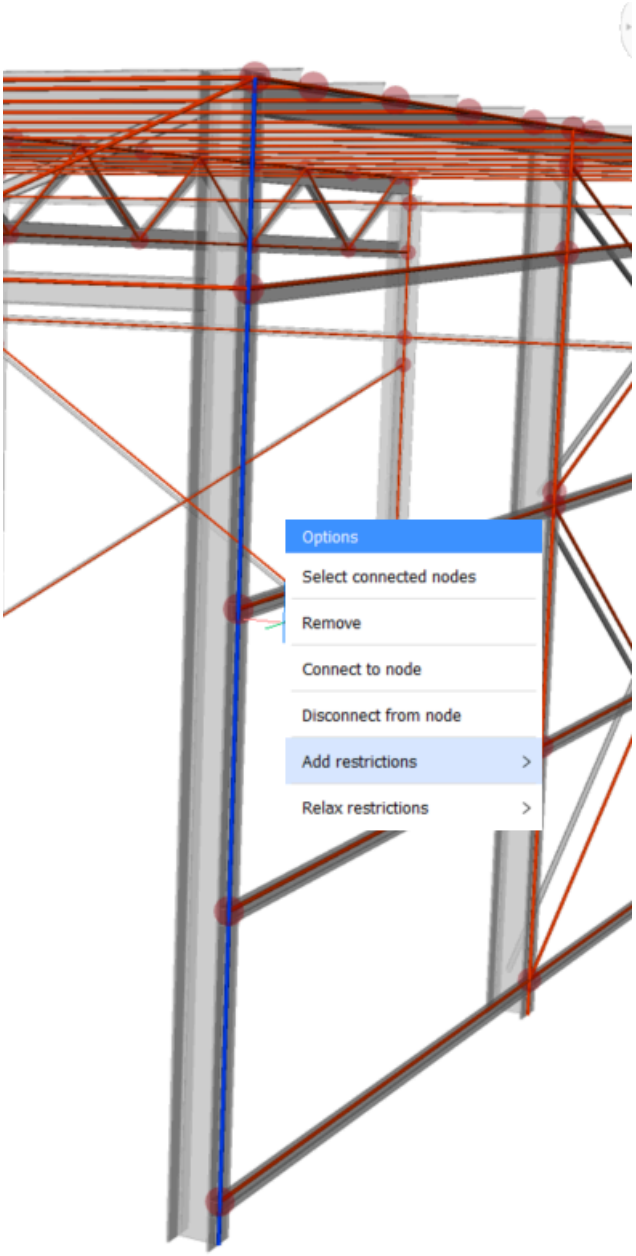
1 Repeat the steps from the previous procedure until step 4.

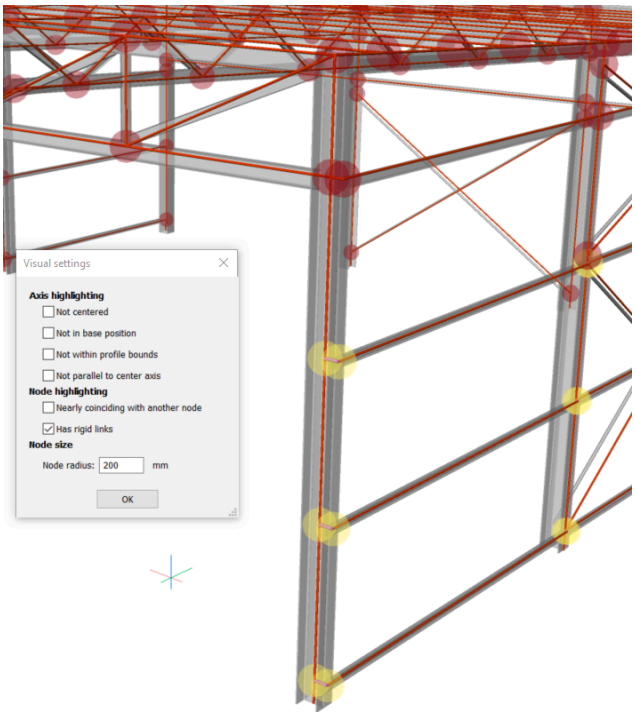
2 Choose **Any deviation** in the **Allowed deviations from center axis**. This allows all kinds of freedom to avoid any rigid links. In the preview of the analytical model, you will see that some axes have been tilted. For example, the small members in the trusses are not centered and the column axis is moved towards the profile bounds to find connections with smaller side members there.



3 Remove the unwanted tie rod nodes as in step 7 of the procedure with a strict deviation setting.

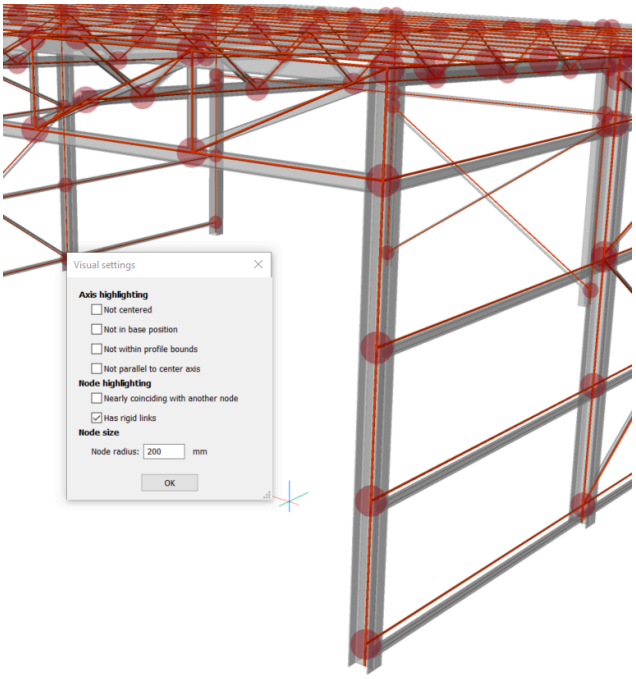
4 In the interactive preview of the analytical model, the column axes were moved towards the profile bounds. In this step, you will center the axis of the columns. You can do this by clicking on the axis of the column, click **Add restrictions** and click **To center**. This introduces rigid links very locally.


Options
Select connected nodes
Remove
Connect to node
Disconnect from node
Add restrictions
Relax restrictions



5 In the visual settings window, select the checkbox **Has rigid links**.

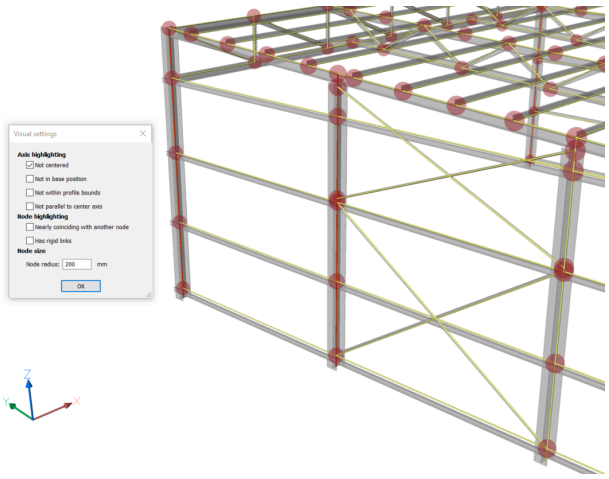
Type **R** in to choose **Recalculate**.



6 In the visual settings window, select the checkbox **Not centered**.

You will see that one of the columns is still not centered because the connection at its top includes not only the larger beams but also one of the smaller ones. You change this by imposing a center

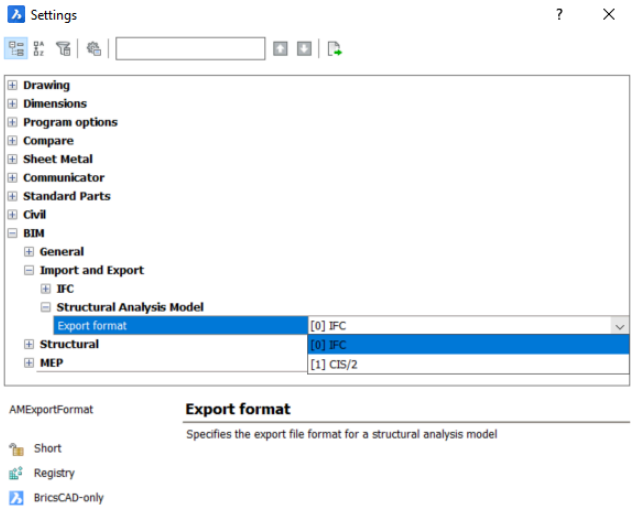
restriction on this column as well.



20.4.1.5 Procedure: exporting an analytical model

1 Before starting the command, open the **Settings** dialog.

2 In the Settings dialog, click the option **Structural Analysis Model** and choose the export format of the analytical model. You can choose between **IFC** and **CIS/2**.



3 Launch the BIMANALYTICALMODEL command.

4 Change the analytical model to your desire.

5 When you have finished the model, type **E** to choose Export and exit.

6 The document window appears where you can name and save the file in the desired folder.

Nota: There are some remarks about the exported result.

- The **node concept** used by BIMANALYTICALMODEL does not completely coincide with **CIS/2** and **IFC** node concept:

  In the BIMANALYTICALMODEL, only connections between linear elements are modeled with a node. For example, columns will not have a floor node if there is no connection to another

element there. During export, missing end nodes are added to be compatible with CIS/2 and IFC export.

In the BIMANALYTICALMODEL, a node always lies exactly on every connected axis. Eccentricities are modeled with explicit visual rigid links (purple bars in preview). During export, each cluster of BricsCAD nodes that are interconnected with rigid links is translated into a single connection node with eccentricities in **CIS/2** and **IFC**.

- The **axis concept** used by BIMANALYTICALMODEL does not always coincide with the **CIS/2** and **IFC** axis concept:

  In the BIMANALYTICALMODEL, there will be one single axis for each physical element. An axis can be attached to more or less than two nodes. During export, an axis that has multiple nodes along its length gets split up into several analysis members, each with one end and one start node, for the **CIS/2** and **IFC** export.
- The concept of **Base Position** in BricsCAD is similar to **cardinal points**. Base positions in BricsCAD form a subset of the cardinal points specified in the **CIS/2** standard. For now, only cardinal positions 1 to 9 are supported.
- In the analysis model, as calculated by BricsCAD, an axis can shift away from the center of the physical entity, essentially to any point within certain bounds. This means that the final position of the analysis axis does not necessarily coincide with a cardinal point or base position of the profile. During export, the section profile is linked to the analysis axis using the closest base position of the profile to the final analysis axis, using the cardinal point concept in **CIS/2** and **IFC**. This means that if you visualize the structure in a **CIS/2** or **IFC** viewer that shows physical members as extrusions of the profile section along the analysis axes, you will see that the structure shown there can slightly deviate from the original structure as modeled or imported into BRICSCAD. The structure as described by the **exported analysis model** is therefore an **approximation of the actual structure**.

model.dwg

20.4.2 AutoComplete BIM data

20.4.2.1 Commands

BIMAUTOMATCH

20.4.2.2 About the AutoComplete BIM data

Instead of assigning compositions to every element manually, BIMAUTOMATCH detects the difference between interior and exterior walls and their respective composition and automatically assigns it to other walls. It can also make distinctions between different floors, different sides of the building, walls with a balcony, etc.

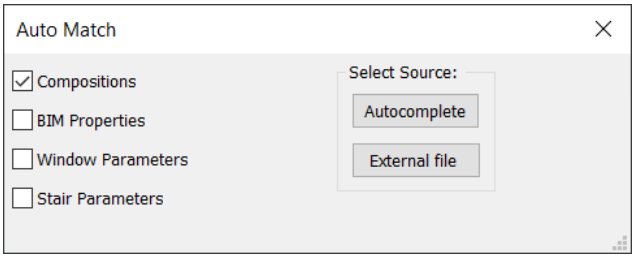
Nota:

Before you use BIMAUTOMATCH make sure that you have Bimified ( ) everything. AutoMatch works best with a fully Bimified model.

20.4.2.3 AutoComplete BIM data in one file

1 Do one of the following:
   - Select **Automatch** from the **Classify** tab in the Ribbon.
   - Select **Model** > **Automatch** in the Quad.
   - Type BIMAUTOMATCH and press Enter.

   You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:

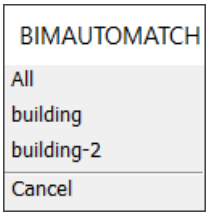


2 Select one of the following:
   - **Compositions**: Copy compositions.
   - **BIM Properties**: Copy BIM Properties.
   - **Window Parameters**: Copy window parameters.
   - **Stair Parameters**: Copy Stair Parameters.

3 Under **Select Source**, choose **Autocomplete**.

4 f the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

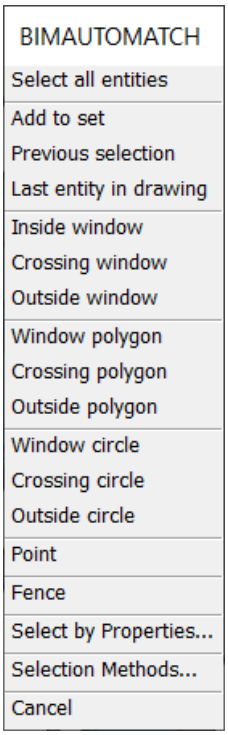
5 **You are prompted:**



   Select the building from the menu.

6 If BIMAUTOMATCH finds a logical distinguishing feature between objects it will automatically use this. When it can not, BIMAUTOMATCH asks you to define the source composition or window.

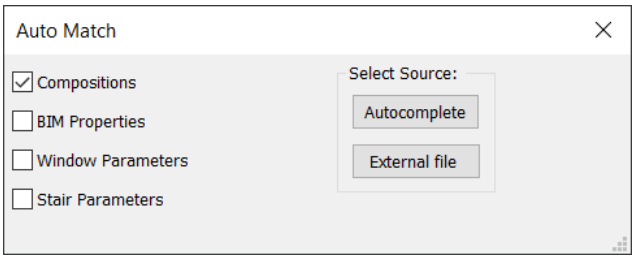
7 **You are prompted:**



20.4.2.4 AutoComplete BIM data with an external file

1 Do one of the following:
   - Select **Automatch** from the **Classify** tab in the Ribbon.
   - Select **Model** > **Automatch** in the Quad.
   - Type BIMAUTOMATCH and press Enter.

   You are prompted: Compositions/BIM Properties/Window Parameters/Stair Parameters:



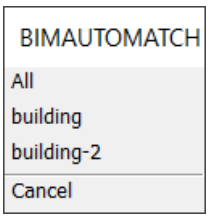
2 Select one of the following:
   - **Compositions**: Copy compositions.
   - **BIM Properties**: Copy BIM Properties.
   - **Window Parameters**: Copy parameters of windows.
   - **Stair Parameters**: Copy properties of stairs.

3 Under **Select Source**, choose **External file**.

4 Select source file and click **Open**.

5 If the file contains multiple buildings, BIMAUTOMATCH asks you to define the source building.

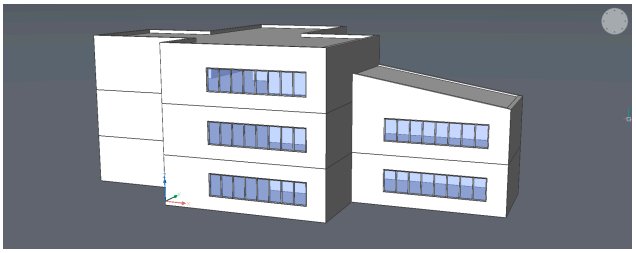
6 **You are prompted:**



7 Select the building from the menu.

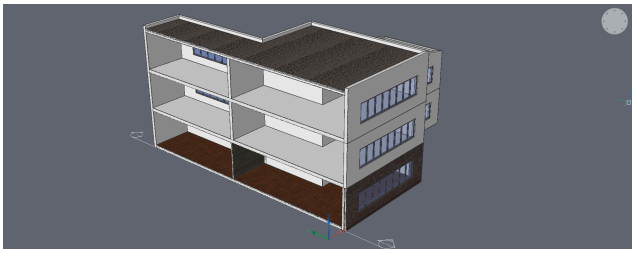
20.4.2.5 Procedure: using autocomplete in one file

1 Open up a file to start from. Here we use a basic rectangular building that looks like this:



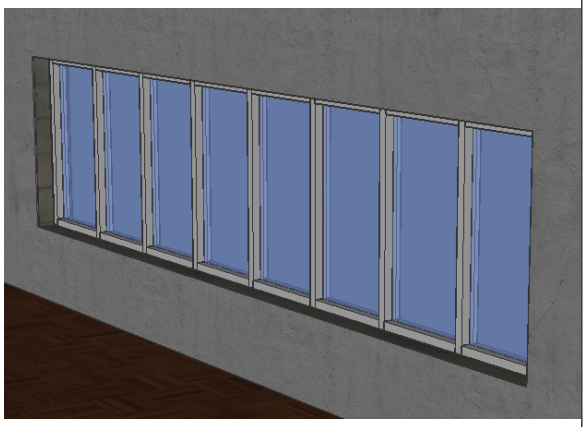
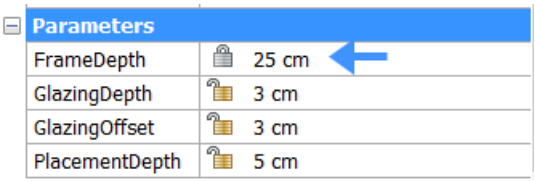
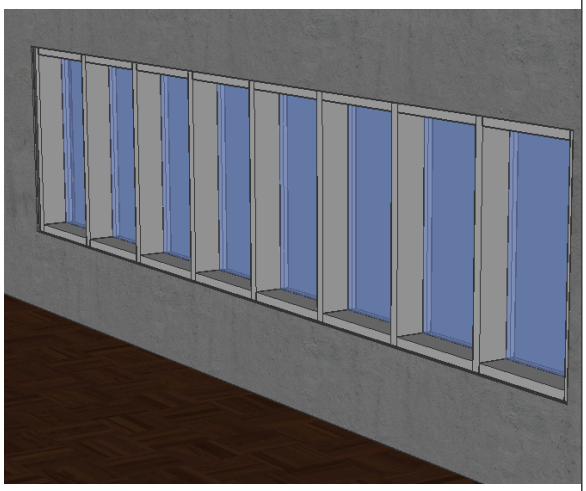
Nota: No BIM data or compositions should be added yet.

2 Add the compositions that you want. Don't do this for all your elements, but only for one element of each type. In our example we give a composition to 1 outer wall, 1 inner wall, 1 floor slab and the roof:



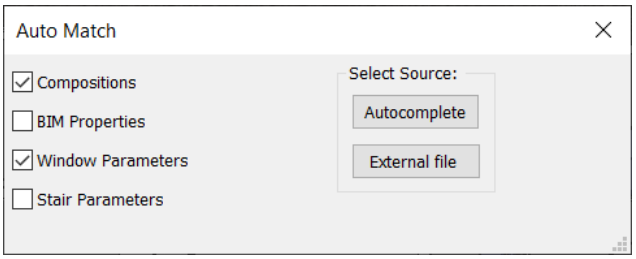
Nota: It could be that you have different types of outer walls, for example the wall facing north that needs more insulation. In this case, you give one of these northern walls a different composition.

3 In this example, we also want to adjust the parameters of our windows. Select only 1 of the windows and change the parameter you want in the properties panel.

	The window before changes.
	Changing the **FrameDepth** in the property panel of the window.
	The window after the changes.

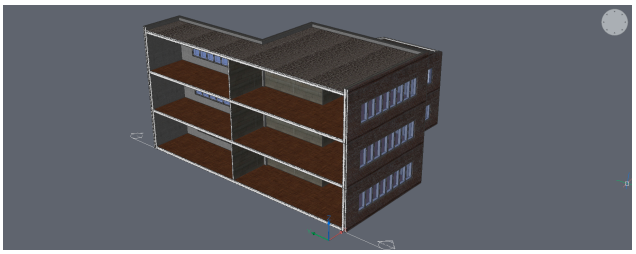
4 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

The following box appears:



We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

5 Select **Autocomplete**. The compositions and window parameters within your file are automatically copied. The drawing looks like this:

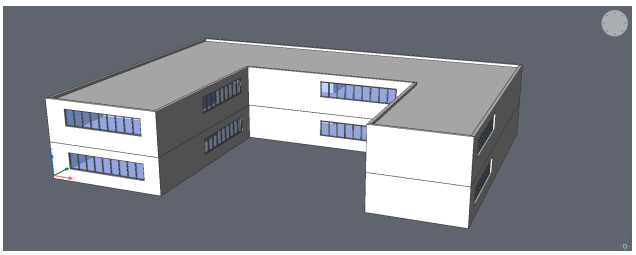


By using BIMAUTOMATCH, we very quickly have all our compositions and window parameters as we want them, without having to worry about forgetting anything.

Nota: You can see that different types of outer walls are no problem. All the walls facing north have been given the correct adjusted composition.

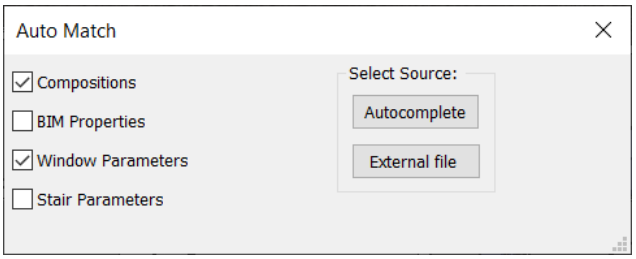
20.4.2.6 Procedure: using autocomplete with an external file

1 Open up a project that is similar in style to another project that you already finished. In this case, we take another rectangular building, similar to the one we looked at before:



2 Select **Automatch** from the **Classify** tab in the Ribbon **or** select **Model** > **Automatch** in the Quad **or** type BIMAUTOMATCH and press Enter.

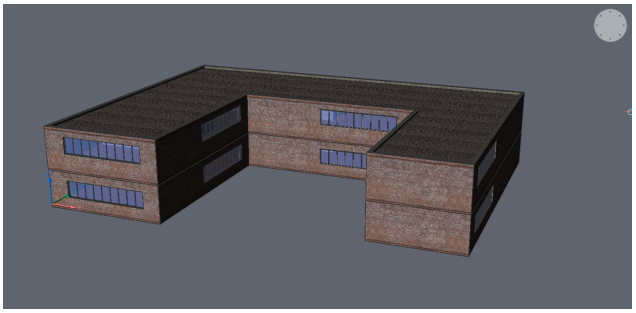
The following window appears:

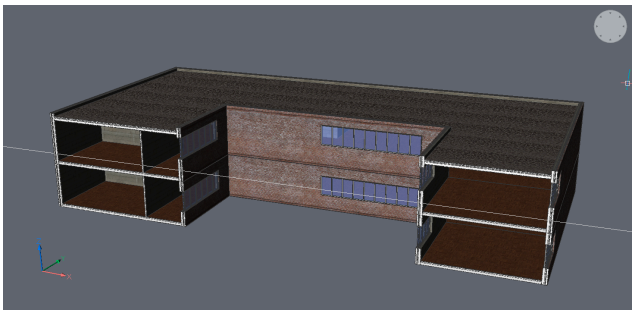


We want the compositions and the window parameters that we chose to be completed in our entire drawing, so we check the boxes from **Compositions** and **Window Parameters**.

3 Select **External file**.

4 Select the file that we made in the previous procedure. The compositions and window parameters from the external file will be copied to your new file. The drawing looks like this:





Nota: It can be handy to make a library of building styles with their typical compositions, in order to use BIMAUTOMATCH with external files in a very efficient way.

20.4.3 Attaching compositions

20.4.3.1 Commands

BIMATTACHCOMPOSITION, BLCOMPOSITIONS

20.4.3.2 About BIM attach composition

The BIMATTACHCOMPOSITION command allows you to easily apply a composition to a selection of solids. The assigned composition can be flipped or redefined to a different reference face. Optionally, you can align solids based on a reference face or the center of a composition.

See BIMATTACHCOMPOSITION for more information about attaching a composition.
See BIM Compositions for more information about managing compositions.

20.4.3.3 Attaching a composition

To attach a composition, you can:

- Use the BIMATTACHCOMPOSITION command.
- Drag a composition from the **BIM Compositions** panel on the solid(s).

Using the BIMATTACHCOMPOSITION command

1 Launch the BIMATTACHCOMPOSITION command.

1 **You are prompted:** Enter the composition name or [Dialog/Entity] <Dialog>.

2 Do one of the following:

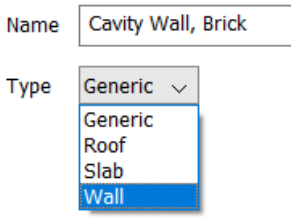
- Type a composition name in the Command line and press Enter.

  Nota: Composition names are case sensitive.

- Press Enter to display the **Compositions** dialog and click on a Composition in the **In Project** or **In Library** list to open the menu of the composition.

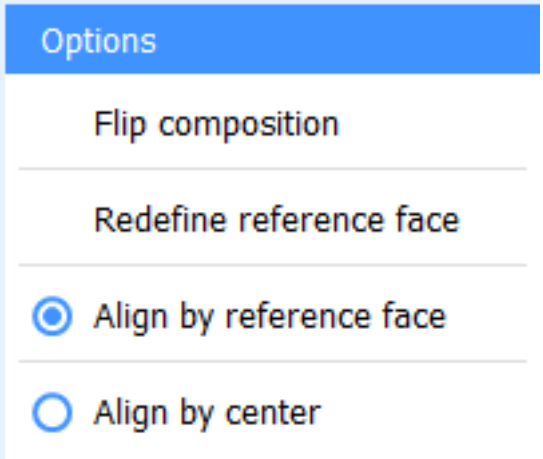
  If you want to select the composition without opening the menu, double-click it.

3 Choose a Composition **Type** from the drop-down menu and press **Select** to accept.



4 Select entities to attach the composition to and press Enter.

5 Choose the reference face. If you hover over the widget, a submenu displays showing different options. For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.



6 Press Enter to accept the reference face.

7 BricsCAD responds: The composition has been assigned to <xxx> element(s).

Nota: Using Bimify building elements are automatically classified. This makes it easy to select the right entities.

Dragging a composition on a solid

1 Select the composition in the BIM Compositions panel (). See Dockable panels.

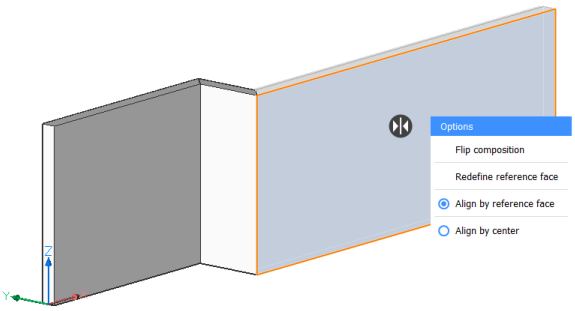
2 Hold down the left mouse button and drag the composition on the solid.

3 The solid face under the cursor highlights, indicating that this face is the reference face to attach the composition.

4 Release the left mouse button to attach the composition.

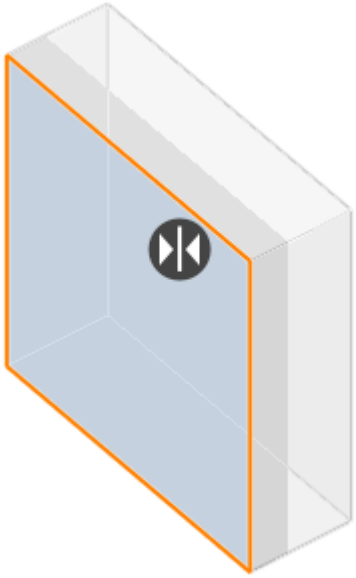
5 A widget appears that allows you to choose the reference face.

6 The reference face is fixed, while the opposite face moves if the thickness of the composition does not fit the current thickness of the solid.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

7 A darker volume indicates the volume and position of the solid after attaching the composition.



8 Press Enter to accept the current reference face and attach the composition.

Nota: When a single ply composition with a variable thickness is selected, it is not needed to select a reference face. The composition is attached to the selected solid.

Dragging a composition on a selection of solids

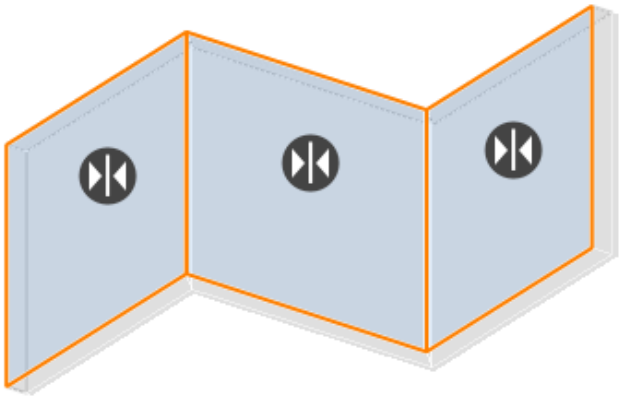
1 Select the solids you want to apply the composition to.

2 Select the composition in the **BIM Compositions** panel.

3 Hold down the left mouse button and drag the composition on any of the selected solids.

4 Release the left mouse button to attach the composition.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

Nota:

- BIM compositions are copied to the Project database upon attachment.
- Attaching a **Wall** composition to a solid, automatically classifies the solid as a wall.
- Attaching a **Slab** or **Roof** composition to a solid, automatically classifies the solid as a slab.

20.4.3.4 Copy compositions from other solids

1 Launch the BIMATTACHCOMPOSITION command.

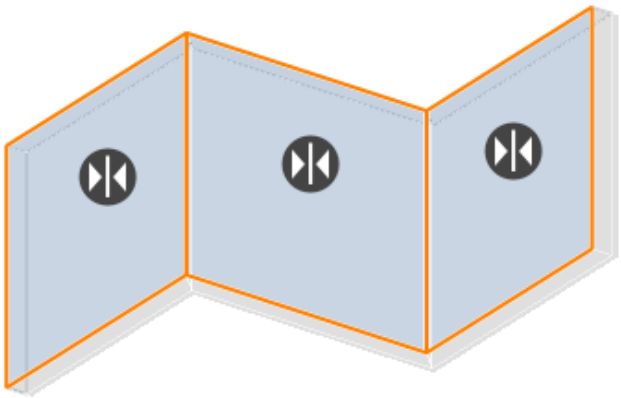
2 Select the **Entity** option.

Prompts you: **Select other entity to get composition:**

3 Select the solid to copy the composition from.

4 Select the solids(s) to attach the copied composition to.

5 A widget appears which allows you to choose the reference face for every segment separately.



For more information about how to redefine the reference face, see BIMATTACHCOMPOSITION.

6 Press Enter to accept the current reference face and attach the composition.

20.4.3.5 Removing compositions from solids

The **Unclassify** option of the BIMCLASSIFY command removes a composition from a solid.

1 Select the solid(s).

2 Choose **Unclassify** from the **BIM** command group in the Quad.

20.4.4 BIM classify

20.4.4.1 Commands

BIMCLASSIFY

20.4.4.2 About

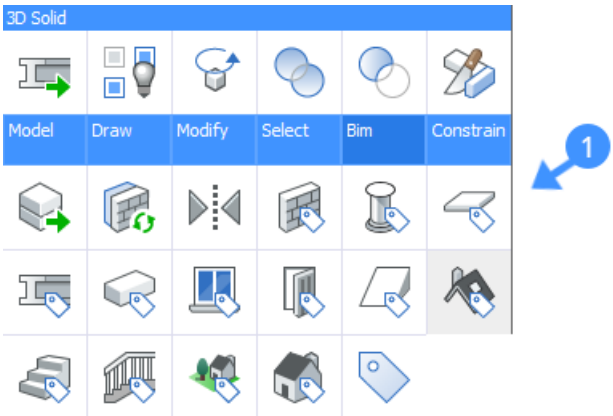
If you have a conceptual model without any classification, you can turn these elements into classified BIM elements using BIMCLASSIFY.

For more information about this command, visit the Command Reference article BIMCLASSIFY.

20.4.4.3 Classifying entities

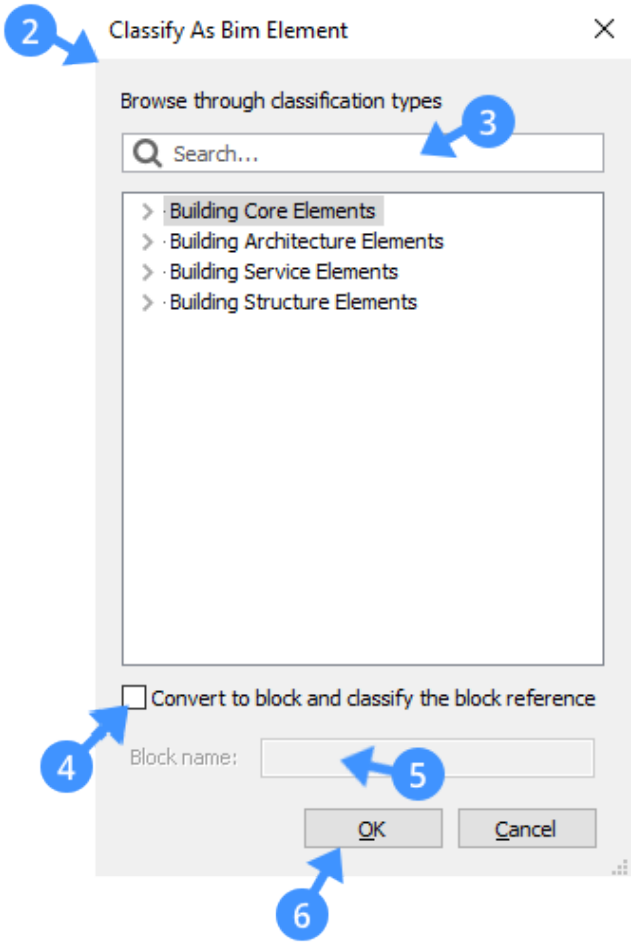
1 Select an entity in the drawing to classify.

2 Choose a classification type (1) in the **BIM** command group in the Quad.



The BIM data is assigned to the selected entity.

3 **(Optional)** Choose the **classify manually** option from the Quad to access the BIM Element classification types. This displays the **Classify as BIM Element** dialog box (2):



Nota: The classification types are grouped into the following categories:

- **Building Core Elements**
- **Building Architecture Elements**
- **Building Service Elements**
- **Building Structure Elements**
- **Spatial Structure Elements**

Do one of the following:

- Enter a specific name of the BIM Element type in the **Search** field (3) to filter the available element types.
- Select a Building Element type from the list.
- (**Optional**) Check the **Convert to block** option (4) and enter a name in the **Block name** field (5).

This option will convert the selected entity to a block and classify the block reference to the defined Building Element type.

- Click the **OK** button (6) to accept it.

4 **(Optional)** Use the Auto ( ) option from the Quad to classify the selected entities automatically.

This automatic classification is done based on the geometrical features of the object. Always check whether it was correctly classified.

5 To remove the assigned BIM data from the entity, select the entity in the drawing and choose the Unclassify ( ) option from the Quad.

The BIM data is removed from the selected entity.

20.4.4.4 Properties of the classified entities

When you classify an entity, a number of properties with regard to the building element type is displayed in the **Properties** panel.

The following illustrations show the properties that are available for windows and doors:

Door Window Glazing Type	
Glass layers	0
Glass thickness 1	0 mm
Glass thickness 2	0 mm
Glass thickness 3	0 mm
Fill gas	
Glass color	
Is tempered	Off
Is laminated	Off
Is coated	Off
Is wired	Off
Translucency	0
Reflectivity	0
Beam radiation transmittance	0
Solar heat gain transmittance	0
Thermal transmittance summer	0 W/m²·K
Thermal transmittance winter	0 W/m²·K

BIM	
Type	Window
Building Element Type	Window
Name	
Description	
Building	
Story	
Space	
Composition	
GUID	0shXs2TdfFB9RulemExNT4
Space bounding	Off
Construction type	Not defined
Operation type	Not defined
Window inset	
Overall height	0 mm
Overall width	0 mm
Sill height	0 mm
Head height	0 mm
Assembly code	
Assembly subtype	
Number	
Entity property sets	

Door Window Shading Type	
External shading coefficient	0
Internal shading coefficient	0
Inset shading coefficient	0

Window Common	
Reference	
Fire rating	
Acoustic rating	
Security rating	
Is external	Off
Infiltration	0 m^3/s
Thermal transmittance	0 $W/m^2 \cdot K$
Glazing area fraction	0
Smoke stop	Off

20.4.5 BIM compositions

20.4.5.1 Commands

BLCOMPOSITIONS, BIMCLASSIFY, BLMATERIALS, BIMATTACHCOMPOSITION

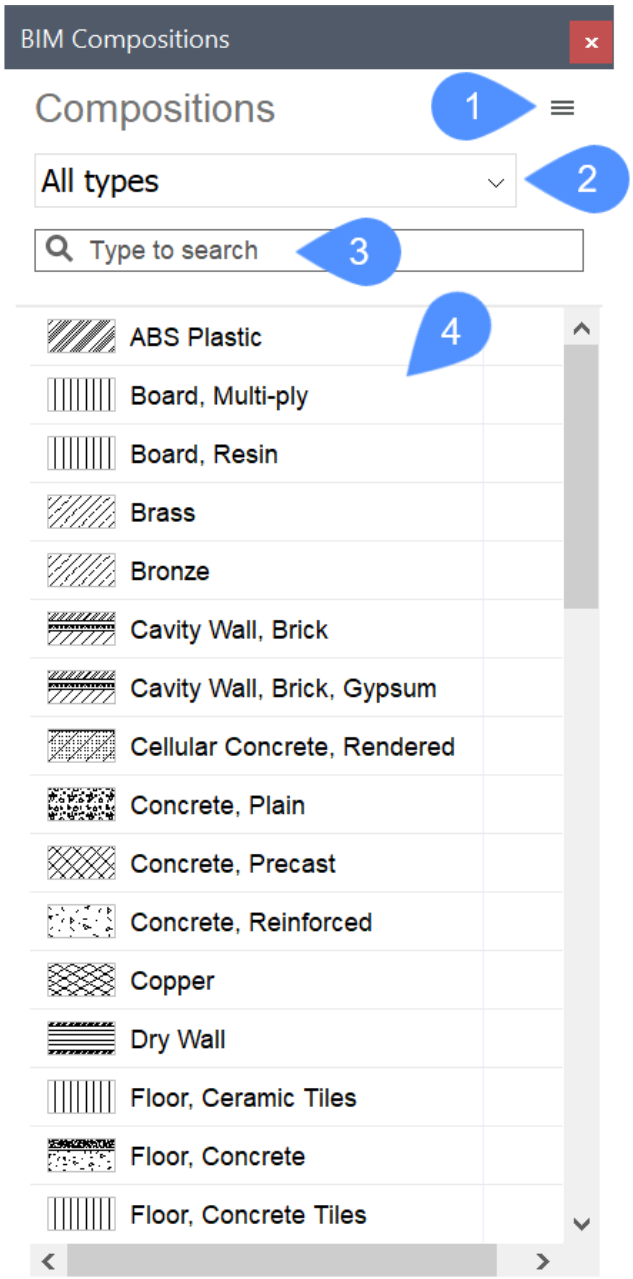
20.4.5.2 About

BIM compositions:

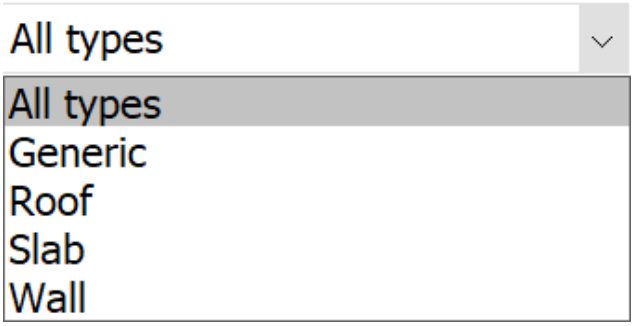
- Are stored in the library database.
- Are either multiply or single-ply.
- Ply materials are managed in the **Physical Materials** editor ( ).
- Can be assigned to any solid in a BIM model.
- Are listed in the **BIM Compositions** panel ( ).
- Are managed in the **Compositions** editor ( ).

For more information about Attaching Compositions, see the procedure article Attaching Compositions.

20.4.5.3 Using the BIM compositions panel



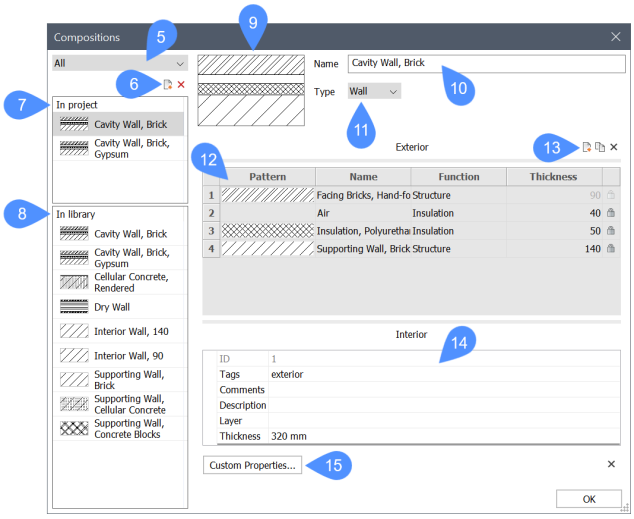
(1) Menu	Open material dialog... Open compositions dialog... Open project dialog... Show only compositions in project...

(2) Composition types list	
(3) Search field	Filters the list of the compositions. Only compositions that contain the current search term are displayed.
(4) Available compositions	All compositions of the selected type in both the **In library** and the **In project** database display, unless the **Show only compositions in project...** menu option is selected.

20.4.5.4 Using the BIM compositions editor

The BIM **Compositions** editor is a dialog box that contains options to create, modify and delete compositions. Using the **Compositions** editor, you can also sort the compositions by category and view the properties of the composition.

The BLCOMPOSITIONS command opens the BIM **Compositions** Editor:



(5) Composition category selection	Filters by category: **Wall**, **Slab**, **Roof**, or **Generic**.
(6) New composition	Creates a new composition.

(7) In project	Displays the compositions of the selected category (**wall**, **slab**...) in the Project database.
(8) In library	Displays the compositions of the selected category (**wall**, **slab**...) in the Library database.
(9) Preview pane	Displays the currently selected composition.
(10) Name field	Displays the selected composition name. You can change the default name.
(11) Category drop-down box	Displays a category for the new composition.
(12) Structure grid	Displays the material and thickness of the composition plies. The ply on top (exterior) is applied to the reference of the solid.
(13) Add ply	Opens the physical materials editor which allows you to choose a material from the library or project database. Here you can also create a new material. Inserts a copy of the selected ply.
(14) Properties grid	Shows the properties of the selected composition.
(15) Custom properties...	Opens the **custom properties** dialog.

20.4.5.5 Creating a new composition

You can create a new composition in either the **Project** or the **Library** database.

1 To open the BIM **Compositions** editor, do one of the following:
   - Select the **Open the compositions dialog...** in the menu on the **Compositions** panel.
   - Type BLCOMPOSITIONS in the Command line.

2 Click the **New** icon ( ) on the left of the compositions toolbar.

   A new (empty) composition is added to the selected database.

3 To add and edit plies, go to **Edit a composition**.

20.4.5.6 Editing a composition

Step 1: Changing name and type

1 Select the new composition.

2 Type a new name in the **Name** field for the composition.

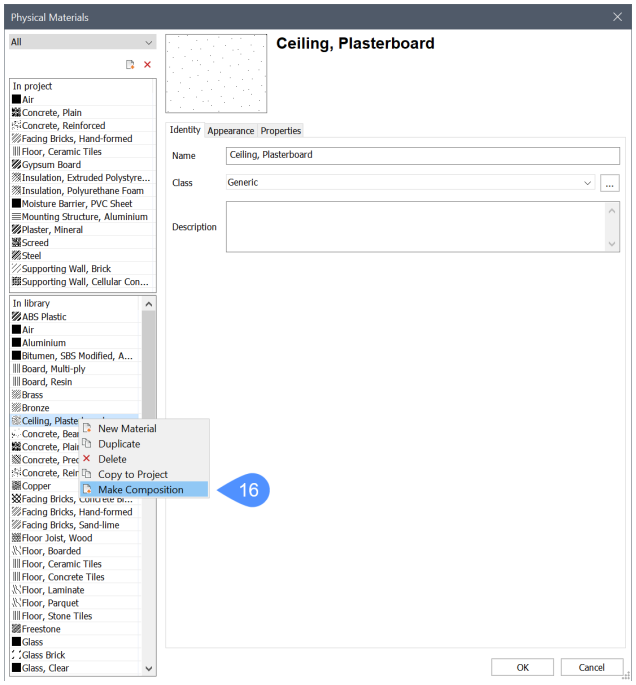
3 Use the Type drop-down box to specify the composition type: **Wall**, **Slab**, **Roof** or **Generic**.

Composition categories are used to filter the list of the compositions and the **Compositions** dialog.

Step 2: Changing the plies

- **Adding, editing and deleting plies**
  - a Click the Add Ply button ( ) on the right of the compositions editor.

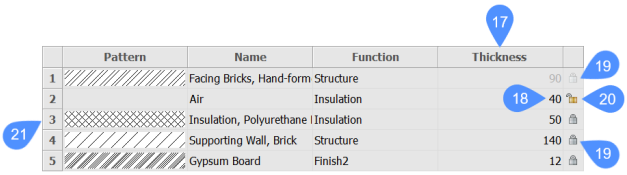
    The **Physical Materials** dialog displays:



  - b Select a material either in the Project **or** the Library database and double-click **or** press the **OK** button to assign the material to ply in the composition.
  - c Double-click on the **Pattern** or **Name** of the ply to edit the material. See the **Physical Materials** article for more information.
  - d To choose another material, pick a material in the Library **or** Project list and press **OK**.
  - e To delete a ply, select the ply and click on the **Delete** tool button.
  - f **(Optional)** Right-click the material and choose **Make Composition** (16) in the context menu to create a single-ply composition.

- **Modifying the thickness and changing the ply order**
  - a Click the **Thickness** field (17) to modify the thickness of a ply in the Structure grid.

    You can only modify the ply thickness if the **Allow Custom Thickness** property of the material is set to **Yes**.



  - b Enter a value in the field (18) and press Enter to accept it. The overall thickness of the composition is updated accordingly.
  - c The plies of a composition are ordered from Exterior (top) to Interior (bottom). Drag the ply

number (21) on the desired position to change the ply order.

Nota: When a composition has been modified, the solids that have this composition attached are updated automatically. Use the BIMUPDATETHICKNESS command apply the changes in the model.

- **Locking/unlocking the composition thickness**

  The overall thickness of a composition is equal to the sum of the thicknesses of the composing plies.

  The thickness of a ply can be either locked or unlocked (see image above).

  - Locked thickness (19): the thickness of the ply is fixed.
  - Unlocked thickness (20): the thickness of the ply is variable.

  Nota: Only one ply in a composition can have an unlocked thickness. As a result, the total thickness of a composition can be:

  - **Fixed:** all plies have locked thicknesses.
  - **Minimal:** the composition contains at least two plies and one ply has an unlocked thickness. The minimum thickness is equal to the sum of the thicknesses of the locked plies.
  - **Free:** a single ply composition has an unlocked thickness.

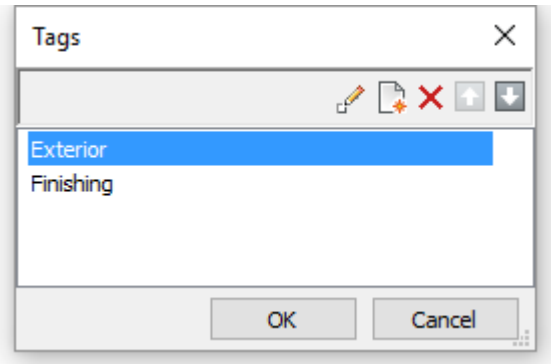
  The thickness of a ply can only be unlocked if the **Allow Custom Thickness** property of the material is set to **Yes**.

- **Adding Tags**

  a Select the **Tags** field.

  b Type the tags, separated by commas. Tags are case sensitive.

  c Click the **Browse** button ( ) next to the tags field in the **Compositions** dialog box to open the **Tags** dialog.



  d Click the **New Item** button ( ) to add a new tag at the bottom of the list.

  e Select a tag in the list and click a tool button:

  -
  -
  -
  -

  f Click the **OK** button to save the changes.

Step 3: Creating new materials

You can create new materials from scratch or copy them from an existing material. See **Physical Materials** for more details.

20.4.5.7 Displaying compositions

Whether the various plies of a composition show in the 3D model is controlled through the LEVELOFDETAIL user preference. The RENDERCOMPOSITIONMATERIAL user preference controls whether the ply material is rendered or not.

Click the icons in the **Home/View** panel in the ribbon or the **BIM** toolbar to toggle the status of the composition display:

- Level of Detail:
- Render Composition Material:

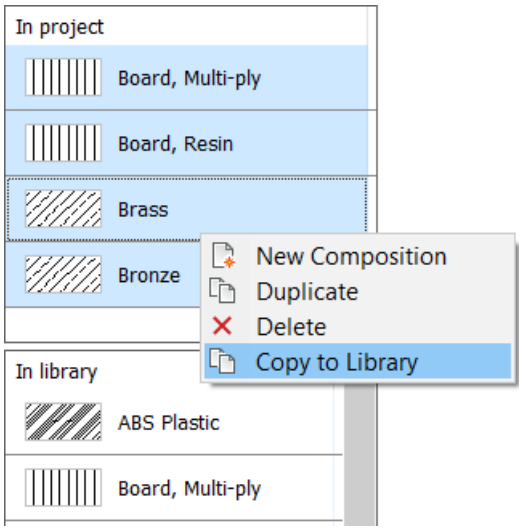
When LEVELOFDETAIL=0, the major faces of a solid are rendered using the material of the outside plies. The minor faces (for example top, bottom, start and end face of a wall) are rendered using the material of the exterior ply.

20.4.5.8 Copying compositions

You can use the following procedure in both the **BIM Compositions** panel and the **Compositions** editor:

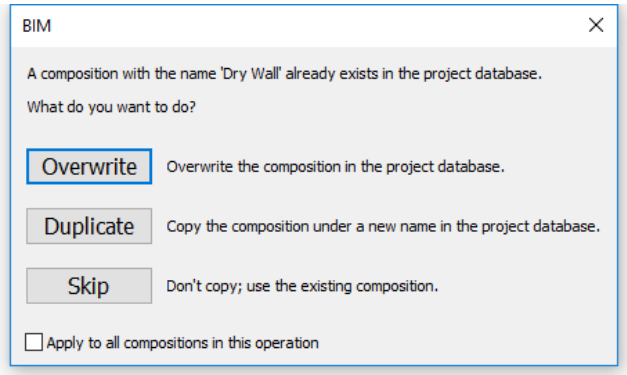
1 Select the composition(s) in the source database. Press and hold the Ctrl key to select multiple compositions.

2 Do one of the following:
   - Drag the selection to the other database.
   - Right-click and choose **Copy to Library** or **Copy to Project** in the context menu.



If the value of the COPYOVERWRITE user preference is 0 (zero), a dialog box displays if the composition already exists in the target library. You can choose to overwrite, duplicate or skip existing compositions.

(Optional) Check the **Apply to all compositions in this operation** option.



20.4.5.9 Deleting compositions

1 Select the compositions in the **In Project** or **In Library** list on the **BIM Compositions** panel or the **Compositions** editor.

2 Do one of the following:
   - Right-click and choose **Delete** in the context menu.
   - Click the **Delete** button (**Compositions** editor only).

Nota: Compositions that are used in the project cannot be deleted from the project database.

20.4.6 BIM data structure

20.4.6.1 Commands

BIMSPATIALLOCATIONS, BIMATTACHCOMPOSITION, BLCOMPOSITIONS, BIMPROFILES, BLMATERIALS, BIMPROPERTIES, BIMPROJECTINFO

20.4.6.2 Project database and library database

A BIM Project consists of dwg entities, classified as **Building Elements**, to which Compositions can be attached. All Compositions being used in a project, are stored in the Project database. The Project database can be stored inside a dwg file, or in a separate file with extension **.bsyslib:**

- When the Project database is embedded in a dwg file, that dwg file contains the complete BIM Project.
- When the Project database is stored as an external file, the BIM Project can consist of multiple dwg files, all using the same Project database.

To reuse compositions and building materials across multiple BIM Projects, a Library database can be specified, in addition to the Project database. The Library database is a **.bsyslib**file, usually stored in a location which is common to several projects. When Compositions and Building Materials from the Library database are used in a BIM Project, they are automatically imported in the Project database. To maintain and extend BIM databases, users can drag Building Materials and Compositions from one database to the other, provided that they have write access to the destination database.

Note	Sample databases are installed in the **Support** subfolder in the **Roamable** root folder. When a BricsCAD update is installed, the **BricsCAD User File Manager** will prompt you whether to overwrite the sample databases or not. Therefore, it is recommended to rename your library databases and/or save them at a different location. The location of the library databases is defined in the **Library DB** tab of the **BIM Project Info** dialog.

20.4.6.3 Materials and compositions

A BIM database consists of Building Materials and Compositions. A Building Material contains information about one specific material: its appearance, cost, manufacturer, etc. A Composition contains information about the structure of a building element, by defining Composition layers. Each Composition layer refers to a Building Material and has a certain thickness.

Flexibility

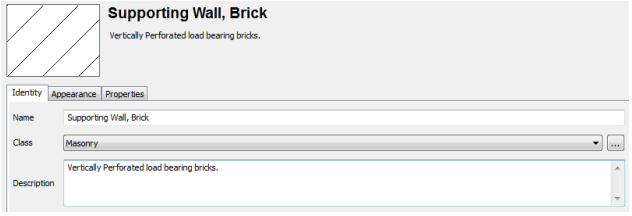
The same flexibility you have in modeling is available in attaching building information. Although Compositions are grouped into **Wall**, **Slab**, **Roof** and **Generic** Compositions, any type of Composition can be attached to any dwg entity. When attached to a 3D Solid, with specific geometrical properties, the thickness of the Composition is used to define the thickness of walls and slabs, and the cross section will show the wall or slab structure by applying the hatch patterns from the various layers on the section geometry. When a 3D Solid is assigned a Composition with a fixed thickness or a minimal thickness, and this thickness can not be applied to the geometry, the section appears in red to indicate the incompatibility between the selected Composition and the geometry of the solid.

BIM Compositions panel

All BIM database features are accessible from the dockable **BIM Libraries** panel.

Materials

A Material definition describes the appearance and properties of a Building Material.



Composition layers

A Composition describes the structure of a building element by an ordered set of composition layers. Each composition layer has the following fields:

- Pattern and Material **Name**: The Building Material for this layer.
- Function: One of (**None**, **Structure**, **Substrate**, **Insulation**, **Finish1**, **Finish2**, **Membrane**). Currently, this field serves for information purposes only.
- Thickness: Defines the thickness of the layer.

20.4.7 BIM ply editing

20.4.7.1 Commands

LEVELOFDETAIL, RENDERCOMPOSITIONMATERIAL, DMPUSHPULL

20.4.7.2 About

The **BIM Ply Editing** function allows you to manipulate all the composition plies in your model. In BricsCAD, each drawing has settings stored inside to control the composition plies, called LOD (LEVELOFDETAIL), and the render materials, called RENDERCOMPOSITIONMATERIAL. These two settings can be turned on/off separately.

20.4.7.3 Composition plies and render materials

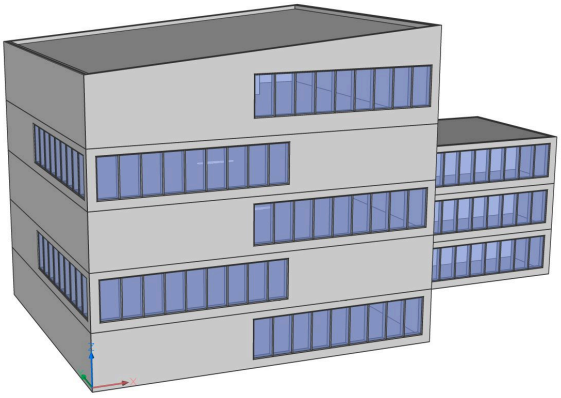
Composition plies display only when LEVELOFDETAIL = 2 (high).

Building elements do not need to be selected, they will be automatically adjusted. When LEVELOFDETAIL = 0 (low) the material definition of the solid is used.

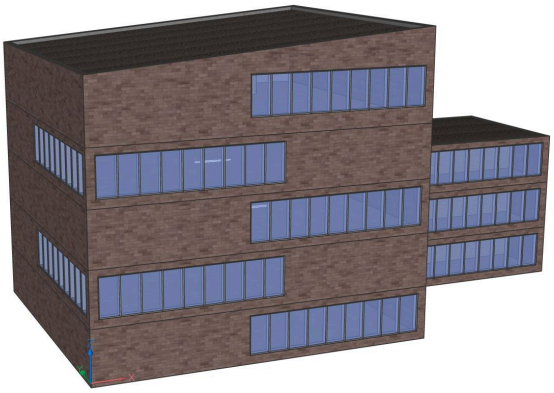
To toggle the value of the LEVELOFDETAIL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:

For more information about this setting visit the article LEVELOFDETAIL.
Render materials display only when RENDERCOMPOSITIONMATERIAL = 1 (on). The render material of each ply is defined in the 3D Render Material setting of the material definition.
To toggle the value of the RENDERCOMPOSITIONMATERIAL user preference, click the icon in the ribbon under the **Home** tab or in the quad under the **BIM** tab:

For more information about this command, visit the article RENDERCOMPOSITIONMATERIAL.



RENDERCOMPOSITIONMATERIAL = 0 (off)



RENDERCOMPOSITIONMATERIAL = 1 (on)

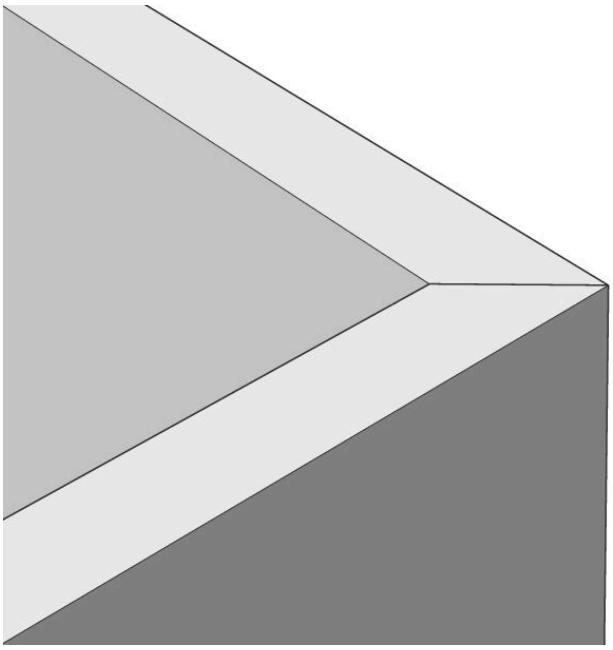
Nota: Only when Visual Styles have their **Face Settings / Material Display settings** set to **Materials and Textures** (e.g. **BIM**, **Modeling**, and **Realistic**), a visual representation of materials is displayed. Composition plies are displayed in the model as well as render materials.

Nota: If the render materials of the selected composition are not downloaded or the composition is incorrect, the entity will be displayed in red.

There are different combinations possible with LOD and RENDERCOMPOSITIONMATERIAL turned on or off.

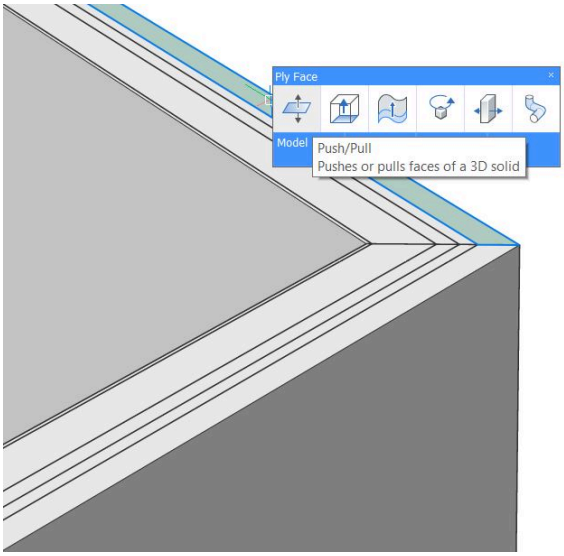
- **LOD =0, RENDERCOMPOSITIONMATERIAL = 0**

  The material definition of the solid is used and no materials are displayed.



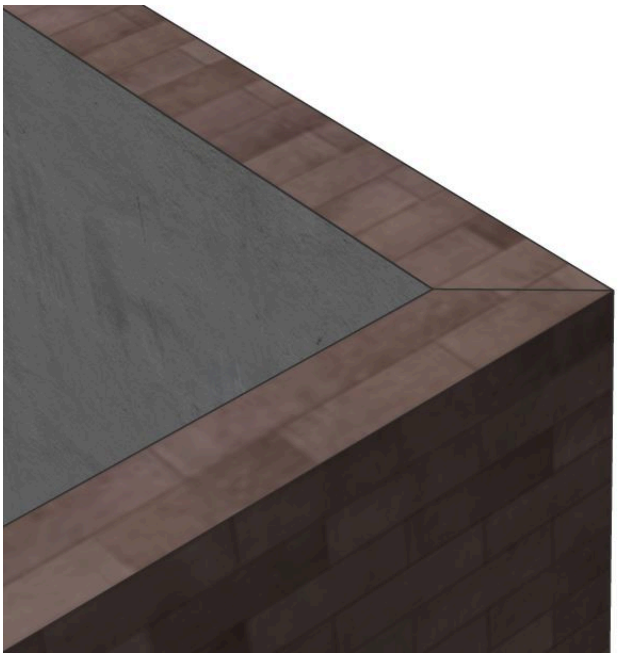
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 0**

  The solid is subdivided into plies, no materials are displayed. When **face detection** is turned on, you can select the faces of different plies to **push/pull** for example.



- **LOD =0, RENDERCOMPOSITIONMATERIAL = 1**

  The material definition of a solid is used. The materials of the exterior and interior face are displayed and the material of the exterior layer will be wrapped around the object.



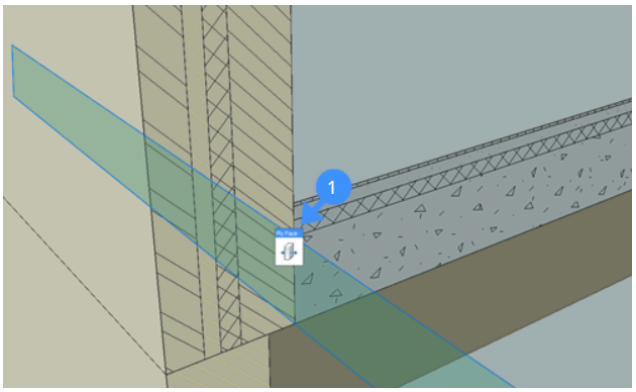
- **LOD =2, RENDERCOMPOSITIONMATERIAL = 1**

  The solid is subdivided into plies. Each ply will be displayed with its own material. When face detection is turned on, you can select the faces of different plies to push/pull for example.

20.4.7.4 Manipulating the composition plies

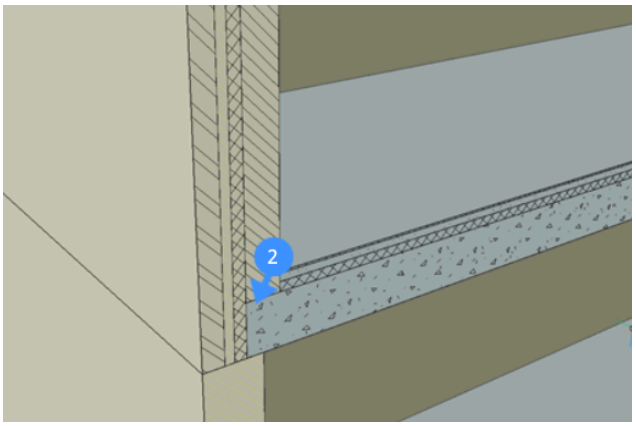
1 Make sure the value of **LEVELOFDETAIL = 2 (High)**.

2 Place your cursor near the composition ply you want to edit and press the Tab key until the **Ply face** (1) highlights.



Nota: The highlighted ply face is displayed in the detected boundary color (default = 95). Face detection must be enabled to do this.

3 Select **Push/Pull** from the Quad and drag the ply to a new position (2).



Nota: Make sure DMPUSHPULLSUBTRACT = 1. This ensures that plies being push/pulled are subtracted from any of the other 3D solids they intersect.

Nota: Make sure ESNAP is turned on to snap to the edge of a ply.

20.4.7.5 Procedure: creating a rooftop detail

It is also possible to create more complex details with extra solids added apart from the existing compositions. As an example, a rooftop detail will be created, starting from a rooftop edge.

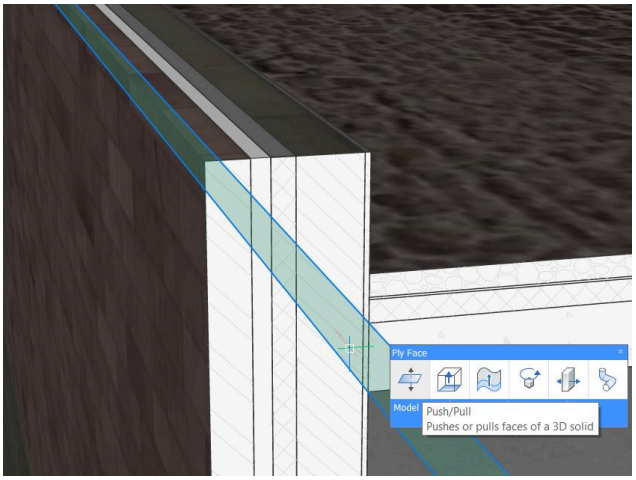
1 tart with this roof-wall connection.

2 Click the **Face detection** icon in the ribbon to enable face detection.

3 Turn on LOD to be able to select ply faces. Click the icon in the ribbon **or** type LEVELOFDETAIL in the Command line, type **2** and press Enter.

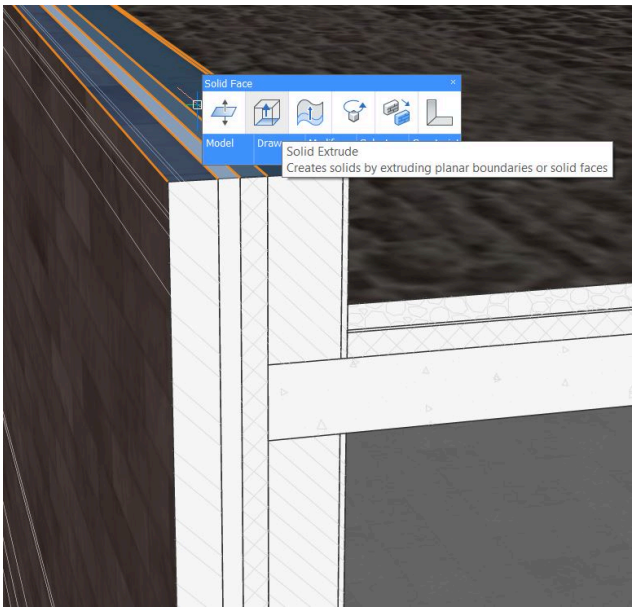
4 Hover over the edge of the concrete ply of the floor slab and press Tab to select the face of the ply.

5 Choose **Push/Pull** in the **model** tab of the Quad.

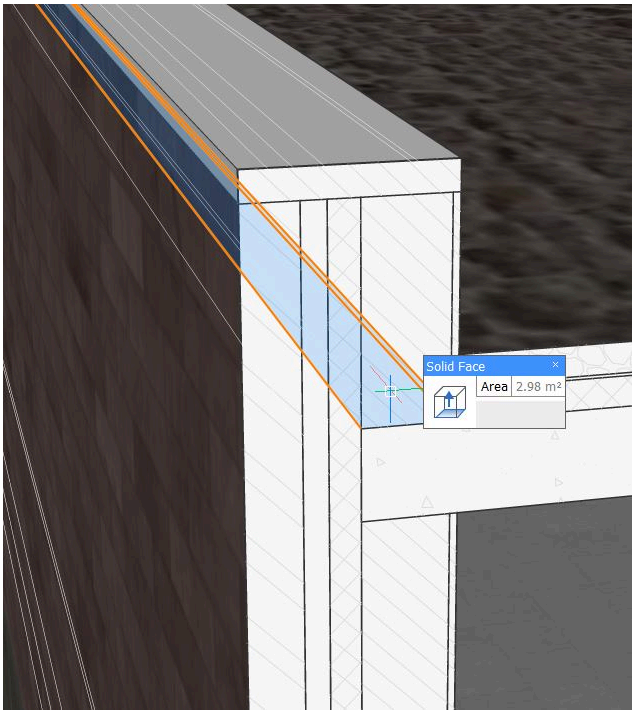


6 Push/Pull the face to the next ply of the wall.

7 To create a wall cap, select the top surface of the wall by pressing Tab and extrude the surface up for 50mm. The created cap isn't classified yet.



8 Delete the part where the insulating building block will come. Hover over the surface of the supporting wall and the gypsum board and press tab to select the surface. Then Extrude till the ply of the wall cap.



9 **Extrude** the same surface again, now a new solid will be created.

20.4.8 BIM project info

20.4.8.1 Commands

BIMPROJECTINFO

20.4.8.2 About

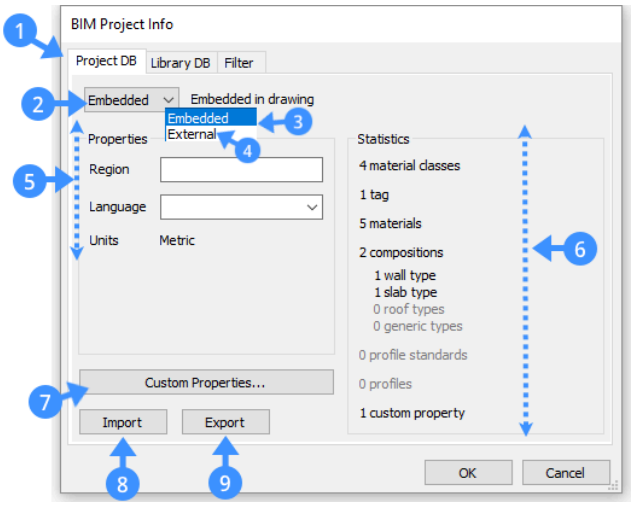
The **BIM Project Info** allows you to control the location, properties, and content of the Project and Library databases. When you launch the BIMPROJECTINFO command in BricsCAD, the BIM project info displays in a dialog box that includes certain components and tabs. Project and library databases of BIM models can be changed using this dialog box.

For more information about this command, visit the Command Reference article BIMPROJECTINFO.

20.4.8.3 Procedure: setting the project database

1 Open the **BIM Project Info** dialog box by typing BIMPROJECTINFO in the Command line.

The following illustration shows the dialog box when the **Project DB** tab is current.



Project database (1)	Shows the BIM project info.
Project database drop-down list (2)	Choose between **Embedded** or **External**.
Embedded (3)	Saves the project library in the drawing file.
External (4)	Saves the project library in a BIM database file (.bsyslib).
Properties (5)	Shows the properties of the BIM project e.g. region, language, and units.
Statistics (6)	Shows the location, properties, and the content of the current project library of the BIM model.
Custom properties (7)	Adds custom properties to materials and compositions.
Import (8)	Imports the selected .xml and .csv library files.
Export (9)	Saves .xml library files in the defined location.

2 To change the project database from **Embedded** to **External** or vice versa, select the desired option in the Project Database drop-down list.

Nota: The external database keeps the database independent from the current drawing. This is especially useful if you want to share the library between multiple drawings.

- When switching from **Embedded** to **External** a **File** dialog displays.

  Do one of the following:

- Type a name in the **File Name** field to export the content of the project library to a new .bsyslib file, then click the **Open** button.

- Select an existing .bsyslib file, then click the **Open** button.

  The current content of the project library is merged with the selected .bsyslib file.

  - When switching from **External** to **Embedded** the content of the external library is copied to the embedded project library.

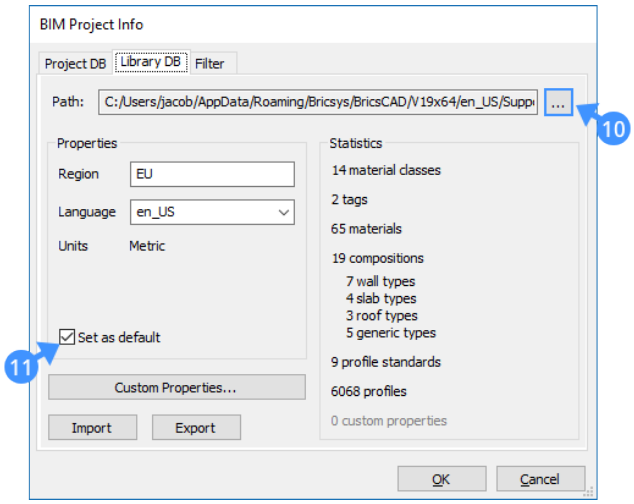
3 To specify the Project Database properties, such as Region and Language, fill out the **Region** field and choose one of the language options in the drop-down list.

4 Optionally, use the custom properties button to add material and composition custom properties in the Project Database.

5 Press the **OK** button to accept it.

20.4.8.4 Procedure: setting the library database

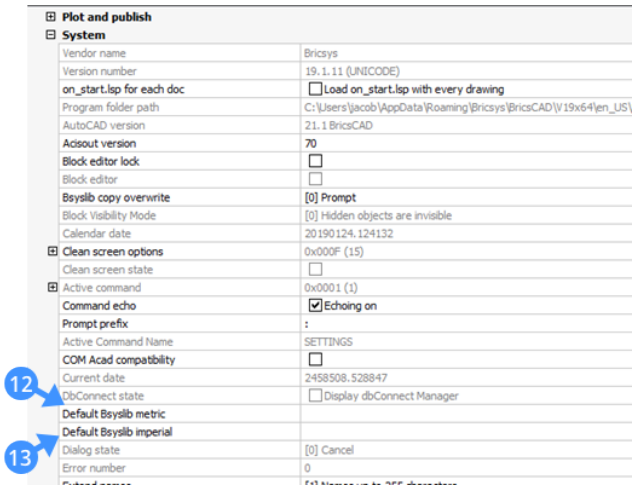
1 Click the **Library DB** tab page in the **BIM Project Info** dialog box. The following illustration shows the Library Database section in the dialog box.



2 To select a BIM database file (.bsyslib), click the Browse (10) button of the **Path** field, then choose a database file in the file dialog.

3 Optionally, check the **Set as default** (11) option to set the selected database as default library database for next projects.

Nota: You can edit the default library settings (12, 13) in the **Settings** dialog under **Program options** > **System**.



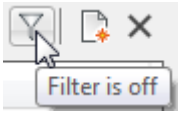
20.4.8.5 The project filter

To each Building Material or Composition, any number of tags can be attached. Any string can be entered as a tag.

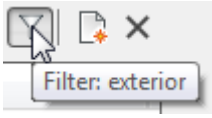
Each of the tags can be used as a filter on the database. There is a button at the top of each list ( ), to switch the filter on or off. If the filter is on, only Compositions and Building Materials that match at least one of the filter tags will be listed.

The **Filter** button ( ) has three states:

- If the project filter is empty, the button is not active (**BIM Compositions** panel) or is not available (**Materials** and **Compositions** editors).
- The project filter is defined, but not active.



- The project filter is defined and active.

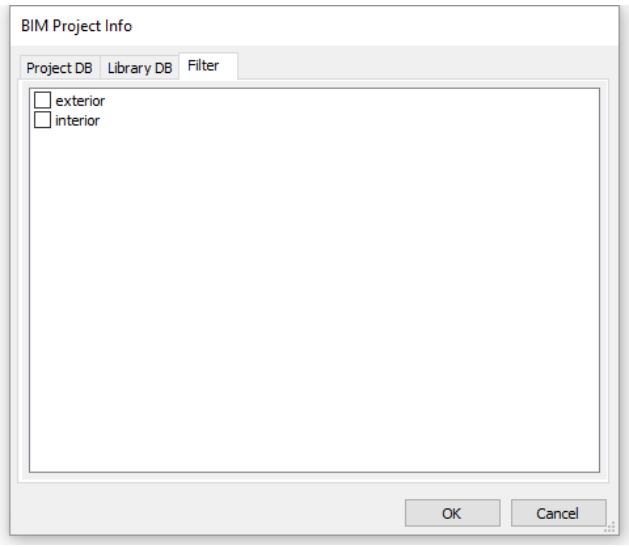


A tooltip shows the current project filter.

20.4.8.6 To set the project filter

1 Select the **Filter** tab page on the **BIM Project Info** dialog box.

2 Check the tags you want to use in the filter.



20.4.9 Using Python scripts (beta)

20.4.9.1 Commands

BIMPYTHON

20.4.9.2 About BIMPYTHON

The new BIMPYTHON command enables you to query and manage data from a model within BricsCAD BIM, with a suitable .py Python script.

The Python scripts can be as simple as obtaining quantities and associated properties of objects and BIM entities, or as elaborate as implementing a series of calculations based on the model parameters available.

BricsCAD does not ship with a Python Shell, so the scripts would need to be prepared in a text or code editor application.

The Python Programming Language, together with its standard libraries, is embedded within BricsCAD BIM so there is no need for you to install them separately unless you have custom packages and libraries which you wish to use in your scripts.

Before using BIMPYTHON, you need to first turn on the virtual Python environment within BricsCAD.

20.4.9.3 Procedure: activating the Python environment in BricsCAD

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMACTIVATEPYTHON in the Command line and press Enter.

3 **You will be prompted:** New current value for BIMACTIVATEPYTHON [1 for ON/0 for OFf] <1 for ON>:

4 By default, the value should display **<1 for ON>**. That means the Python environment is already activated.

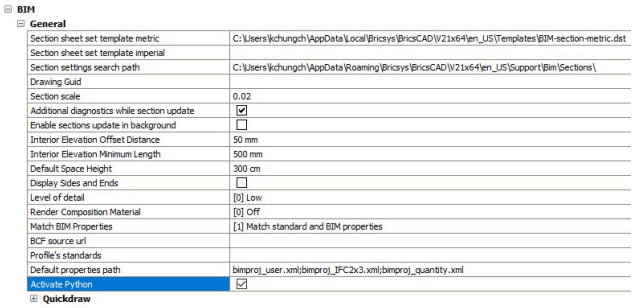
However, If the value shows **<0 for OFF>**, type **1**in the Command line and press Enter to confirm.

Or

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Access the **Settings** dialog box.

3 Tick the box where it says **Activate Python** under the **General** subcategory within **BIM**.



BIM

General

Section sheet set template metric	C:\Users\kchungch\AppData\Local\Bricsys\BricsCAD\V21x64\en_US\Templates\BIM-section-metric.dst
Section sheet set template imperial	
Section settings search path	C:\Users\kchungch\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support\Bim\Sections\
Drawing Guid	
Section scale	0.02
Additional diagnostics while section update	☑
Enable sections update in background	☐
Interior Elevation Offset Distance	50 mm
Interior Elevation Minimum Length	500 mm
Default Space Height	300 cm
Display Sides and Ends	☐
Level of detail	[0] Low
Render Composition Material	[0] Off
Match BIM Properties	[1] Match standard and BIM properties
BCF source url	
Profile's standards	
Default properties path	bimproj_user.xml;bimproj_IFC2x3.xml;bimproj_quantity.xml
Activate Python	☑

Quickdraw

4 Close the dialog box to confirm your updated settings.

20.4.9.4 Procedure: setting up a Python script with the BIMPYTHON module

1 Before gaining access to the API, import or 'call' any desired modules you wish to have in the script.

1 If they are not part of the Python Standard library, make sure they have been installed separately beforehand.

You can "import" a standard module, such as math and "import as" an external one, for example Pyplot, which is a collection of functions in the Matplotlib package:

```
import math
import matplotlib.pyplot as plt
```

2 With your desired libraries "called" in, the next step is to gain access to the API in order to query the elements within the BricsCAD model. This is the range of bim objects and serves as an entry point to the model.

3 Import the briqpy.bim_model:

```
from briqpy import bim_model
```

4 Now, you can query the model with these example statements.

```
# Display info about the lengths of walls in the model
lengths = [wall.prop('Length') for wall in bim_model.filter(Type='Wall')]
print(f'wall lengths. max: {max(lengths)}, avg: {sum(lengths)/len(lengths)}')
```

```
# Create a selection and print the objects
bim_model.filter(Type='Wall', IsExternal=True).select()
for wall in bim_model.filter(Type='Wall', IsExternal=True,
 Length=max(lengths)):
    print(wall)
```

The mappers of the briqpy.ObjectRange are chainable. This is an example to get the parts of the roof that are close to a wall:

```
# Get all parts of roof within 40cm range of all walls
roof_parts = bim_model.filter(Type='Roof').parts()
```

```
roof_parts_close_to_wall = bim_model.filter(Type='Wall').within_distance(40, 'cm',
 search_range=roof_parts)
```

Whereas this is an example of a function statement to filter:

```
# Filter roof parts longer than 50 project units
def is_long(obj):
return obj.prop('Length') > 50
roof_parts.filter(is_long)
```

You can also export and show data in various formats:

```
# create a dictionary list
wall_info = [
    {   'Handle': wall.prop('Handle'),
        'Length': wall.prop('Length'),
        'Height': wall.prop('Height')
    } for wall in bim_model.filter(Type='Wall')]
```

```
# export to .json
import json
file = open('path/to/file.json', 'w+')
file.write(json.dumps(wall_info, indent=4))
file.close()
```

```
# plotting a histogram
import matplotlib.pyplot as plt
import pandas as pd
df = pd.DataFrame(wall_info)
df.hist();
plt.show()
```

```
# export to .csv
df.to_csv(r'path/to/file.csv', index = False, header=True)
```

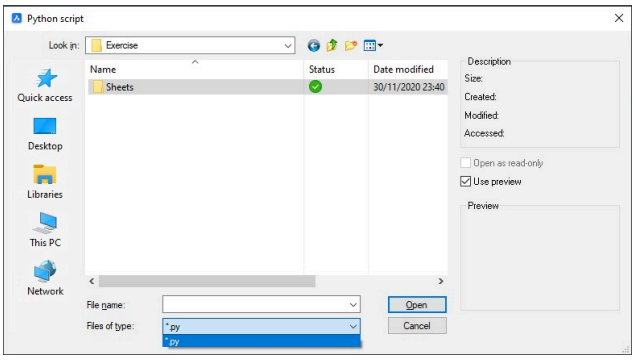
For more information on the BricsCAD API and its various classes, please visit the API chapter below.

20.4.9.5 Procedure: executing the Python script

1 Open a new or a BricsCAD file where you would like to run a Python script.

2 Type BIMPYTHON in the command-line and press Enter.

3 A dialog box is displayed where you can only select a Python script file (*.py). Select it and click **Open** to execute the file.



4 Unless you have specified the data to be exported or displayed in an external program outside BricsCAD, BricsCAD reports the output in the Command line panel.

20.4.9.6 API dictionary

Classes and syntaxes

class briqpy.Object- defines BricsCAD BIM Objects

```
prop(prop_name)
```

- Returns the property value of the Object with the given name.

```
distance_to(  other_obj, units='mm', distance_mode='exact' )
```

- Calculate the distance between two Objects.
- Options for argument *units*: any units value form insunits ('Centimeters', 'Feet', 'Parsecs' etc.) and these abbreviations: 'mm', 'cm, 'm', 'km', 'ft'.
- Options for argument *distance_mode*: 'bbox_center', 'bbox', exact'.

```
parts()
```

- Return the ObjectRange containing the sub-elements of this object.

```
parent()
```

- The opposite of **parts**(), returns the parent object of this sub-element.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Return the ObjectRange of objects that are within the distance of argument *distance*.
- Options for argument *units*: see distance_to.
- Options for argument *distance_mode*: see distance_to.

```
select()
```

- Add Object to selection

```
deselect()
```

- Remove Object from selection.

```
__eq__() and __hash__()
```

- Makes Object interoperable with e.g. python set or dictionary.

class briqpy.ObjectRange - defines collection of BricsCAD BIM Objects

```
filter( function )
```

- Filtering this range with a function parameter.

```
filter( **conditions )
```

- Filtering this range with conditions given as keyword arguments.

```
parts()
```

- Return the ObjectRange of all the parts of all the elements in this range.

```
parents()
```

- Return the ObjectRange of all parent objects of elements in this range.

```
within_distance(  distance, unit='mm', distance_mode='exact',
 search_range=bim_model )
```

- Returns the ObjectRange of objects that are within distance to any object in this range.

```
select()
```

- Add Objects to selection.

```
deselect()
```

- Remove Objects from selection.

```
__len__ ()
```

- Return the number of Objects in this range.

20.4.10 Classification codes

20.4.10.1 About

Assigning classification codes organizes the BIM and increases the level of detail. For instance, they allow to organize library materials and project reports, estimate project costs and define product specifications. Currently, **GUBIMClassCA**, **GUBIMClassES**, **Master Format**, **NLSfb**, **OmniClass**, **UniClass2015**, **Uniformat**, and **VMSW** are available classification systems in BricsCAD.

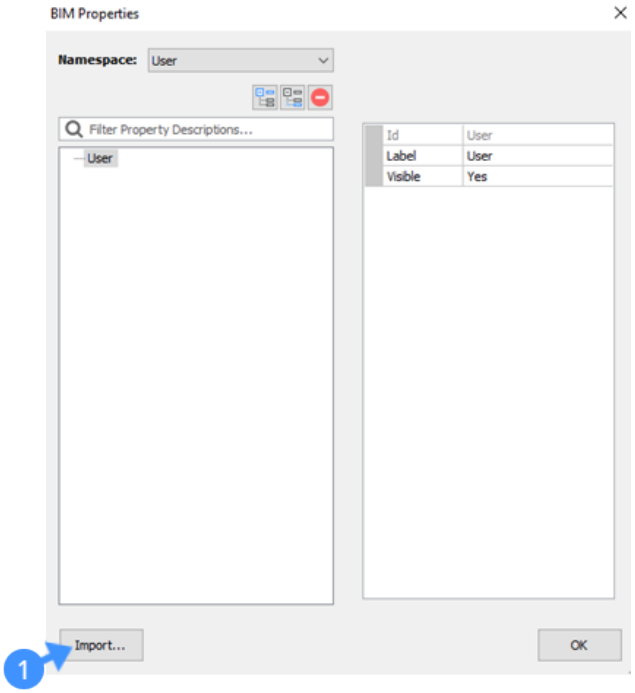
For more information about this command, visit the Command Reference article BIMPROPERTIES.

20.4.10.2 Assigning classification codes to the BIM

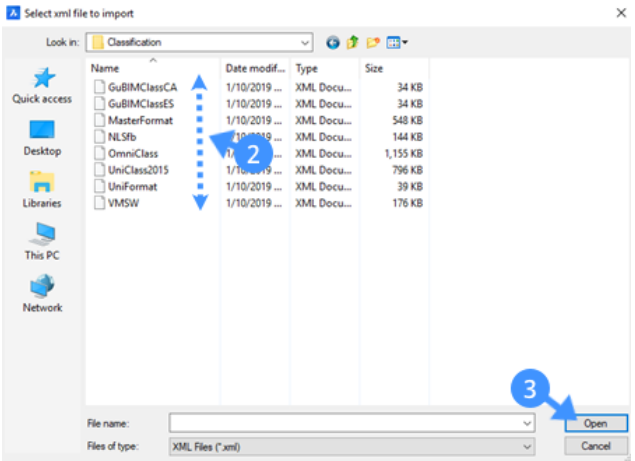
1 Open the BIM Properties dialog box from the Quad (**BIM** > **Model**).

Nota: Quad should be in **No Selection** state:

A dialog box appears:



2 Click the **Import** (1) button at the bottom-left corner of the dialog box. A file dialog displays:



3 Choose the classification system on the list (2), and click **Open** (3) to import it into the BIM.

Nota: If the file dialog does not show the **Classification** folder, you should manually enter this path in the address bar: C:\Users\<User>AppData\Roaming\Bricsys\BricsCAD V19x64\en_US\Sup Support \Bim\Classification.

Available classification systems:

GUBIMClassCA	Classification system for the BIM Users group of Catalonia.
GUBIMClassES	Unified classification system of building elements designed for the AEC industry in Spain.

Master Format	Standards for commercial and institutional building projects in the US and Canada.
NLSfb	Standards for the AEC industry in the Netherlands.
OmniClass	Classification system for the construction industry.
UniClass2015	Standards for all sectors of the UK construction industry.
Uniformat	Standards for classifying building specifications, cost estimating, and cost analysis in the US and Canada.
VMSW	Standards for the AEC industry in Belgium.

4 The classification system is added to the **namespace** (4), the selected classification system is added as a property set (5), and the classification code (6) is attached to this property set.

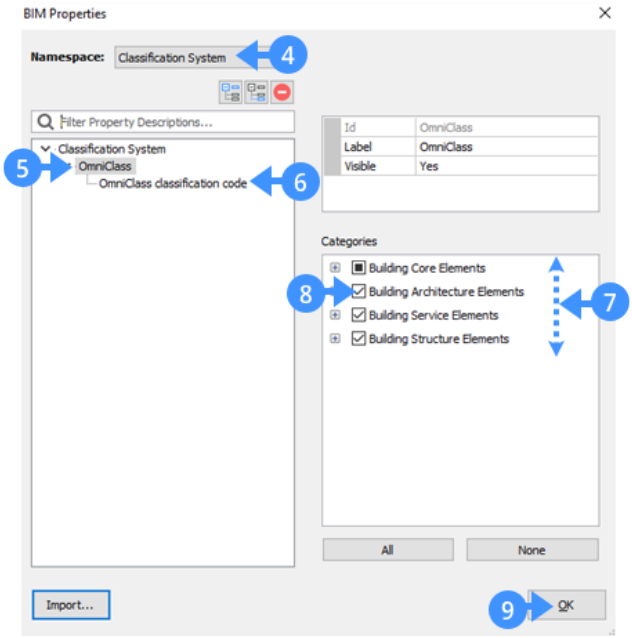
By default, the classification system is applied to all sorts of building elements (7).

Optional: Deselect the categories of building elements to which the classification system is applied.

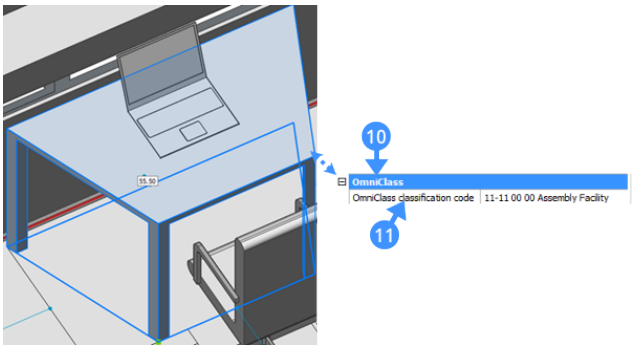
To do so, check the box next to the category name (8) (all elements in this category will be excluded), or expand the list and check the box next to the element type only.

Click the **OK** button (9) to finalize the procedure.

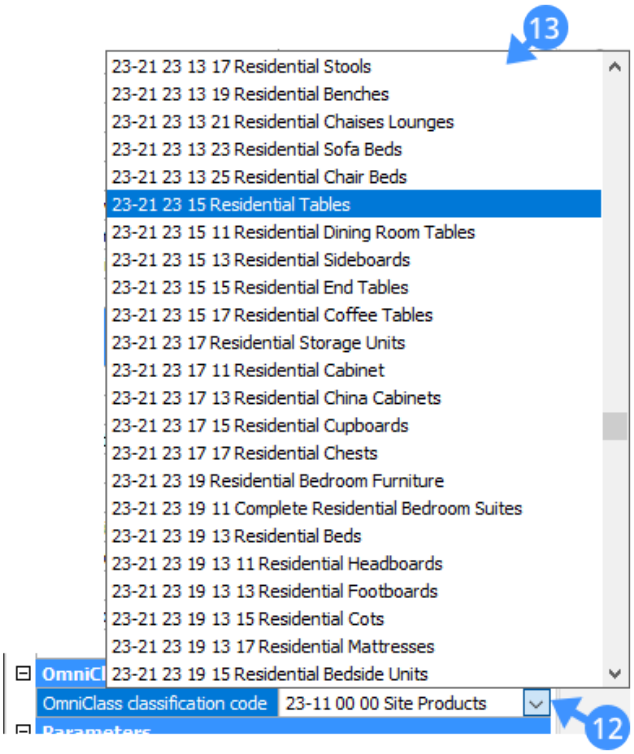
Orimport another classification system following the previous steps.



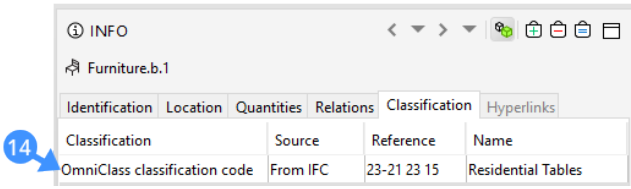
5 Select the element to which you want to assign a classification code, and open the properties panel. The classification system (10) and code (11) should be available in the properties panel.



6 Click the arrow icon on the right-hand side (12), and select the classification code for the selected element in the context menu (13).



Nota: Exporting the BIM to an IFC file contains the classification system for each element with assigned classification codes. These properties are exported with proper labels and descriptions. The following image shows the OmniClass Classification (14) in Solibri after exporting and importing BIM with the **OmniClass Classification code** from Bricsys to Solibri.



20.4.11 Creating and editing BIM spaces

20.4.11.1 Commands

BIMSPACE, BIMUPDATESPACE, BIMIFY

20.4.11.2 About

You can create BIM Spaces from enclosed spaces. When the wall elements enclose a space in your model, you can use the **BimSpace** tool to define a BIM Space. These spaces are solids so they can be easily manipulated. The 3D space geometries added to the current model are associated with space tags. The space tag is a block that contains a hatch and attributes. The default attributes of the Space tag include a space name and a space number. The values of each attribute can be changed using the properties panel.

For more information about this command, visit the Command Reference article BIMSPACE.

20.4.11.3 Defining a BIM space

1 Select the **Space** tool under **Home** > **Classify** > **Space or** type BIMSPACE in the Command line.

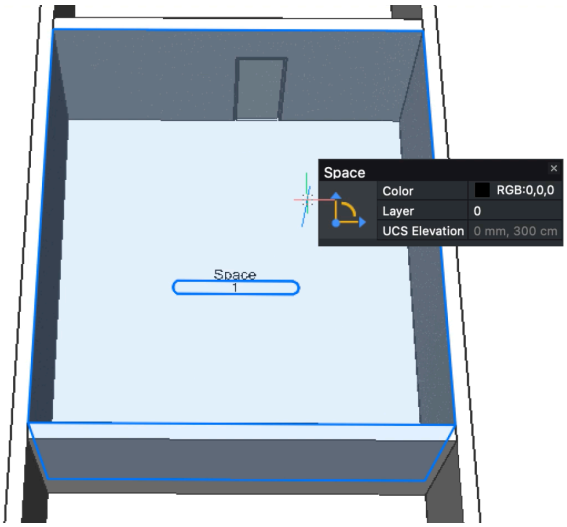
You are prompted:*Pick a point inside the space or [Edit];*

2 To define a BIM space, pick a point inside the space area in your BIM model.

3 Move the cursor inside the space area to pick a point, the **Dynamic UCS** enables you to define the bottom plane of the space. A green checkmark will appear next to the cursor to indicate a possible location. A red cross will appear if the plane isn't available.

4 In this plane, an area enclosed by wall elements or any space bounding elements is found and a space tag is placed in the center of the space.

5 **Space Bounding elements** can be Walls, curtain walls, slabs, columns, but also hatches and polylines can be counted as space bounding elements.



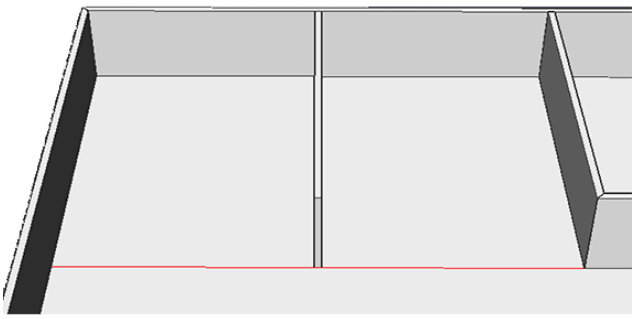
6 Edit the properties of the BIM Space in the properties panel.

7 Change the representation from solid to footprint for a less notable visualization.

20.4.11.4 Procedure: creating BIM spaces from virtual boundaries

There are cases in which a virtual boundary is needed. For example, separate parking spaces, large auditorium spaces for energy analytics, etc.

1 Divide the Space with linear entities. This can be hatches or lines.

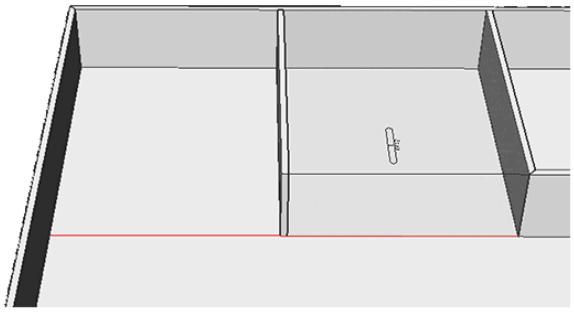


2 Select the dividing elements (the red line).

3 Classify them as a space bounding element: **Home** > **Classify** > **Wall**.

Orin the Quad under **BIM**.

4 Type BIMSPACE in the Command line and pick a point inside the space.



Nota: If the line doesn't divide the space, go to the **Properties** panel and then to **BIM**. Check if **Space bounding** is turned on.

20.4.11.5 Creating BIM spaces automatically

BIMIFY and BIMQUICKBUILDING create BIM spaces and automatically assign information to them. For example, the story and building of the space (see BIMSPATIALLOCATIONS). The commands will search for space bounding elements and create spaces wherever possible.

These spaces act like manually made spaces and can be edited according to your own needs.

20.4.12 Custom properties

20.4.12.1 Commands

BIMPROPERTIES

20.4.12.2 About

BricsCAD provides a way to assign your own properties to desired building elements in a single command. This is called assigning custom properties. The approach is first to create a custom property set, then assigning it to the building elements to which the property set is to be applied. The method is carried out by the BIMPROPERTIES command, which allows you to create, edit, and delete the custom property set in a dialog box. This dialog box also allows you to add properties to the currently selected property set. Important to note is that these custom properties are also included in IFC export.

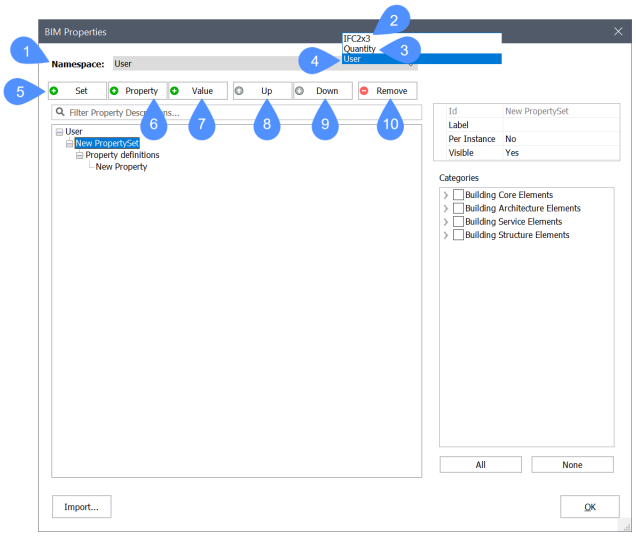
For more information about this command, visit the Command Reference article BIMPROPERTIES.

To learn more about creating user-defined quantity definitions, visit the Procedure article for Customizing BIM Quantities.

20.4.12.3 Using the BIM properties dialog box

1 Open the **BIM Properties** dialog box by typing BIMPROPERTIES in the Command line. The dialog box displays for specifying and editing properties of BIM projects. The following illustration shows the

dialog box.



(1) Namespace	Groups property sets in namespaces.
(2) IFC2X3 namespace	Shows the IFC2x3 properties.
(3) Quantity namespace	Shows some hardcoded quantities ***Nota:*** These have been deprecated and replaced by the Quantities in the **IFC2x3** namespace.
(4) User namespace	Shows the custom properties.
(5) Add property set button	Creates a new property set.
(6) Add property button	Creates a new property definition.
(7) Add value button	Creates a new value definition.
(8) Move up button	Moves the selected property or value up the list.
(9) Move down button	Moves the selected property or value down the list.
(10) Remove button	Removes the selected property set, property definition or value definition.

2 To create a custom property set, select the **User** option in the namespace drop-list, then click the Add

Property Set. A new property set is added to the **User** space. To edit the property set do the following:

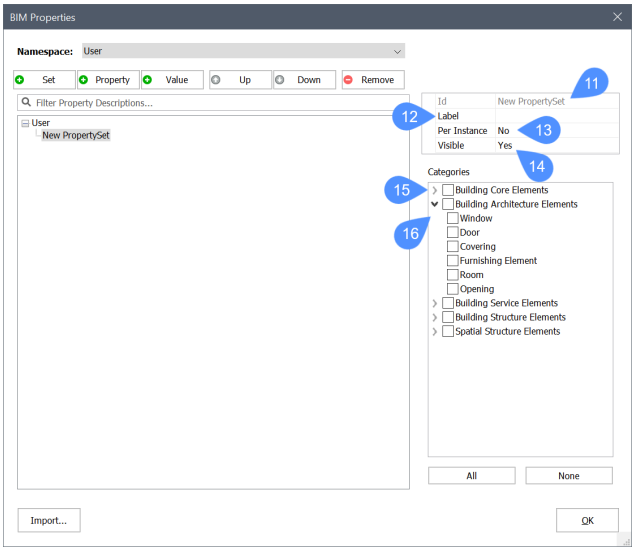
- Type a name in the **Id** (11) field.
- Type a name in the **Label** (12) field.

The **Label** name appears in the properties tree. If the **Label** name is not defined, the **Id** name displays. The **Id** must be unique.

Optionally, you can choose to have the custom property be only applied to specific instances (13), and choose the visibility (14).

- Select the building element categories (15) the property set applies to.

When you expand one of the categories, a different type of building elements (16) displays, check the box next to its name to associate with the property set. The following illustration shows the **BIM Properties** dialog box after inserting a new property set.

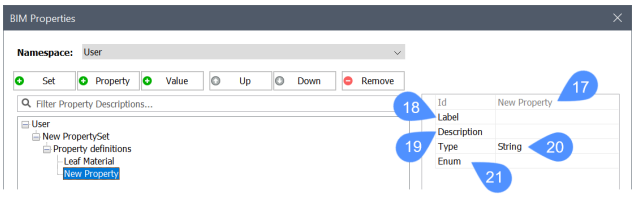


3 To add a property to the currently selected property set, first, select a property set, then click the Add Property button to add a new property.

4 To edit the property do the following:

- Type a name in the **Id** (17).
- Type a name in the **Label** (18).
- Optionally, type a description in the **Description** (19).
- Select the property type. The options are **Boolean**, **Integer**, **Real**, and **String** (20).
- Optionally, define enumerated values (21).

The following illustration shows the **BIM Properties** dialog box after inserting a new property.



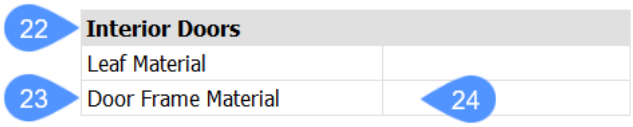
5 To remove the property set/property, select a property set/property in the tree, then click the **Remove** button to remove.

6 Press the **OK** button to accept it.

Now if you select a building element, which is associated with the property set, you will see its newly assigned properties (22) are placed in a new category (23) in the properties panel.

To assign a value to your custom property, select it in the properties panel, then type a new value in its field (24).

The following illustration shows the custom properties in the properties panel.



20.4.13 Customizing BIM quantities

20.4.13.1 Commands

BIMPROPERTIES

20.4.13.2 About

Entities have a set of properties which can be viewed in the **Properties** Panel. If an entity has been assigned a certain **BIM classification** or **Type** (e.g. **Wall** or **Slab**), additional properties will be displayed such as IFC Common properties and Base quantities. Which properties are displayed in the panel is managed in the **BIM Properties** dialog, which is opened by the BIMPROPERTIES command.

20.4.13.3 Namespaces

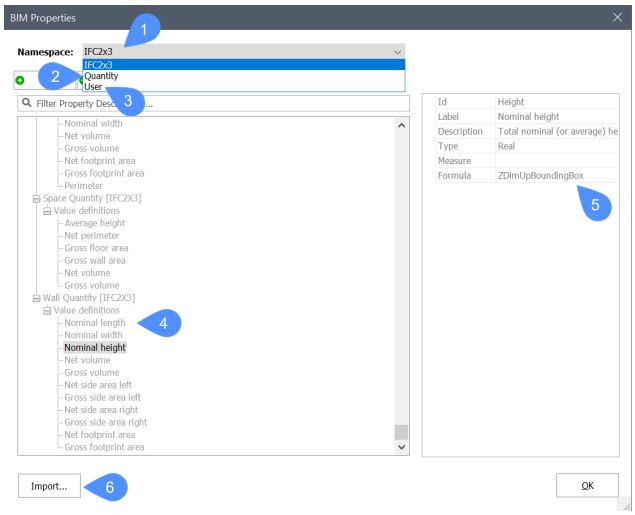
In the BIM Properties dialog, by default three namespaces are available: **IFC2x3** (1), **Quantity** (2) and **User** (3). The **IFC2x3** namespace contains properties that are defined by the IFC2x3 schema. This contains both common properties (e.g. Wall Common properties, such as **Fire Rating** and **Acoustic Rating**), and Quantities (e.g. **Wall Quantity** > **Nominal lenght** (4)). These Quantities are based on so-called Core quantities (5) of the solid. See Core quantities for more information.

The **Quantity** namespace contains some hard-coded quantities of BIM objects also based on the Core quantities, much like in the **IFC2x3** namespace.

> ***Nota:*** This **Quantity** namespace is deprecated. It has been replaced by the Quantities in the **IFC2x3** namespace, however it is still available in the **BIM Properties** dialog for backward compatibility with versions V20 and older.

The **User** namespace exists for the user to create their own property definitions. See also the Procedure article Custom Properties for more information on custom properties.

It is also possible to import namespaces, e.g. the **IFC4** namespace. This can be done by clicking the **Import** button (6) on the bottom left.



Nota: using this method the namespace will only be imported for this particular project. If you want namespaces to be imported over all your projects, open the **Settings** dialog and under the **BIMGeneralDefault properties** path you can add a new path. By default, it should take you to *C:\Program Files\Bricsys\BricsCAD Vxx en_US\Support*, where you can find for instance the **bimproj_IFC4.xml** containing the **IFC4** namespace.

20.4.13.4 Core quantities

Core quantities are quantities of solids that are calculated based on the geometric features of that solid, and that can be used in user-defined quantities. These core quantities are also the basis for the quantities defined in the **IFC2x3** and **IFC4** properties as described in the previous section.

For example, under **Wall Quantities** in the **IFC2x3** namespace, the **Nominal Height** of a wall is defined as the **ZDimUpBoundingBox** of the solid, which is a Core quantity. A full list of Core quantities can be found below.

Core Quantity	Description	Measure	Type
NumberOfPlies	Number of plies in the attached composition.		**Integer**
XDimUpBoundingBox	X dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
YDimUpBoundingBox	Y dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**
ZDimUpBoundingBox	Z dimension of the bounding box found with global Z axis rotation freedom.	**Length**	**Real**

Core Quantity	Description	Measure	Type
XDimFreeBoundingBox	X dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
YDimFreeBoundingBox	Y dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
ZDimFreeBoundingBox	Z dimension of the bounding box found with 3D rotation freedom.	**Length**	**Real**
DistanceBetweenMajorSurfaces	Distance between major surfaces.	**Length**	**Real**
FirstMajorSurfacePerimeter	Perimeter of the first (largest) major surface.	**Length**	**Real**
SecondMajorSurfacePerimeter	Perimeter of the second major surface.	**Length**	**Real**
LinearAxisLength	Length of the linear element's axis.	**Length**	**Real**
VariablePlyThickness	Thickness of a variable ply.	**Length**	**Real**
FirstMajorSurfaceNetArea	Area of the first (largest) major surface after subtracting openings.	**Area**	**Real**
FirstMajorSurfaceGrossArea	Area of the first (largest) major surface before subtracting openings.	**Area**	**Real**
SecondMajorSurfaceNetArea	Area of the second major surface after subtracting openings.	**Area**	**Real**
SecondMajorSurfaceGrossArea	Area of the second major surface before subtracting openings.	**Area**	**Real**
IsMajorSurfacesParallel	Indicator if the major surfaces are parallel.		**Boolean**
TotalSurfaceNetArea	Area of the solid's surfaces after subtracting openings.	**Area**	**Real**
TotalSurfaceGrossArea	Area of the solid's surfaces before subtracting openings.	**Area**	**Real**

Core Quantity	Description	Measure	Type
FootprintNetArea	Area of the bottommost solid's surfaces after subtracting openings.	**Area**	**Real**
FootprintGrossArea	Area of the bottommost solid's surfaces before subtracting openings.	**Area**	**Real**
ProjectedNetArea	Area of the solid's surfaces projected to global XY plane after subtracting openings.	**Area**	**Real**
ProjectedGrossArea	Area of the solid's surfaces projected to global XY plane before subtracting openings.	**Area**	**Real**
CrossSectionNetArea	Area of the cross section for linear elements after subtracting openings.	**Area**	**Real**
OuterSurfaceNetArea	Area of the outer surface for linear elements after subtracting openings.	**Area**	**Real**
NetVolume	Volume of the solid after subtracting openings.	**Volume**	**Real**
GrossVolume	Volume of the solid before subtracting openings.	**Volume**	**Real**

20.4.13.5 User-defined quantities

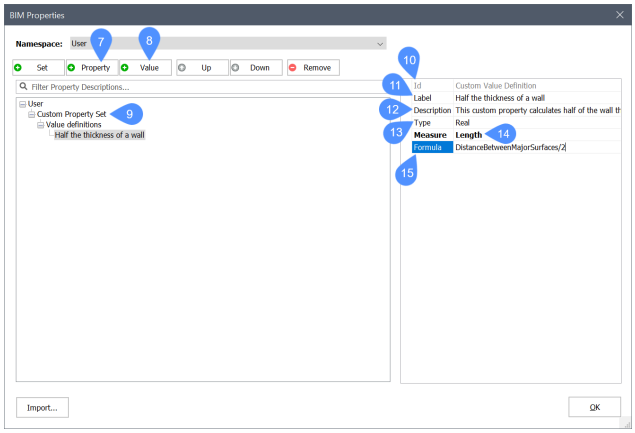
In the **User** namespace it is possible to create custom properties inside a custom property set using the **Property** (7) button, see Procedure article Custom Properties.

It is also possible to create user-defined quantities, using the **Value** (8) button. In the image below, a user-defined quantity was created in a **custom property set** (9). This user-defined quantity (or Value Definition) has the following attributes:

- **Id** (10): a unique identifier.
- **Label** (11): the name of quantity which will show up in the properties panel. This can be the same as the ID, however **the ID must be unique**.
- **Description** (12): a description of the quantity.
- **Type** (13): the type of the quantity. Typically, a quantity will be defined as a **Real** value.
- **Measure** (14): the measure of the quantity, i.e. is it a **length** value, an **area**, **volume** or **mass**?

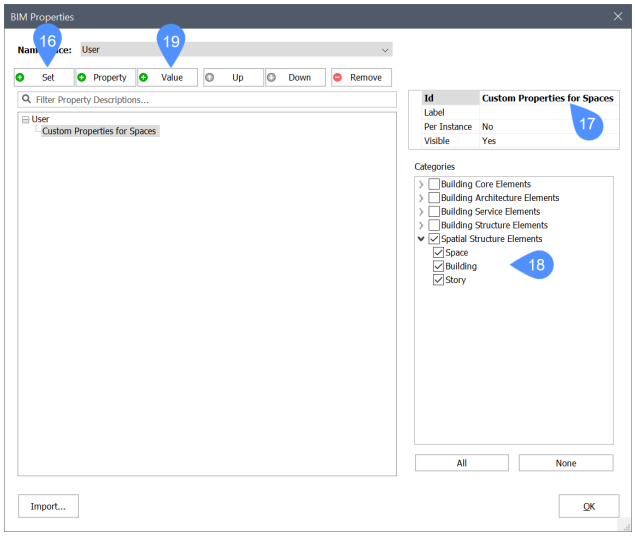
- **Formula** (15): the actual expression of the quantity. Here Core quantities can be used, as well as mathematical expressions as defined in the Procedure article Working with Parameters and Constraints.

In this example, a user-defined quantity is defined that displays half the thickness of a wall, which is calculated as **DistanceBetweenMajorSurfaces/2**.



20.4.13.6 Procedure: creating a user-defined quantity

In this part you will learn how to make user-configurable quantities. By way of example, we will define a quantity of Spatial objects such as a room. The quantity will be the **Compactness** which can be calculated as the volume of the space divided by the sum of the areas on all sides of the space.



1 Open the **BIM Properties** dialog by typing BIMPROPERTIES in the Command line.

2 Set the **User** namespace as current namespace.

3 Create a new Property Set by clicking the **Set** button (16), and give it a name (17).

4 Assign the Property Set to one or more categories (18). In this example we will assign it to spatial elements.

5 Create a new Value Definition by clicking the **Value** button (19).

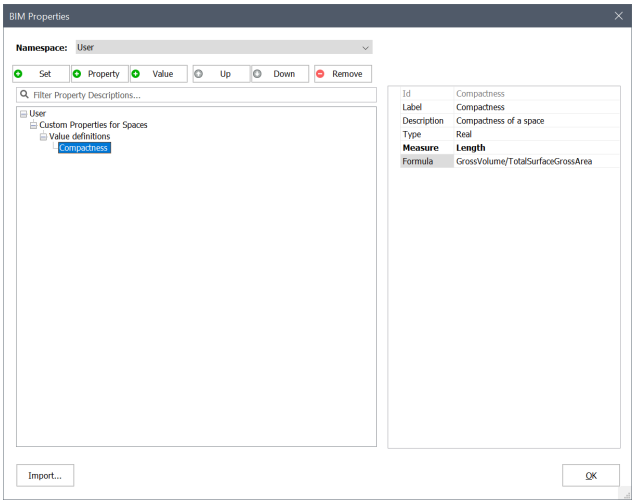
6 Give this Value Definition an **Id** and a **Label**, e.g. **Compactness**.

7 **(Optional)** Give this Value Definition a description.

8 Assign the correct **Type** to this Value Definition. Because our Value Definition will be based on a calculation of two **Real** values, this will also be a **Real** value.

9 Assign a **Measure** to the Value Definition. Because the result of the calculation will be a volume divided by an area, we will end up with a **Length** measure.

10 Fill in an expression in the **Formula** field. The volume of a space is a Core quantity with the name **GrossVolume**. The sum of areas of the space on all sides is also a Core quantity with the name **TotalSurfaceGrossArea**. Hence, we can define the **Compactness** = **GrossVolume/ TotalSurfaceGrossArea**.



11 Click **OK** to close the **BIM Properties** dialog.

12 Create a volume that represents a space, either by creating surrounding elements like walls and a slab, and by defining a space inside using the BIMSPACE or BIMIFY commands, or by creating a solid and using BIMCLASSIFY to classify it as a **Space**.

13 Select the space. In the **BIM Properties** panel you should now see your user-defined property under a separate property set.

20.4.14 Load bearing direction for slabs

20.4.14.1 Commands

BIMSETLOADBEARINGDIRECTION

20.4.14.2 About BIMSETLOADBEARINGDIRECTION

In BricsCAD BIM, the BIMSETLOADBEARINGDIRECTION command allows you to define 0, 1, or 2 load bearing directions for a slab.

After defining the load bearing direction in the 3D model, you can indicate the load bearing direction in the floorplan using the **Manual** option of the BIMTAG command.

20.4.14.3 Specifying the load bearing direction(s)

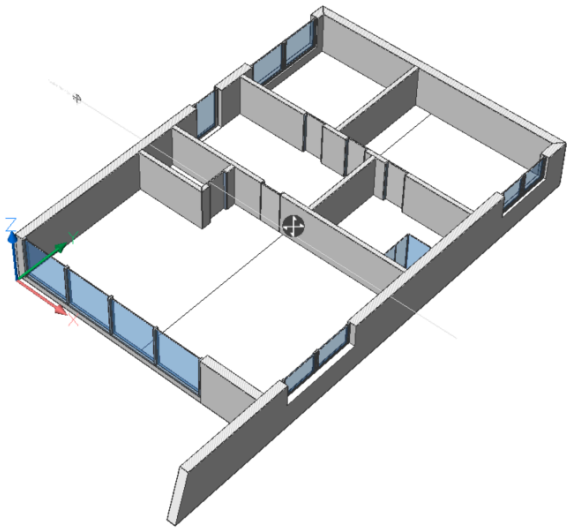
Only solids which are classified as a **BIM Slab** can be processed by the BIMSETLOADBEARINGDIRECTION command. Use the BIMIFY or BIMCLASSIFY command to classify a solid as a **BIM Slab**.

1 Launch the BIMSETLOADBEARINGDIRECTION command.

1 **You are prompted:** Select a slab

2 Click the slab. It may help to set the **Clip Display** property of a plan section to select the slab.

A widget displays at the center of the slab.



3 Repeatedly click the widget. Press Enter or right-click to confirm the loadbearing direction(s).

20.4.14.4 Adding a load bearing symbol in plan section

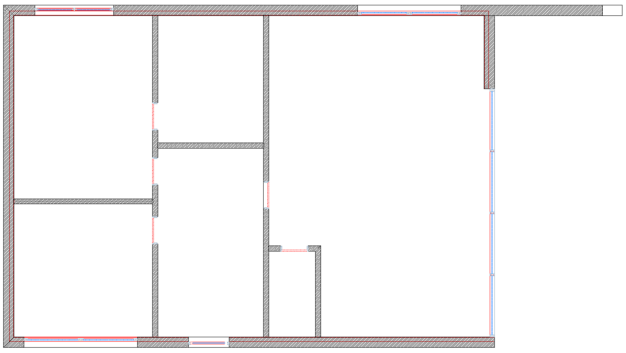
1 When you insert the load bearing tag in a plan section, you need to select a line, which is generated by the slab. Since slabs usually overlap with walls, you need to set the **Hidden Lines** > **Show** property of the **BIM Section** entity to **Yes**.

Hidden Lines

Show	Yes
Color	Red
Layer	0
Linetype	Continuous
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer

Next, execute the BIMSECTIONUPDATE command of the plan section.

2 Open the **Plan** section.



3 Launch the BIMTAG command.

4 Choose the **Manual** option.

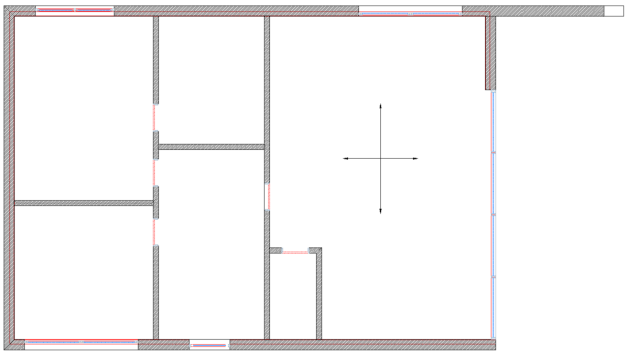
5 Choose the **Change Tag Type** option.

6 Choose the **Load-Bearing Direction** tag type.

7 Click on a point on the edge of the slab.

8 The tag displays, attached to the cursor.

9 Click to place the tag.



20.4.15 Physical materials

20.4.15.1 Commands

BLMATERIALS

20.4.15.2 Opening the physical materials editor

Do one of the following:

- Launch the BLMATERIALS command.
- Click the **Browse** button (...) of the **Material** property of a component in the **Mechanical Browser**.

Component	
Name	support_final
Description	
File	C:\Program Files\Bricsys\E
Extension type	
Insert as	External component
Material	Aluminium ... x
Name	Aluminium
Description	
Density	2700.00 kg/m³

20.4.15.3 Creating physical materials

You can create physical materials in either the Project or the Library database. You can create new materials from scratch or copy them from an existing material.

Creating a physical material from scratch

Select a material in either the **In Project** or the **In Library** material list.

Do one of the following:

1 Click the **New Material** button ( ) on the **Physical Materials** dialog.

2 Right-click and choose **New Material** in the context menu.

A material named **New** is added to the selected database. Now you can edit the new material.

Adding a physical material as a copy of an existing material

Select a material in either the **In Project** or the **In Library** material list. Do the following:

1 Right-click on the material you want to copy and choose **Duplicate** in the context menu.

2 A material named **<existing_material>-2** is added in selected database.

3 Adjust the copy of the material.

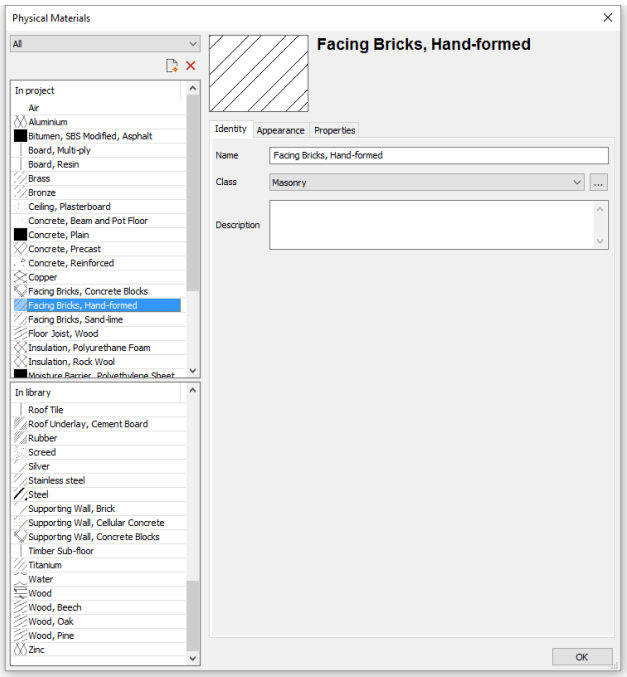
20.4.15.4 Editing a physical material

The various settings of a physical material definition are divided into three groups under three tabs on the **Physical Materials** dialog:

- **Identity**: **Name**, **Class** and **Description** of the material.
- **Appearance**: The BIM Materials layers the composition is composed of.
- **Properties**: Properties of building material such as **Cost**, **Manufacturer**, **Thickness**...

Select the material you want to edit in either the **In Project** or in the **In Library** list on the **Physical Materials** dialog.

Identity tab page

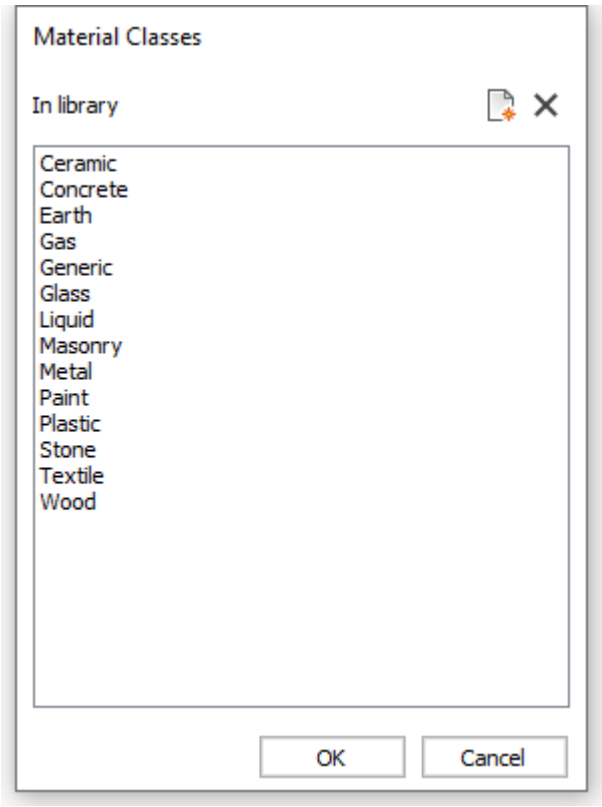


1 Type a **Name** for the material.

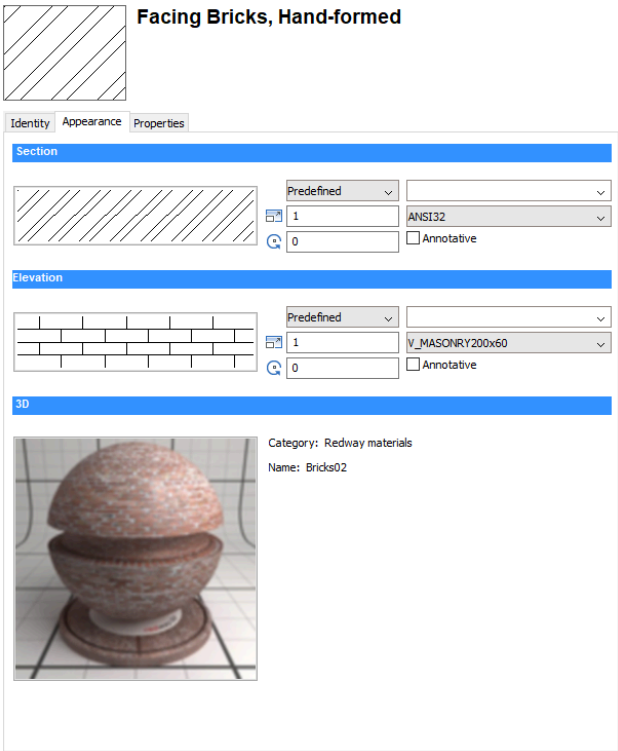
2 Select a **Class** from the drop-down list*.

3 **(Optional)** Type a **Description**.

* Click the **Browse** button (...) to open the **Material Classes** dialog where you can create new material classes.



Appearance tab page



- **Section settings**

The **Section** settings define the hatching of the section boundary lines.

a Click the **Hatch Type** button and select the type of the hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom**.

b Do one of the following to select a predefined or custom hatch pattern:

- Click the **Hatch** preview and select the pattern in the the **Hatch Pattern** dialog box.
- Click the **Hatch Pattern Name button** and select the pattern in the list.

c Define the **Scale**.

The final pattern scale in a section is the product of the **Scale** setting of the physical material and the value of the **Scale** property of the **BIM** section. As a result, the printed scale of the hatch pattern is independent of the section plane scale.

d Set the **Rotation** angle.

e In the **Layer** field, either type a name or click the arrow button and select a layer in the list.

f Optionally, check the **Annotative** option.

- **Elevation settings**

  The **Elevation** settings define the surface hatching of the section background.

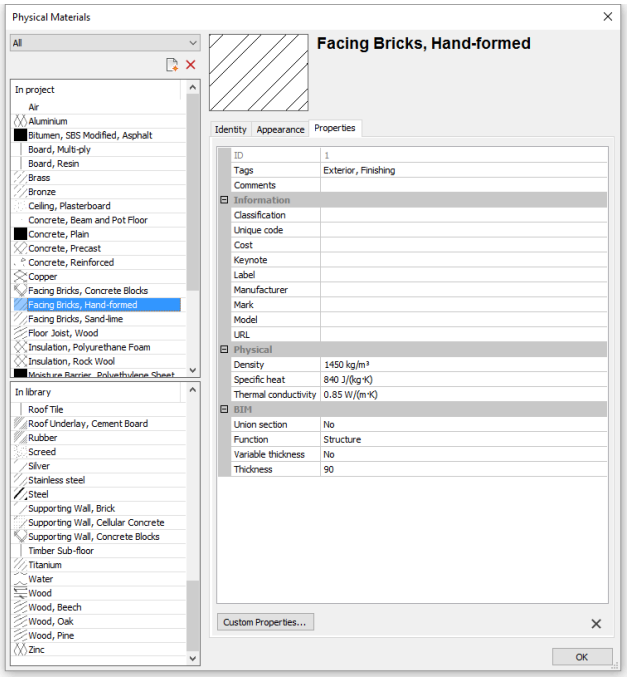
  See **Section** settings above, except for the **Scale** setting. The elevation hatch is independent of the **Scale** property of the **BIM** section.

- **3D settings**

  The **3D** settings define the rendering material.

  Double-click the preview image and select a material in the **Select Render Material** dialog box.

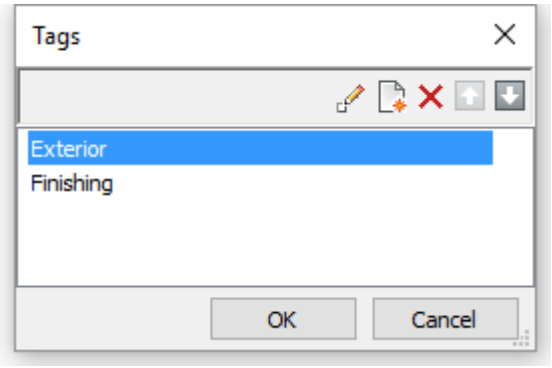
Properties tab page



1 Edit the **Tags** field: tags are separated by commas.

Tags are case sensitive.

Click the **Browse** button (...) to open the **Tags** dialog.



2 Click the **New Item** button () to add a new tag at the bottom of the list.

3 Select a tag in the list and click a tool button:

- : Edit the selected tag.
- : Delete the tag.
- : Move the tag up.
- : Move the tag down.

4 **(Optional)** Expand the **Information** properties and fill out the necessary fields.

5 **(Optional)** Expand the **Physical** properties and specify the physical properties of the material.

6 Expand the **BIM** properties.

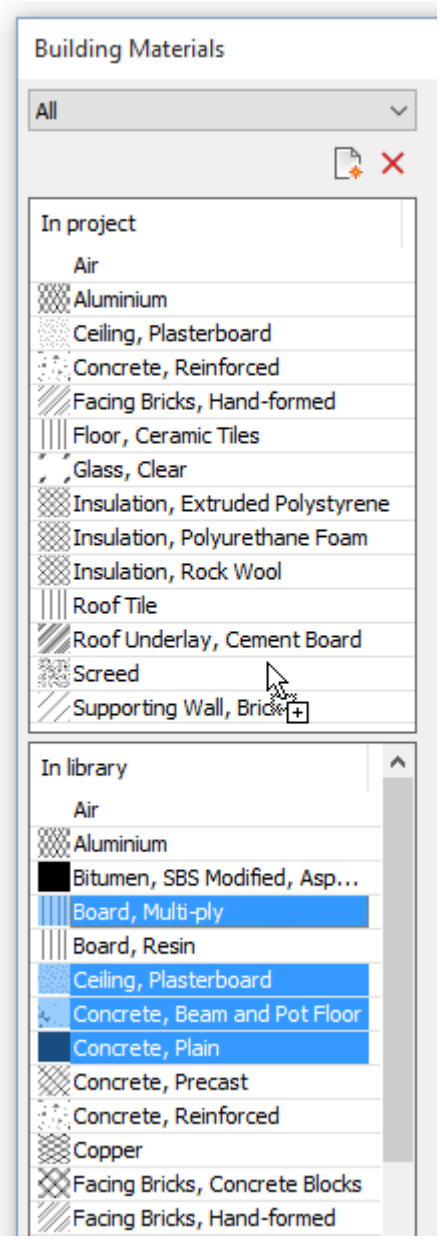
- **Union Section**: Select **Yes** to remove connection lines in sections between different solids of this material. Select **No** to keep the connection lines.
- **Union Elevation**: Select **Yes** to remove connection lines in elevations between different solids of this material. Select **No** to keep the connection lines.
- Select a **Function** from the drop-down list.
- **Variable Thickness**: select **Yes** to allow a variable thickness. Select **No** to not allow a variable thickness.
- Type a default thickness in the **Thickness** field.

7 Click **OK**.

20.4.15.5 Copying materials between the project and library databases

1 Select the material(s) in the source database.

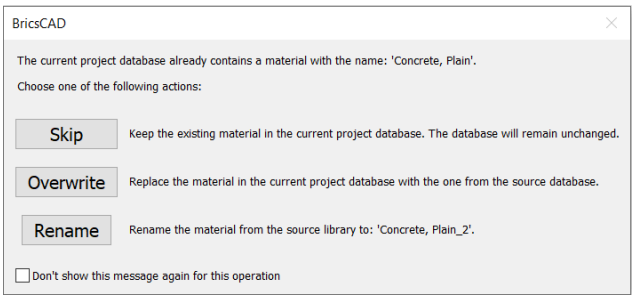
1 Press and hold the Ctrl-key to select multiple materials.



2 Drag the materials to the target database.

3 The selection set is copied.

If one or more materials already exist in the target database, an alert box displays:



Click the appropriate button to either overwrite, duplicate or skip the existing material. If the **Apply to all materials in this operation** option is selected, the option applies to all existing materials. Duplicated materials have an index number added to their name.

20.4.15.6 Deleting physical materials

1 On the **Physical Materials** dialog box, select the material(s) in the Project or Library database.

2 Click **Delete**.

Nota: You cannot delete materials that are used in a composition or a mechanical component.

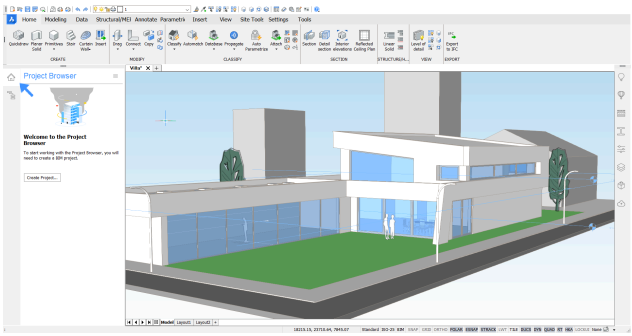
20.4.16 Project browser

20.4.16.1 Commands

DATAEXTRACTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

20.4.16.2 The project browser panel

You can find the **Project Browser** panel on the left-hand side of the screen.



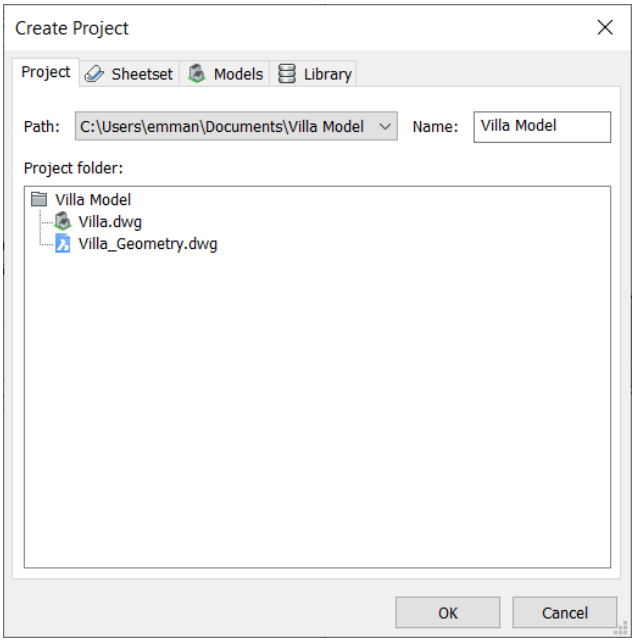
Nota: If the **Project Browser** doesn't display in the panel, right-click in the panel and select **Panels** > **BIM Project Browser**

20.4.16.3 Create a BIM project

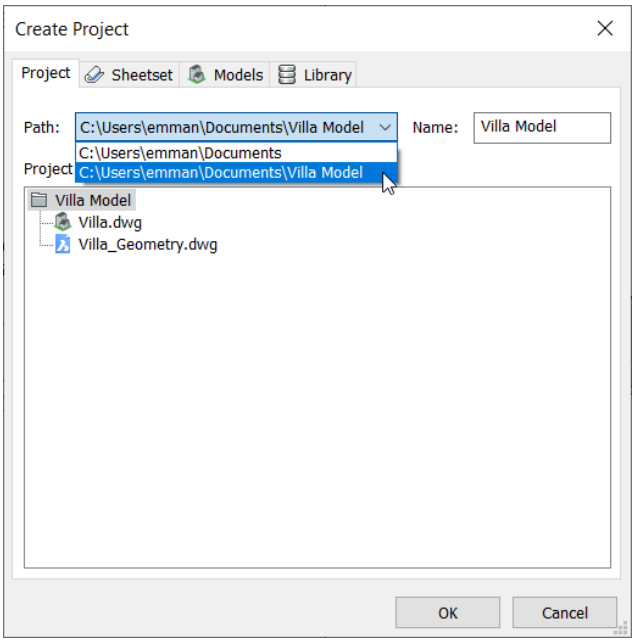
To create a BIM project, you must select one or more model drawings.

Project

1 Click the **Create Project...** button in the **Project Browser**. A dialog box appears.



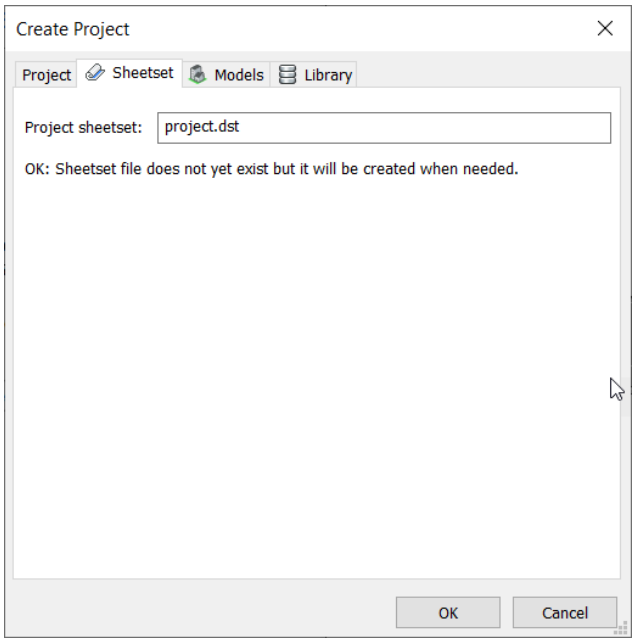
2 **(Optional)** Click the **Path** list button and select a parent folder.



3 **(Optional)** Type a name in the **Name** field (the default name is the name of the parental folder).

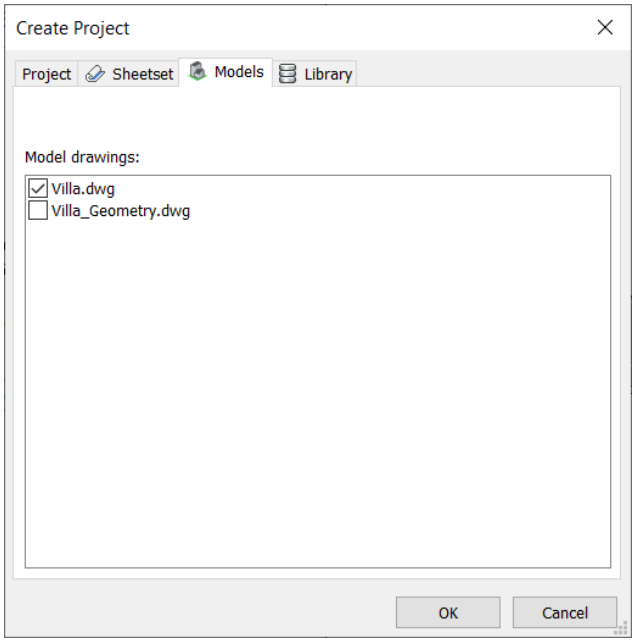
Sheetset

Click the **Sheetset** tab. Enter a name for the **Project sheetset** (the default name is project.dst).



Models

Click the **Models** tab. Here, the files in the selected parent folder (and subfolders) are displayed. Check all the models you want to incorporate in your drawing.



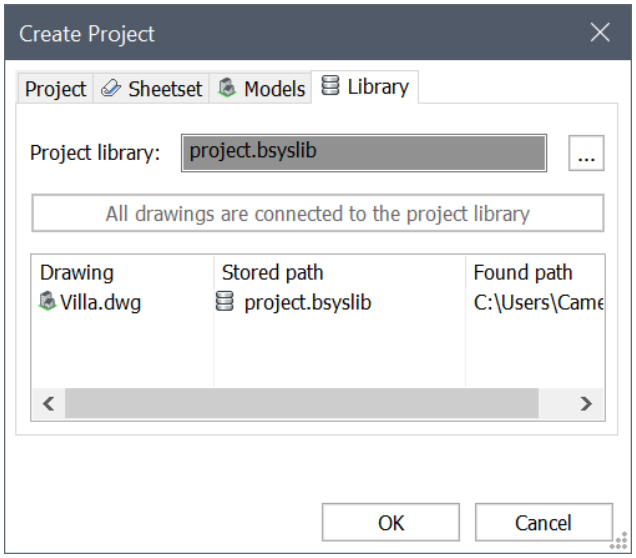
Library

The **Project Library** can either be **external** or **embedded**. Sharing the **project library** means that you can use the same library over different drawings, this is called an external library. The shared library is stored in an external BIM Database File with the .bsyslib extension. An embedded library stores the **Project Library** in the drawing file.

- To make an external library:

Nota: By default, the library is set to **Embedded**.

a Go to the **Library** tab. Click the **Browse** button (...) of the **Project Library** option. Enter a name for the **Project library** (the default name is project.bsyslib).



b Click **OK** to display the **Sheetset Setup** dialog box.

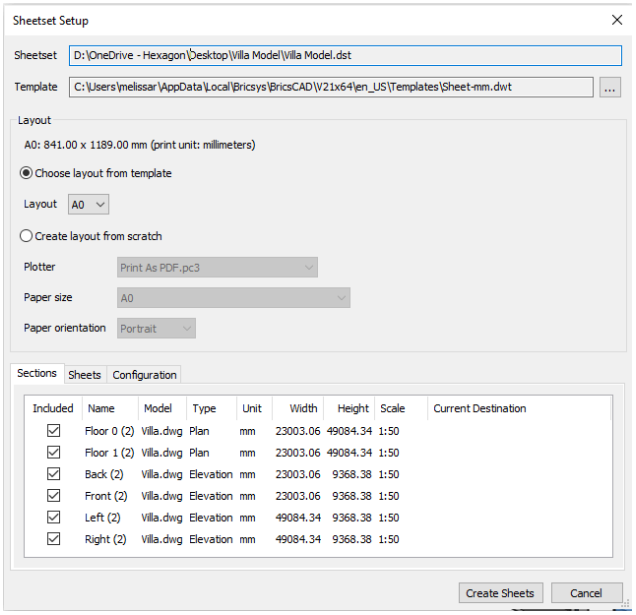
20.4.16.4 Sheetset setup

Sheetset: Shows the path of the sheetset file.

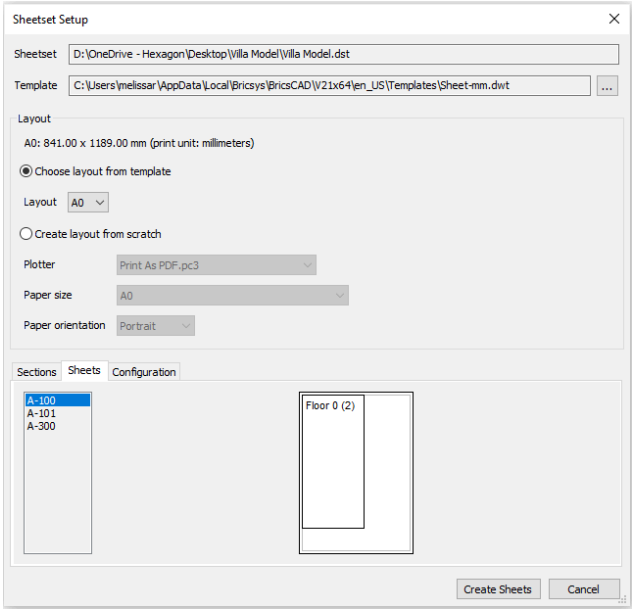
Template: Shows the path of the current template. Click **Browse** (...) to change the template.

Layout: Selects which layout (different paper sizes) the section results will be generated.

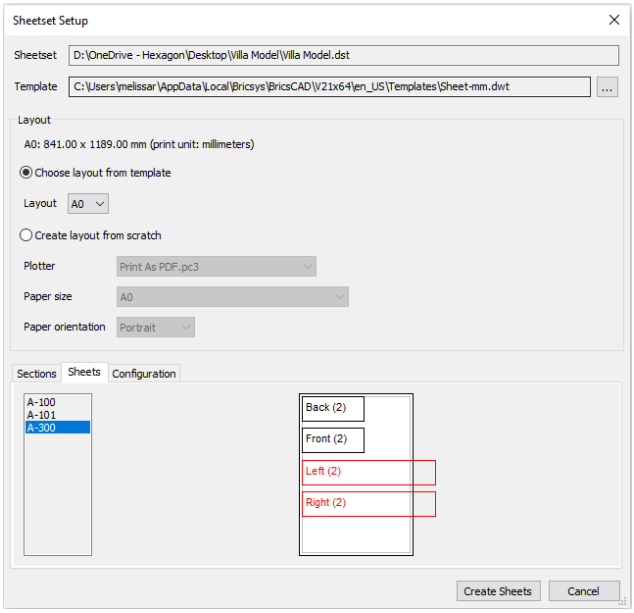
1 The **Sections** tab lists all sections. For each section with an empty **Current Destination** field, a viewport and sheet view are generated.



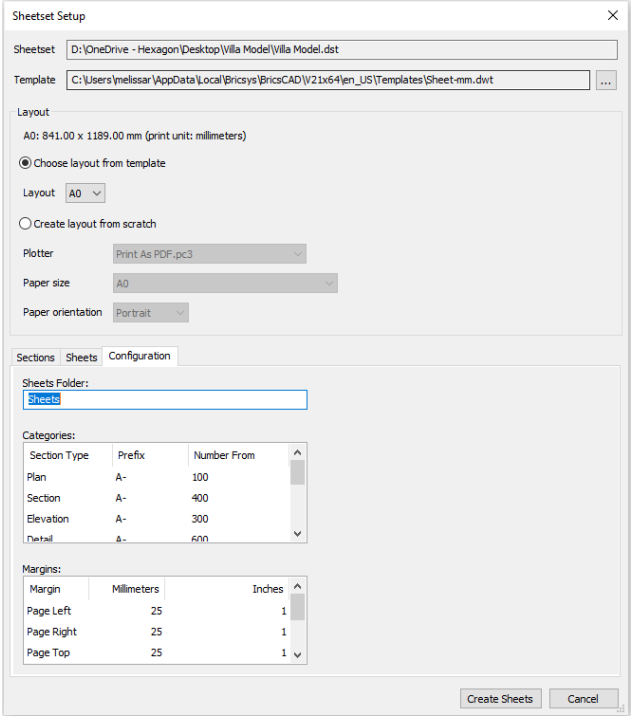
2 Click **Sheets** to preview each sheet (within the selected layout).



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

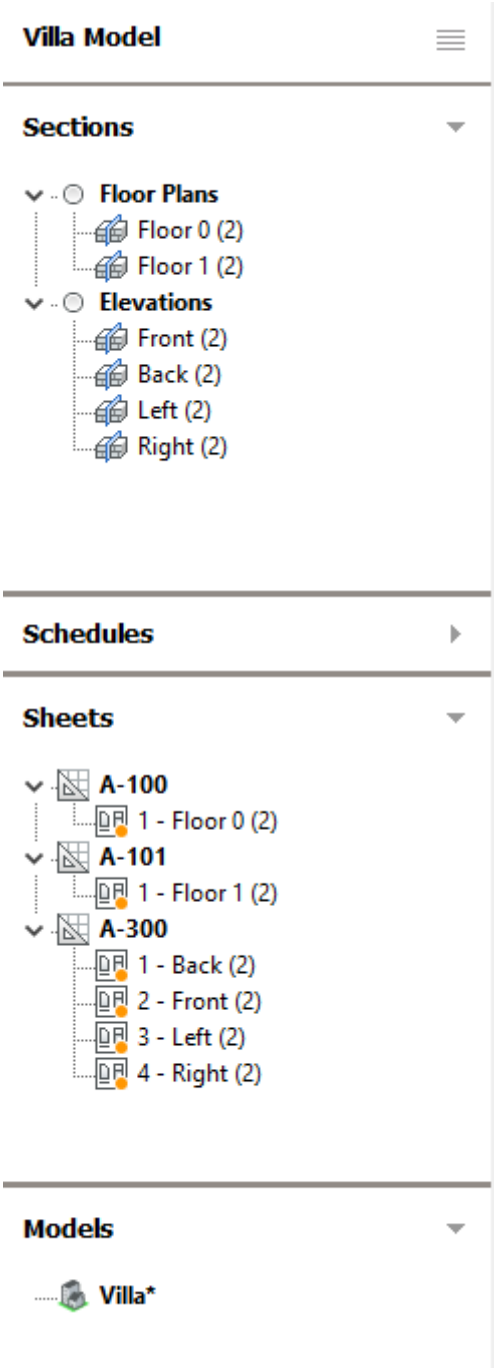


3 Click the **Configuration** tab. Slowly double-click (or click and hit Space/Enter) to change the properties. Under **Sheets Folder** you can change the name of the folder where the sheets will be stored.



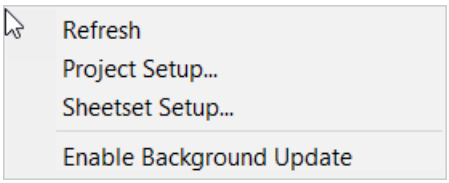
4 Click **Create Sheets**. An overview of the project displays and the sheets are generated.

Nota: The drawings with an orange fill circle icon are out of date. Right-click on them and select **Update** to match changes in the models included in the project.



20.4.16.5 Editing a project

Click on the **Menu** button (☰). A context menu displays:



- **Refresh**: Refreshes the Project Browser to match changes in the model.
- **Project Setup...**: Opens the Project Setup dialog box.
- **Sheetset Setup...**: Opens the Sheetset Setup dialog box.
- **Enable Background Update**: Calculates section results automatically (without interrupting user workflow). Sections are assigned a color; Queued: blue, Outdated: red and In progress: green.

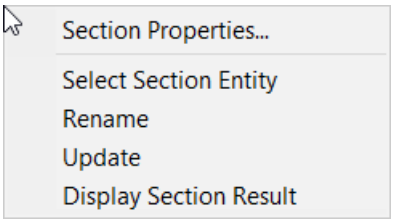
20.4.16.6 Creating schedules

See the Schedules article for more information about Schedules.

20.4.16.7 Working in the project browser

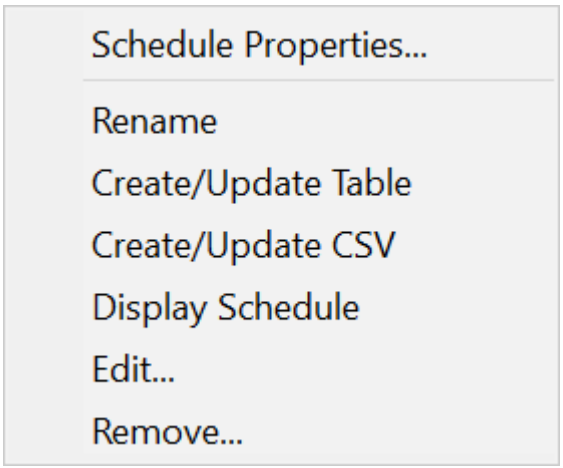
Right-click an item in the BIM **Project Browser** to display a context menu.

Sections



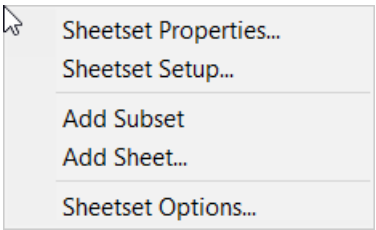
- **Section Properties...**: Opens the **Section Properties** dialog box where you can see and edit the properties of a section, such as the name and the result path.
- **Select Section Entity**: Selects the section entity in the project model drawing.
- **Rename**: Renames the section entity.
- **Update**: Recalculates the section result.
- **Display Section Result**: Displays the section result in the corresponding sheet and zooms in on its viewport. If not already open, the drawing is opened.

Schedules



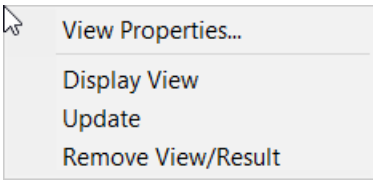
- **Schedule Properties...**: Opens the **Schedule Properties** dialog box that allows you to modify the properties of a schedule.
- **Rename**: Renames the schedule.
- **Create/Update Table**: Creates a sheet with the schedule or updates the sheet.
- **Create/Update CSV**: Creates a CSV file with the schedule information and saves it in the folder where the model file is located or updates the CSV file if it already exists.
- **Display Schedule**: Displays the schedule in the resulting drawing. If not already open, the drawing is opened.
- **Edit**: Opens the **Wizard Page** dialog box that allows you to edit the schedule.
- **Remove...**: Displays the **Remove** dialog box that allows you to choose between **Schedule Entity** and **Sheet View/Result** and removes the schedule.

Sheets



- **Sheetset Setup...**: Opens the **Sheetset Setup** dialog box that allows you to create a layout from scratch.
- **Add Subset**: Adds a subset in the **Sheets** tab.
- **Add Sheet...**: Adds a new sheet in a subset in the **Sheets** tab.
- **Sheetset Options...**: Opens the **Sheet sets** tab in the **Settings** dialog box that allows you to modify the sheet sets options.

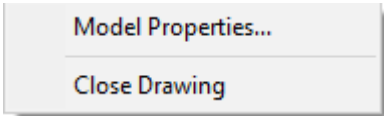
Sheet views



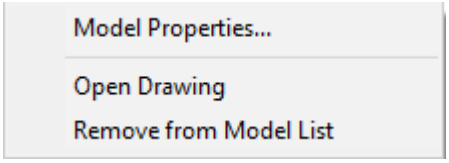
- **View Properties...**: Opens the **Sheet View Properties** dialog box that allows you to modify the properties of a sheet view.
- **Display View**: Displays the corresponding result and zooms in on its viewport.
- **Update**: Updates the sheet view. It executes the BIMSECTIONUPDATE command for the related **BIM** section.
- **Remove View/Result**: Removes the view and result from the corresponding sheet.

Model drawings

Context menu of a currently open model:



Context menu of a currently closed model:



- **Model Properties...**: Opens the **Model Properties** dialog box that allows you to see more information about the model.
- **Close Drawing**: For a currently open model, it closes the model.
- **Open Drawing**: For a currently closed model, it opens the model.
- **Remove from Model List**: Removes the drawing from the project model drawings.

20.4.17 Spatial locations

20.4.17.1 Commands

BIMATTACHSPATIALLOCATION, BIMSPATIALLOCATIONS

20.4.17.2 About

Spatial locations allow you to specify building and its stories in a project. In BricsCAD, a BIM model file can contain one site and multiple buildings and a building can contain multiple stories. When the entities are classified as a building element in the project, these classified entities have a **Building** and a **Story** property. Every building element in the BIM model resides in a particular story within a particular building.

For more information about this command, visit the Command Reference article BIMSPATIALLOCATIONS.

20.4.17.3 Procedure: assigning a building and story to a building element

1 Launch the BIMSPATIALLOCATIONS command.

2 Click the **New Building** icon ( ) to add a new building to the model.

3 Fill out the properties grid to specify a new name or value of the existing property.

4 Click the **New Story** icon ( ) to add a new story in the selected building.

5 Fill out the properties grid to specify a new name or value of the existing property.

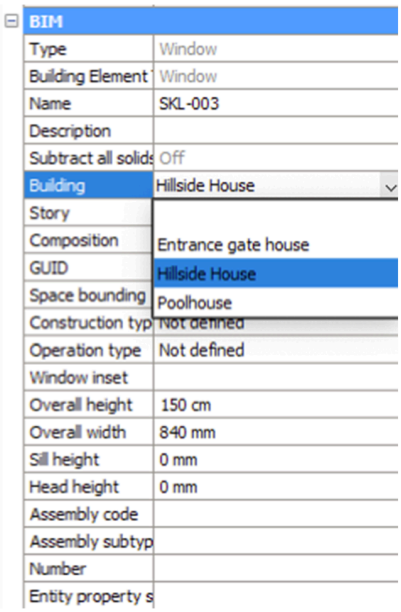
6 **(Optional)** Click the **Delete** icon ( ) to remove the selected building or story.

7 **(Optional)** Click **Import Spatial Locations** button ( ) to import a spatial location from a .txt file.

8 Select building element(s) in the drawing area to associate them with the appropriate spatial locations.

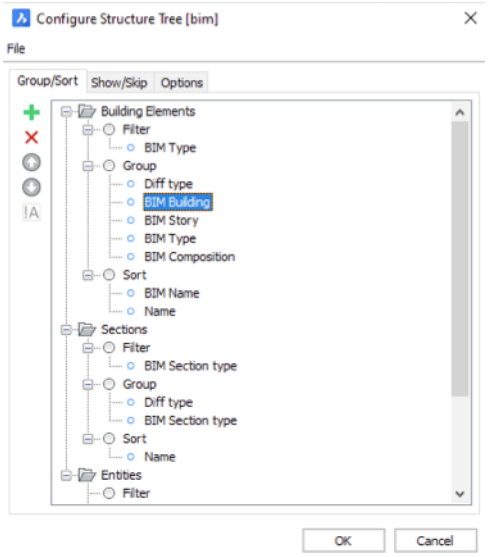
9 The properties of the selected building elements display in the **Properties** panel.

10 Do the following to assign the building and story to these building elements:
- Expand the **BIM** section in the **Properties** Panel.
- Click the **Building** field, then use the arrow to select a building.
- Click the **Story** field, then use the arrow to select a story.



20.4.17.4 Viewing the spatial locations in the structure browser

The **Structure browser** allows to view all of the entities in a drawing. By default, the structure of the building elements is organized in a particular form. Building elements are grouped by Building, Story, BIM type, and by composition respectively. This organization can be fully configured. Right click in the **structure browser** and click **Configure**, a new window will open. See the STRUCTUREPANEL command for more info.

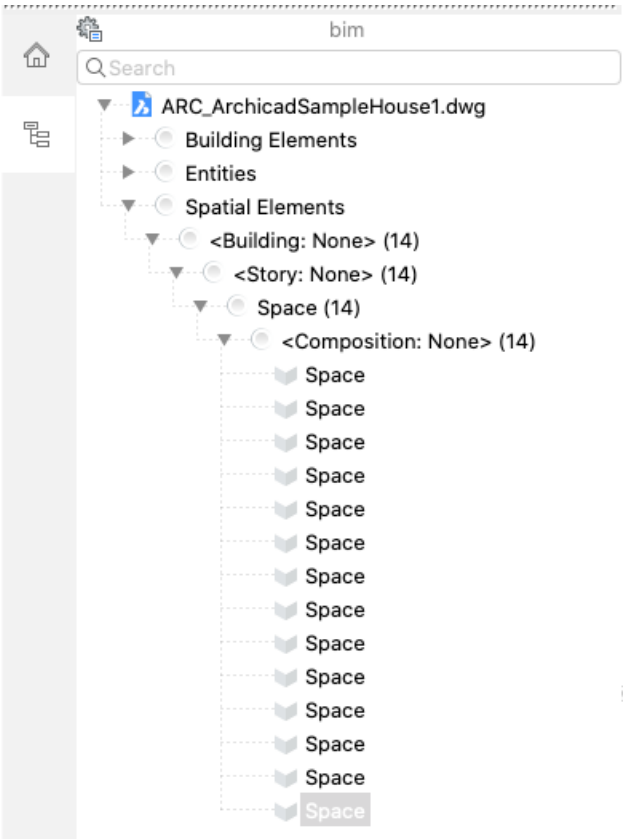


If there is no **BIM Building** or **BIM Story** in the **structure browser**, you can add them manually. See Structure Panel for more info.

20.4.17.5 About BIMATTACHSPATIALLOCATION

Assigning new Spaces can be done using the BIMSPACE command. Spaces are not always assigned to the according buildings because they don't have a spatial location yet. Use the BIMATTACHSPATIALLOCATION command to assign a spatial location to spaces.

For more information about this command, visit the Command Reference article BIMATTACHSPATIALLOCATION.



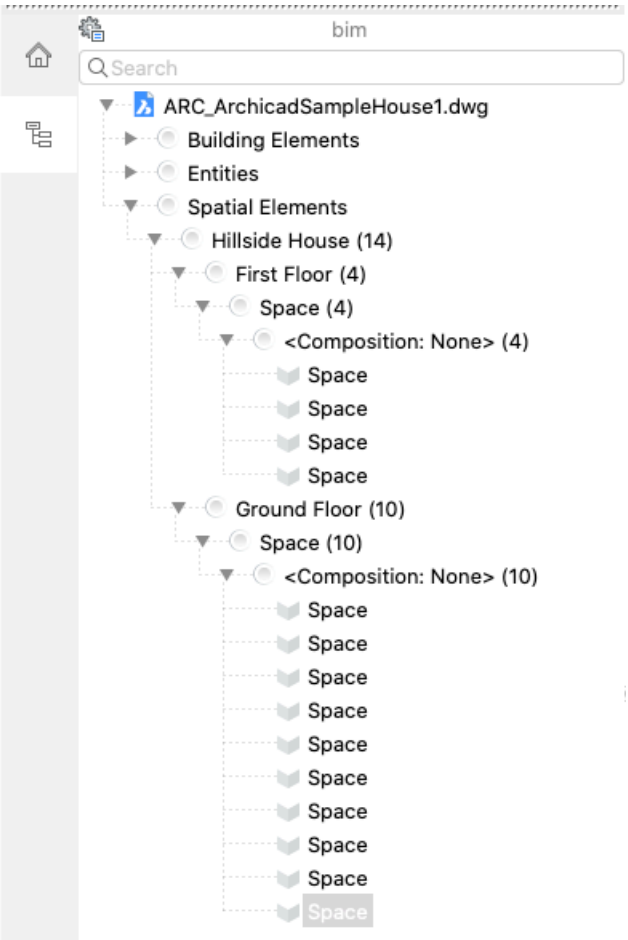
20.4.17.6 Procedure: attaching spatial locations

1 Launch the BIMATTACHSPATIALLOCATIONS command.

2 **You are prompted:***Enter location number or [Auto attach locations/Unattach current location]:* Press **A** to auto attach the spaces to their according building and floor level.

3 Select the spaces to update and press Enter.

4 **(Optional)** Press U to unattach the spatial location.



20.4.18 Structure browser

20.4.18.1 Commands

STRUCTUREPANEL

20.4.18.2 About

The **structure browser** is an interactive tree that displays the entities in the current model. Using the configurable structure tree on the **Structure Browser** the BIM model can be organized in a way that you want to view the elements. Once you configured the tree, you can easily save this structured tree as a .cst file. By default, the .cst files are stored in the **Support Folder**.

Place the cursor over a toolbar or ribbon panel, then right-click and choose **Structure** in the context menu.

Or

Type STRUCTUREPANEL in the Command line, then press Enter.

20.4.18.3 Procedure: using the structure browser

1 Open the **structure browser**.

By default, the **structure browser** displays on the left-hand side on your screen. With a **search** field at the top of the panel (1).

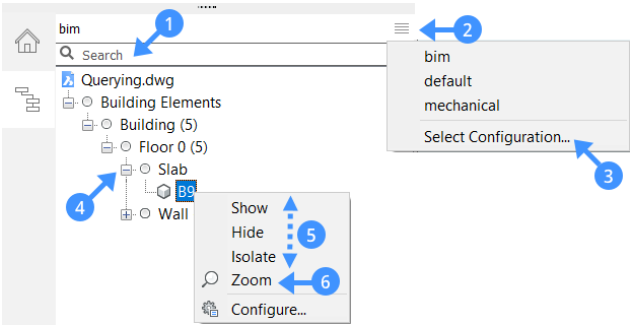
2 Click the horizontal stripes button (2) to load a configuration file in the list. Click **Select Configuration...** to select a .cst file, which specifies a configuration (3). The current configuration displays on the left. Based on the selected file the BIM model will be organized in a tree-structure.

3 Click the expand/collapse icon (4) of a node to expand or collapse the node. Right-click on a group name to select **Expand All** or **Collapse All** and expand or collapse all the subnodes at once.

You can **Show**, **Hide** or **Isolate** the entities, selected in the **structure browser**, in your model space, by right-clicking on the selected names (5).

This context menu also provides you to **Zoom** (6) into a specific entity in the model.

The following illustration shows the **Structure Browser**.



20.4.18.4 Procedure: querying in the structure browser

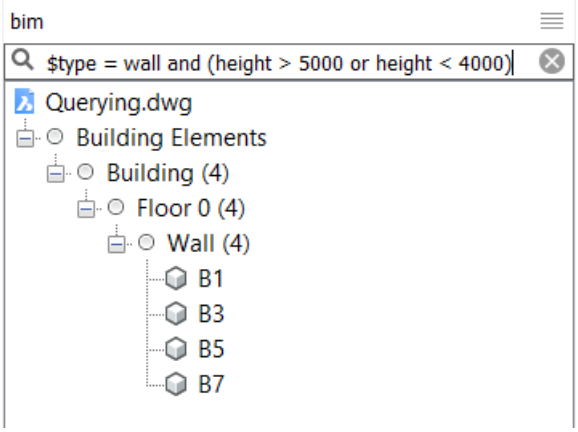
1 Typing something in the **Search** field at the top of the **structure browser** will issue a simple Text search: e.g. if you type 'wall' in the search field and press **Enter**, the **structure browser** will be filtered to everything with 'wall' in its name, including walls and curtain walls.

2 Text searches can be useful for a quick search, but more powerful queries are possible using expressions.

3 An expression starts with the **$** E.g. if you enter *$type = wall* and press **Enter,** the browser will be filtered on entities that have the BIM type = wall. Thus, using this expression, the curtain wall doesn't show up in the browser anymore.

4 Expressions can be more complex, such as filtering on multiple properties. E.g., you can enter *$type = wall and height > 5000* to filter on all walls with a height over 5000 mm.

5 Logical expressions can be nested using brackets: *$type = wall and (height > 5000 or height < 4000).*



20.4.18.5 Procedure: managing the tree-structure

The tree structure of the **Structure browser** represents the hierarchical set of rules in a graphical form. Each rule has a filter property, a grouping property, and a sorting property. Organizing the properties in the **Structure Browser** enhances the performance of your project while querying the entities among others. When you have a tree for your BIM model you can establish a hierarchy across its building element types, buildings, stories, etc. You can use the hierarchy to manage the properties at various levels of the data display.

Do the following to configure the tree-structure:

1 Right-click on the name of your drawing in the **Structure Browser**, then choose **Configure** in the context menu.

The **Configure Structure Tree** (7) dialog box displays.

2 Select any existing rule in the structure tree, then click the **Add icon** (8) or right-click and choose **Add Rule** in the displayed context menu. This menu can also be used to delete and rename the existing rule.

The new rule is added below the previously selected rule. Type a new name for the newly created rule by clicking its name field (9).

3 Select the **Group** (10) node of the rule, then use the Add icon or right-click and choose **Add Grouping Property** (11) in the context menu.

The **Select Property** dialog box displays. Select a property on the list that you want to use to group the properties in your **Rule** setting. Then, click the **OK** button. The new property is added to the **Group** node.

If you use the **Filter** node to add a property, the selected property will be added in this node.

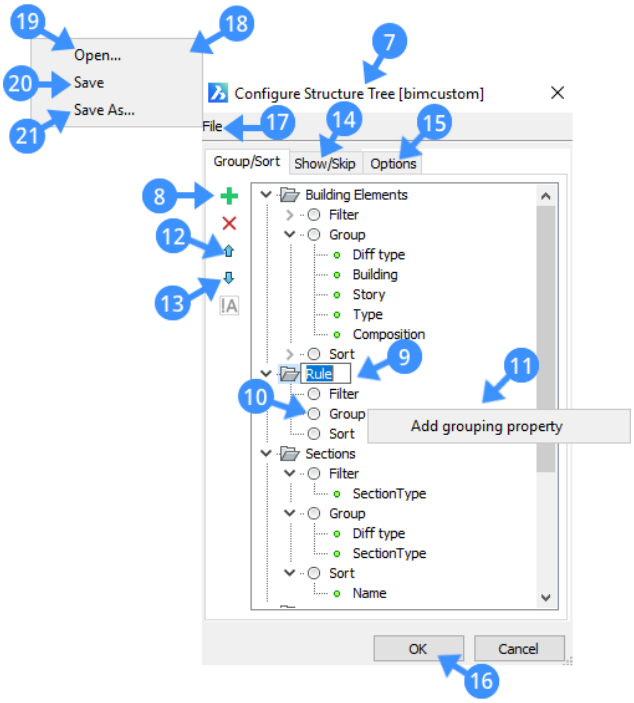
4 To move an item in the tree, select the item, then click the **Move Up** (12) or **Move Down** (13).

5 Select the **Show/Skip** (14) tab to sort the entity types or exclude them on the **Structure Browser**.

6 Select the **Options** (15) tab to set the entities action in the model using the structure tree. You can here specify whether or not selecting an entity in the **Structure Browser** also selects the entity in the model space.

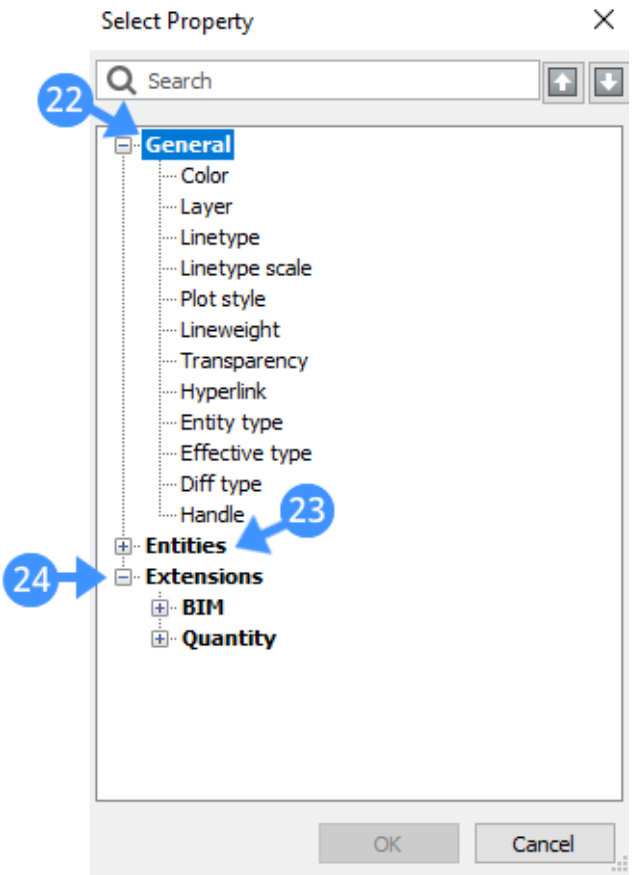
7 Do one of the following to save the configured tree:
- Click the **OK** (16) button.
- Click the **File** (17) menu in the dialog box and choose one of the available options on the list (18).

The following illustration shows the **Configure structure tree** dialog box.



Open (19)	Opens a file dialog to select another .cst configuration file.
Save (20)	Saves the current configuration and keep the file dialog open.
Save as (21)	Saves the current configuration under a different name and keep the file dialog open.

The following illustration shows the **Select Property** dialog box.



General (22)	Contains general properties.
Entities (23)	Contains various types of entity properties.
Extensions (24)	Contains various types of properties for **BIM** and **Quantity** extensions.

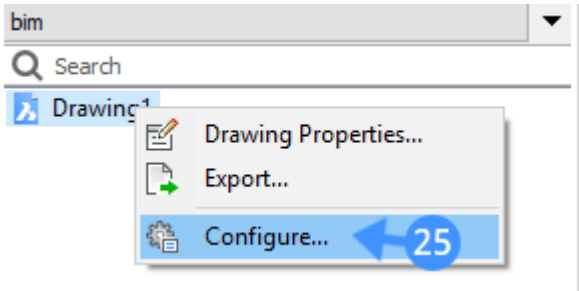
20.4.18.6 Procedure: managing the tree-structure with custom properties

In BricsCAD, the user-defined/custom properties are available to add in the **Select Property** dialog using the GUI. The **Select Property** dialog provides a list of dynamic properties for the drawing based on the current workspace. Note that custom properties, through the **Select Property** dialog box, can be used to group the tree structure.

Do the following to group the tree-structure using the existing custom properties:

1 Right-click on the name of your drawing in the **Structure Browser**, then choose **Configure** (25) in the

context menu.



The **Configure Structure Tree** dialog box displays.

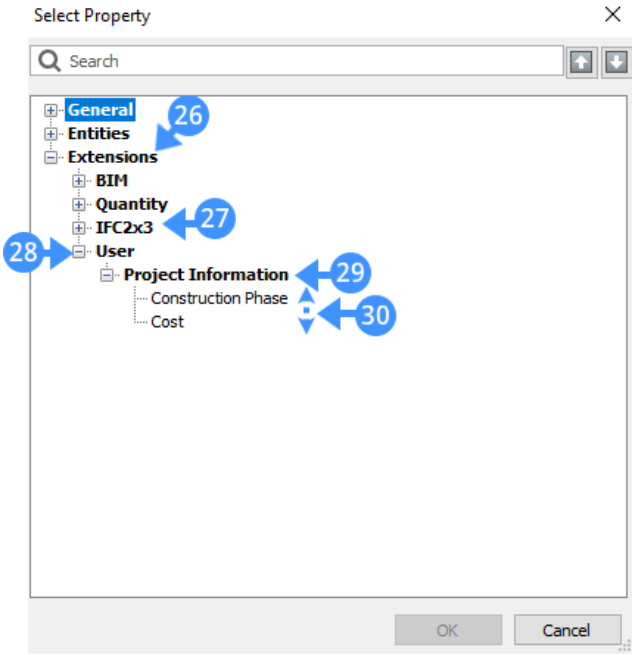
2 Select the **Group** node of the rule, and click the **Add rule or property**(+) icon.

Or right-click on the Group node and choose **Add Grouping Property** in the context menu. The **Select Property** dialog box displays (see below image for the **Select Property** dialog). If you have custom properties, these properties will be shown under **Extensions** (26) and grouped by its namespace.

3 Click the expand/collapse icon of a node to expand and collapse the node.

4 The following image shows the **IFC2X3** (27) and **User** (28) namespaces in the **Select Property** dialog.

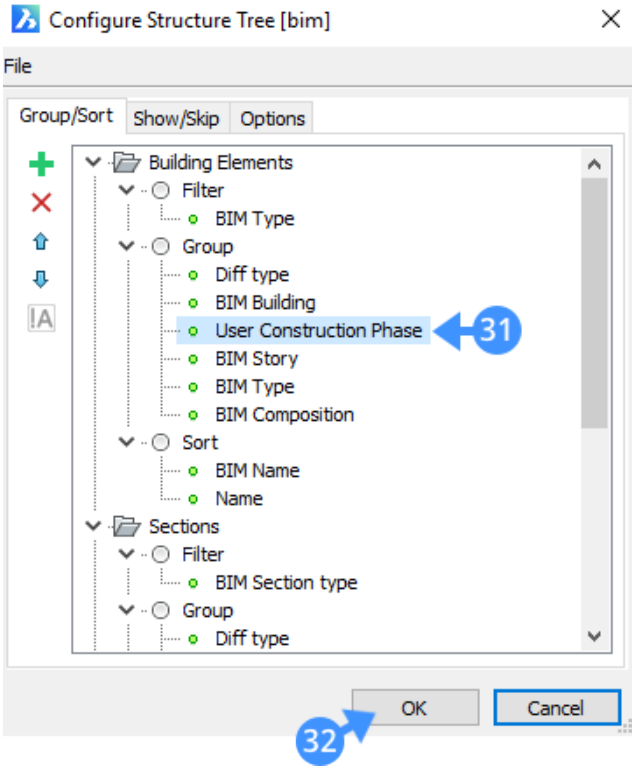
For example, the **User** namespace is expanded to display the property set (29) and properties (30) that are in this node.



5 Select the property to use for grouping in the tree structure, and press **OK**.

The property is added (31) to the **Group** node.

6 Press **OK** (32) to use the property for grouping entities in the **Structure Browser**.



20.5 Assistenza alla Progettazione

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

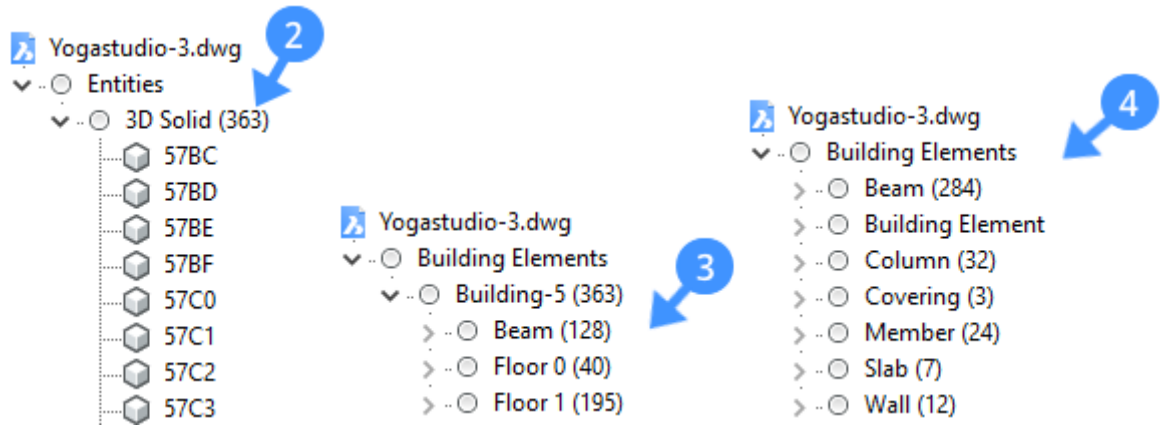
Choose BIMIFY from the Quad or type BIMIFY in the Command line and choose **Entire model**.

When the BIMIFY process is completed, an overview of the result (1) is provided in the **Prompt History** panel.

```
: BIMIFY

BIM data assigned to 3 of 13 object(s), skipped 10 pre-assigned object(s)
 Time taken to classify 0.17 s
Building 'Building':    <Existed>
The spatial location has been assigned to 3 element(s).
Skipped: 10 element(s).
Found  2 profile(s).
 Time taken for creation of hatches 0.07 s
 Time taken for creation of volumes 0.02 s
Successfully created 1 room.
 Time taken in total: 0.10 s
Determined outer walls
 Time taken in total: 0.03 s
Creating Sections
```

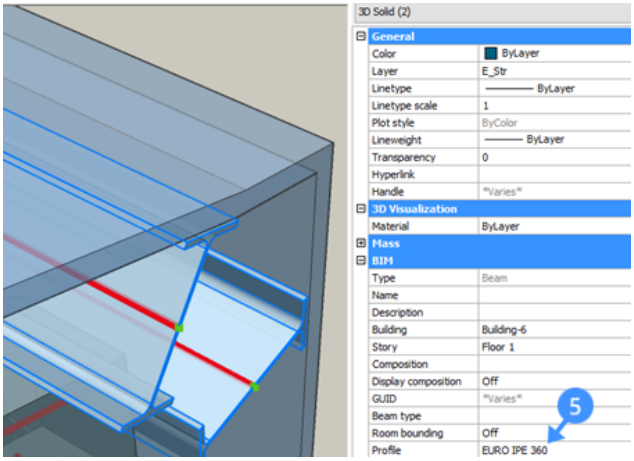
Bimify creates new spatial locations if necessary.



Left to right:

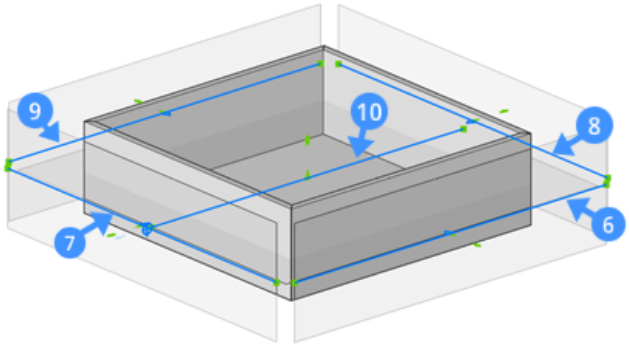
- ***Before Bimify:*** *The structure tree displayed only 3D solids (2)*
- ***After Bimify:*** *The entities are sorted into their corresponding building, story (3) and building element type (4).*

Nota: The **structure browser** can be customized to control the content of the tree.



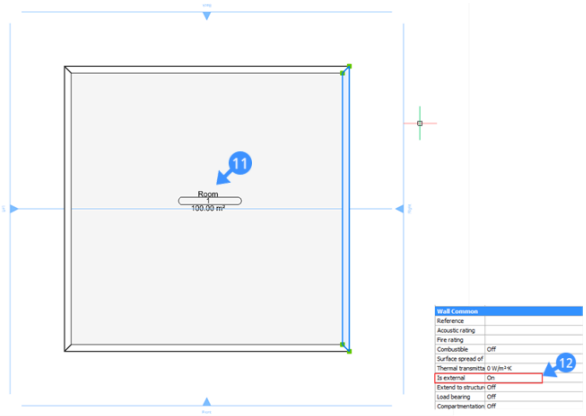
The profile data of a beam in the properties panel (5). BIMIFY recognized the profile of these model elements as structural members and detected the axis location of the profile.

The elevation views and floor plan sections are created last.



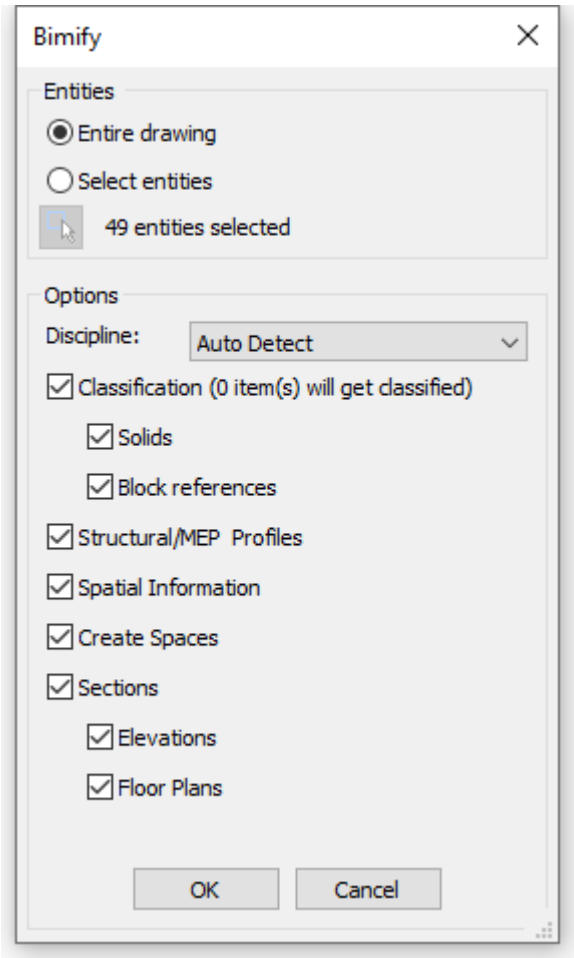
This building contains front (6), Left (7), Right (8) and Back (9) views as well as a floor plan (10).

For this simple building, one room and 4 external walls have been detected.



*The BIM room (11) and **Is External** (12) property under the **Wall Common** properties.*

If you choose to use the **Advanced** option of BIMIFY, a dialog box will appear:



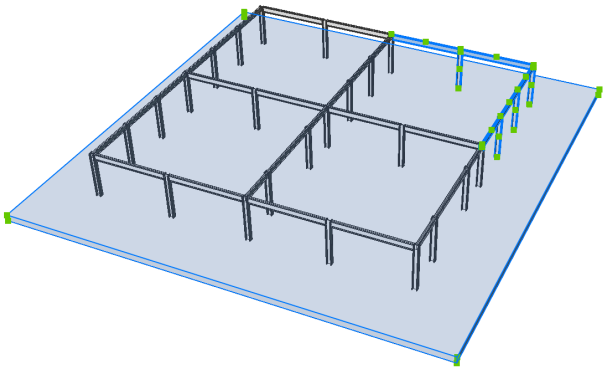
Discipline	Auto Detect Architectural Structural MEP Mixed model Defines the discipline of the drawing manually. This information helps improve the accuracy of Bimify.
Classification	Classifies **Solids** and **Block References** automatically.
Structural/MEP profiles	Assigns columns, beams, members or 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.

Nota: To disable the autoclassification option for solids or blocks simply uncheck the related checkbox. By default, all checkboxes are ticked. However, BricsCAD remembers your previous settings. Ticking the Spatial Information checkbox automatically includes the Classification option. This information is needed to detect the other elements.

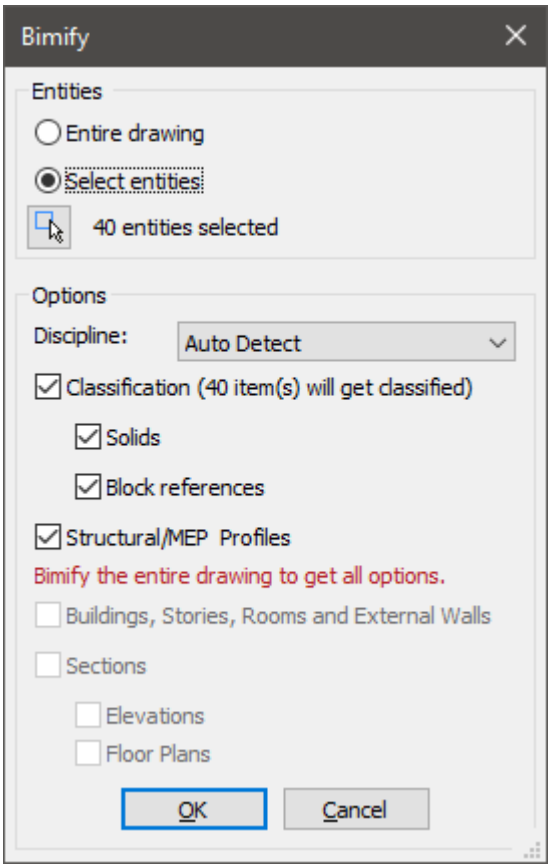
20.5.1.4 Procedure: bimify a selection of your BIM Model

1 Select the part of your drawing you want to Bimify.



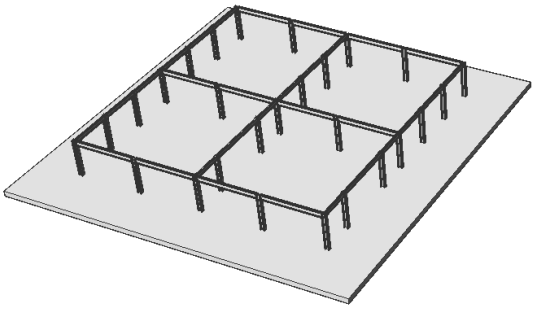
2 Type BIMIFY in the Command line and type in **A** for Advanced.

A dialog box displays:



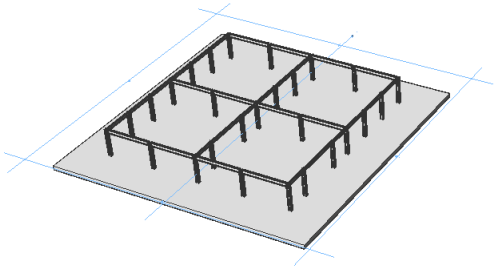
Nota: If you only select a part of your model, only solids, blocks and profiles can be classified. BricsCAD can only detect entities like stories, spaces, and sections when the whole drawing is selected.

3 When the Bimify process is completed, an overview of the result is provided in the **Prompt History** panel.



```
Model Layout1 Layout2 +
:
:
: BIMIFY
Entities in set: 16
BIM data assigned to 1 of 85 object(s), skipped 57 pre-assigned object(s)
 Time taken to classify 0.82 s
Found  9 profile(s).
:
```

BIMIFY classified the selected part of the model. No sections for floor plans or elevations are created.



```
Model Layout1 Layout2 +
Select entities <Drawing>:

BIM data assigned to 1 of 85 object(s), skipped 57 pre-assigned object(s)
 Time taken to classify 0.79 s
Building 'Building-2':    <Created>
The spatial location has been assigned to 85 element(s).
Found  57 profile(s).
Successfully created 0 rooms.
 Time taken in total: 0.00 s
Determined outer walls
 Time taken in total: 0.00 s
Creating Sections
```

This is the result if BIMIFY was launched on the whole drawing and all the boxes in the dialog box were checked. The entire model is classified additionally there are sections for the floor plan and the 4 elevations of the model.

20.5.2 Using propagate

20.5.2.1 Commands

BIMPROPAGATE

20.5.2.2 The five flavors

Before getting into the BIMPROPAGATE workflow it is important to note that there are five variants of BIMPROPAGATE, each working on a limited set of situations. These five variants (or flavors) were designed to guide new users through the process of BIMPROPAGATE. The tool has so many different uses that it can seem overwhelming to a new user; these variants help keep the tool more clear and concise, as they are designed for more specific use cases. However, they are still distillates from the original BIMPROPAGATE command, so every problem you can solve with a particular flavor you can **also** solve using the standard BIMPROPAGATE command. A more advanced user will even be able to solve problems that can not be solved by any of the particular flavors.

The BIMPROPAGATE variants are:

- BIMPROPAGATEPLANAR: Propagation of connections between planar elements such as walls, slabs and roofs.
- BIMPROPAGATELINEAR: Propagation of connections between linear elements such as beams, columns, pipes and ducts, and maximum one planar element.
- BIMPROPAGATEPATTERN: Propagation of a single element or group of elements on a flat surface to multiple locations and grids; can be used to propagate light fixtures, light switches, windows, air diffusers, columns on a grid.
- BIMPROPAGATEEDGES: Propagation along the edge of a planar solid; can be used to propagate railings, gutters, borders, wall caps, etc.
- BIMPROPAGATECORNER: Propagates details connected to multiple planar reference solids coming together in one node.

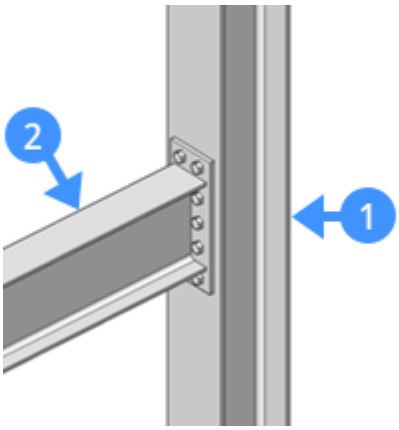
The general workflow of BIMPROPAGATE consists of two major steps: defining the detail volume, and choosing where to apply this detail.

20.5.2.3 Defining the detail volume

You are prompted:

Select reference solids. These will not be copied, but used to match other locations. [selection options (?)]:

The first step of every propagation is specifying the **reference solids**. These are the solids that define the situation where this particular detail should be applied. For instance, the image below shows a column-beam connection using an end plate and several bolts. This detail was modeled for this particular beam connecting with this particular column, and now we want to propagate this detail for every situation where a column and beam of these types come together. Thus, the reference solids in this case will be the column (1) and the beam (2). Later in the process, BIMPROPAGATE scans the model for situations where similar solids occur, and tries to map the detail onto these locations.



Nota: This is an example of a situation that can be solved by BIMPROPAGATELINEAR.

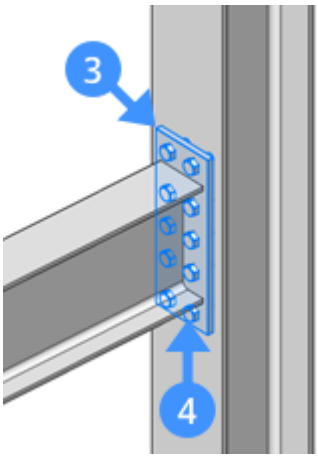
Press **Enter** to stop selecting reference solids.

The second step is specifying the detail objects or sub entities. These are the objects that you wish to propagate or copy throughout the model. They can be solids, block references or even faces (e.g. a hole in the reference solid can be propagated by selecting its faces).

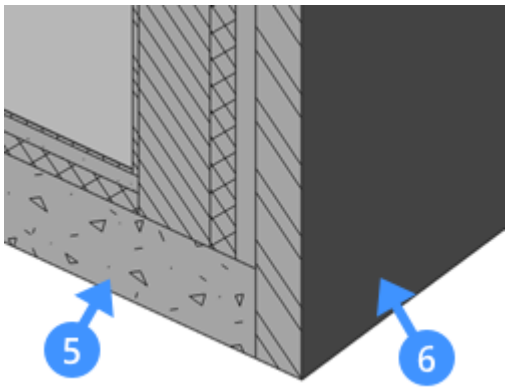
You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

In the example at hand the detail objects will be the end plate (3) and the bolts (4).



Nota: In some cases detail objects are not even required. Let's say we want to propagate the connection between the floor slab and the wall shown in the image below. Our reference solids are then the floor slab (5) and the wall (6), and no detail objects are required. In this case you can skip this step by pressing Enter or right click.



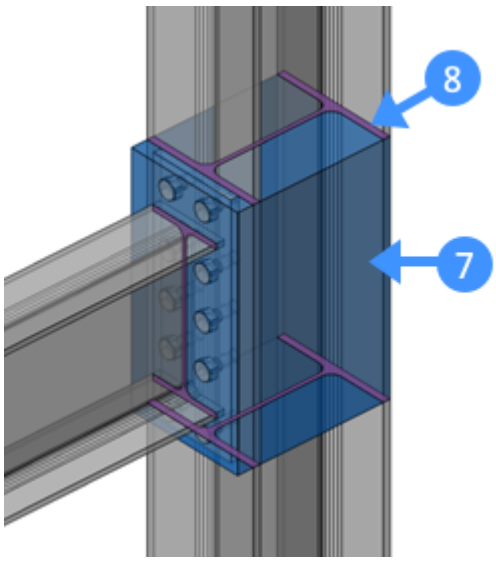
If we were to model extra solids in this connection (e.g. an insulation block or a skirting board), these extra solids would then be detail objects.

Nota: This is an example of a situation that can be solved by BIMPROPAGATEPLANAR.

Once the **reference solids** and the **detail objects** are specified, a detail volume is defined automatically. The visual style is temporarily set to X-ray, and the view is automatically zoomed in onto the detail volume. Everything that is inside this volume and is either a detail object or part of a reference solid will be copied to the other locations when executing BIMPROPAGATE.

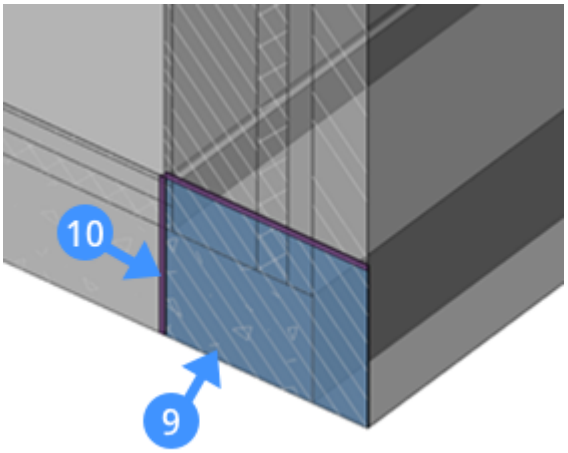
A detail volume can be either a **3D volume** or a **2D slice through a linear detail**.

In our example of the column-beam connection, the detail volume will be as shown in the image below. This is an example of a 3D volume. The connection of the reference solids to the detail volume is highlighted in purple (8).



Something interesting to note: In this particular detail, there are holes in the column where the bolts are bolted into the column. These holes are not necessarily detail objects, but since they are part of a reference solid and they are inside the detail volume, they will be copied as well. Thus, in this respect, BIMPROPAGATE does more than a 'dumb' copy of objects.

The detail volume of our wall-slab connection will look different. As shown in the image below, the detail (9) will be planar, and displayed in a section view. This is because the detail can be propagated as an extrusion of this 2D detail slice. Here as well, the locations where the reference solids connect to the detail volume are highlighted in purple (10).



Nota: The detail of BIMPROPAGATEPLANAR and BIMPROPAGATEEDGES will always be a 2D slice through a linear detail.

You are prompted:

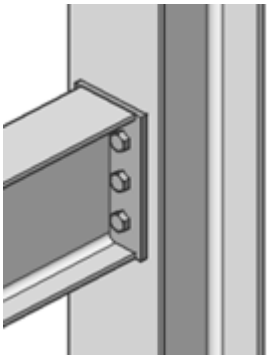
Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

You are prompted: Propagate detail as an extrusion of this 2D section? [Yes/No/Inflate first] <Yes>:

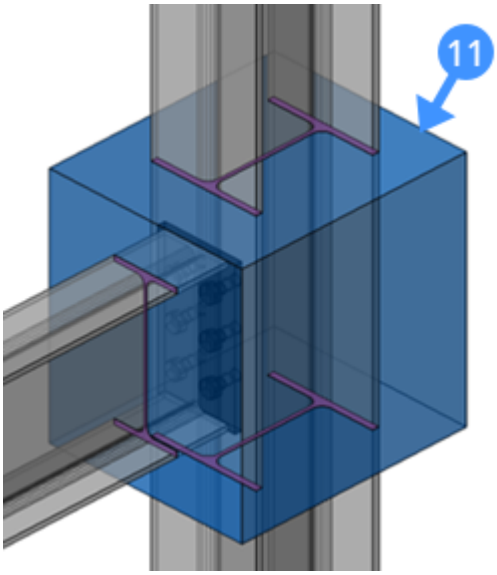
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Usually, the detail volume as it is proposed by BIMPropagate will be sufficient for what the user is trying to achieve. However, it is possible to inflate the detail to include a larger volume. This feature has two particular use cases:

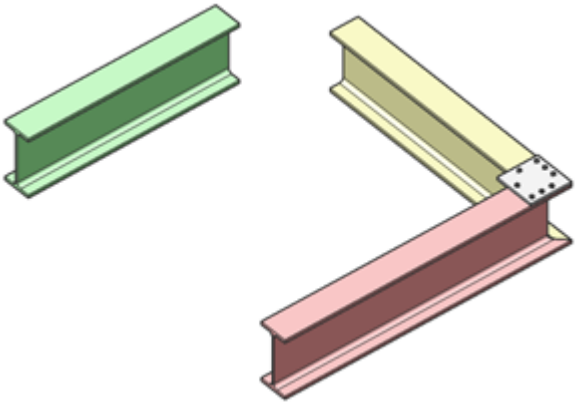
- If you want to replace an already existing detail with a new detail that is **smaller** than the original one. For example, we want to replace the previously proposed column-beam connection by the following:

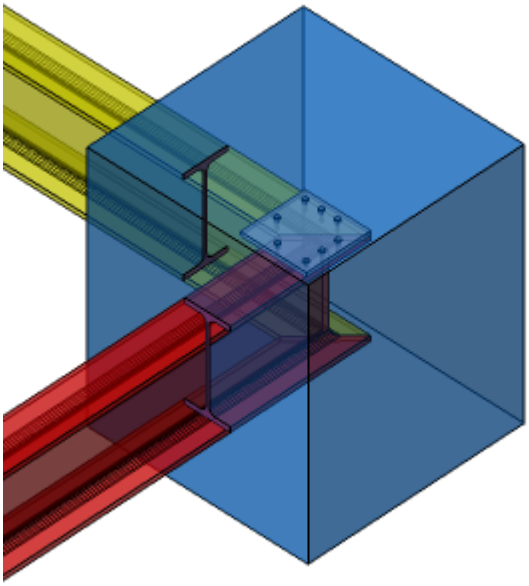
  In this case, we will have to inflate the detail volume (11) so that it is larger than the original detail:

- You might want to include a larger part of the reference solid, for example to connect with target solids that are further apart. In the image below, we want to propagate the connection between the **red** and the **yellow**beam, so that the **yellow** and the **green** beam have a similar connection.



To achieve this, we need to inflate the detail volume so that it overlaps the end faces of the red and the yellow beam.



Once the detail volume includes everything you want to propagate, you can go over to the second step: choosing where to apply the detail. For 3D detail volumes there is one last choice to make: Propagate the detail as a **Block** or as a **Copy**. If you choose to propagate the detail as a block, then it will group all detail objects into a new **Block Definition** and copy around references to this **Block Definition**.

20.5.2.4 Choosing where to apply the detail

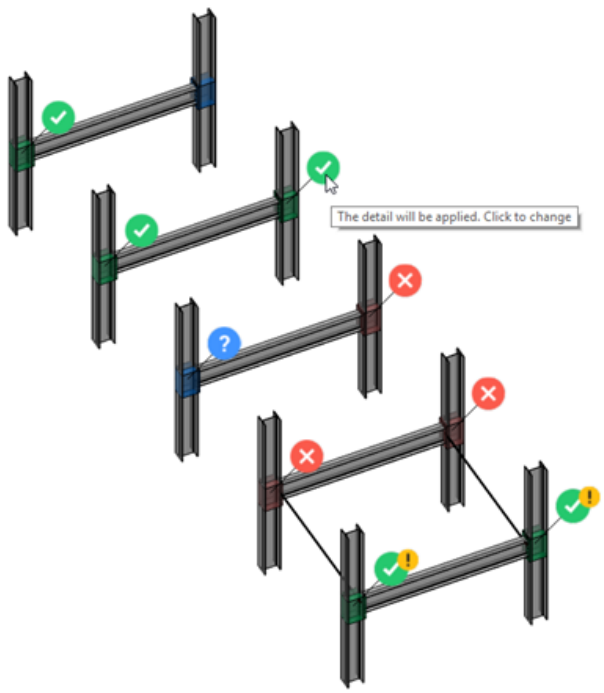
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

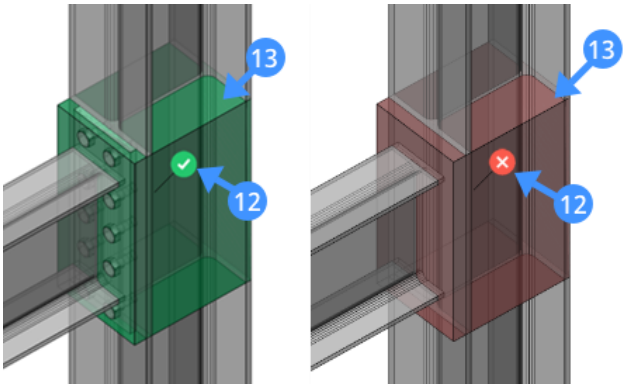
In this step the view zooms out showing all the possible suggestions on screen. Each suggestion is a location where this detail could be applied, and is indicated by one of the following icons (or widgets):

	The detail will be applied.
	The detail will not be applied; it was manually toggled off by the user.
	The detail will not be applied, because of one of several possible reasons. Hovering over the icon will provide more information.
	The detail will be applied, but interferences were detected.
	The detail will be applied, but there are several alternatives. Hovering over the icon will allow you to cycle between alternatives.

Hovering over a widget reveals some more information. Clicking a widget changes its status: you can toggle suggestions individually by clicking their respective widgets. Pressing Enter will then apply all the widgets as they are shown at that point: suggestions with a green checkmark will be applied, others will not be applied.



A suggestion also shows a preview of what the detail will look like. In the images below, you can both see the widgets (12) and the previews (13).



Widgets can also be **selected** using a window selection. A selected widget is highlighted by a blue border. Once one or more widgets are selected, pressing Enter will no longer apply **all** the suggestions, but **only those that are currently selected:**

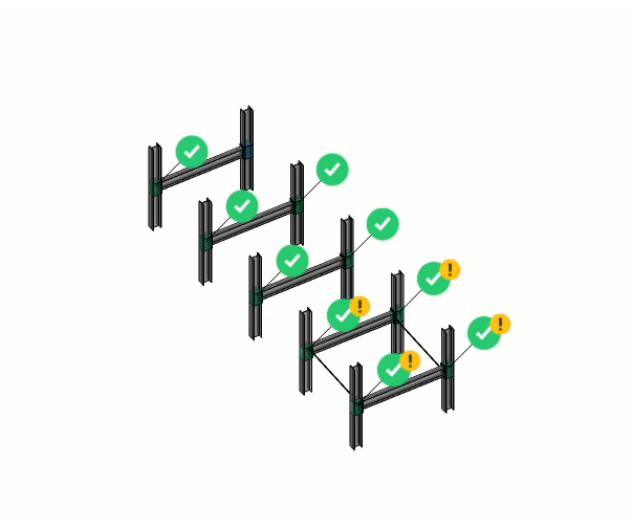
You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:

This procedure can be used to gradually solve all the suggestions.

In the video below, the following steps are taken:

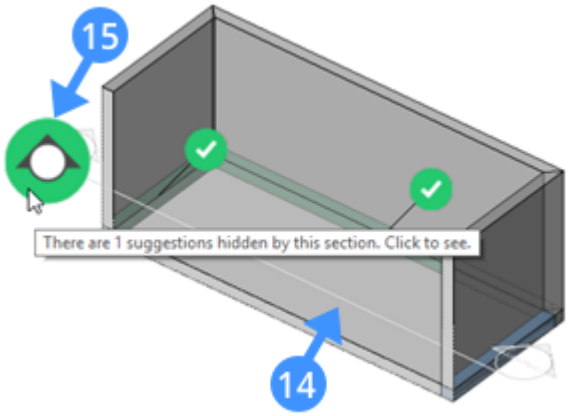
- Four widgets are selected and toggled off
- Enter is pressed: the four selected suggestions were all toggled off, so they are not applied, and the four suggestion are removed from the view
- Four more suggestions are selected
- Enter is pressed: the four selected suggestions are applied, and removed from the view
- There is one suggestion left; Enter is pressed to accept this suggestion, and the BIMPROPAGATE command is terminated.



It is important to emphasize that BIMPROPAGATE only processes entities that are currently visible. As a result, hidden entities, or entities on a hidden or frozen layer are not included when BIMPROPAGATE is scanning the model for locations where the detail can be applied. This can be a useful when executing BIMPROPAGATE in a large model; e.g. when you want to propagate a steel connection, only turn on those layers that contain steel members. This will improve the propagation performance.

It is possible to activate or move existing section planes during BIMPROPAGATE.

- A section plane (14) can be activated by double-clicking it.
- Left-click it once to move it.
- When a section plane is activated, a widget (15) might appear next to it indicating that there is one or more suggestion currently hidden because of the active section. Click the widget to deactivate this section and show the hidden suggestions.



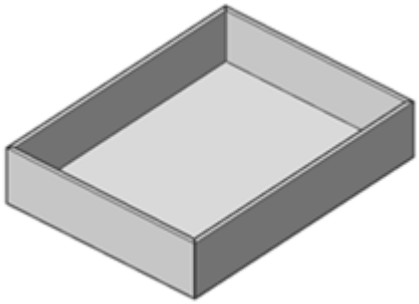
20.5.3 Propagate planar

20.5.3.1 Commands

BIMPROPAGATEPLANAR

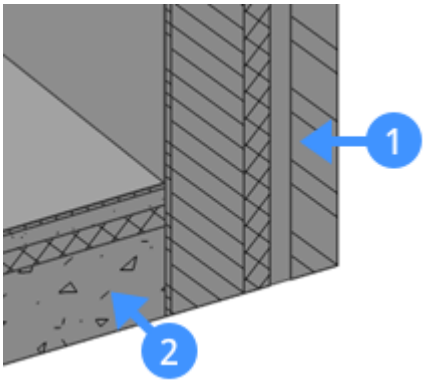
20.5.3.2 Procedure: propagating a wall-slab connection

1 Consider the following situation of four walls and one slab, created with **QuickDraw**.

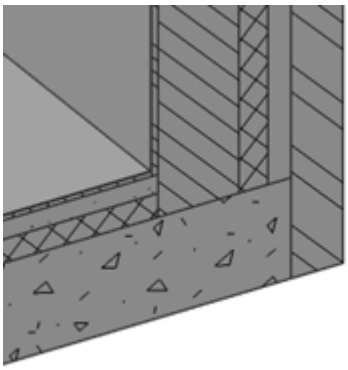


2 The four walls were given a composition, as well as the floor slab. Without any further editing or modeling, creating a vertical section through the building will show the connection one of the walls (1)

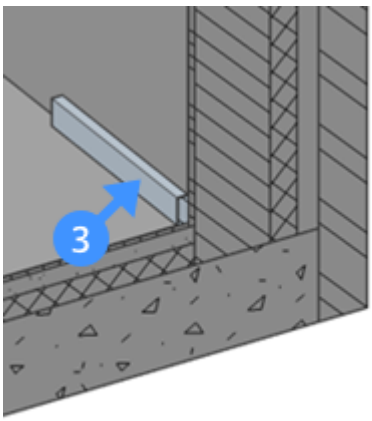
and the slab (2) to be something similar to the following image.



3 While this section is active, we can **Push/Pull** the different ply faces to obtain the desired connection between the wall and the slab:



4 We can detail this connection even further, for example by locally adding a skirting board (3). This was done by simply drawing a box and classifying this solid as a **Building Element**.



5 Now that we have this detailed connection, we want it to be applied to every location where a wall and slab of this type come together. Let's deactivate the section and launch BIMPROPAGATEPLANAR.

6 **You are prompted:**

Select planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

Select the wall and slab, and press Enter.

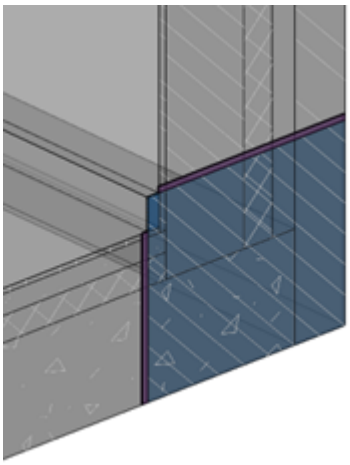
7 **You are prompted:**

Optional. Select detail objects to be copied as part of the connection. [selection options (?)]:

Select the skirting board solid, and press Enter.

8 **You are prompted:**The detail section varies too much along the extrusion direction.

In this particular case, BIMPROPAGATEPLANAR cannot find a solution: because the skirting board is not modeled along the entire length of the wall, BIMPROPAGATEPLANAR finds contradicting detail volumes. This problem can be solved easily by activating a section where the skirting board is present (e.g. the section we used earlier to create this detail) and trying again. If we now follow the same steps as before, BIMPROPAGATE finds a detail volume as shown in the image below.



9 **You are prompted:**

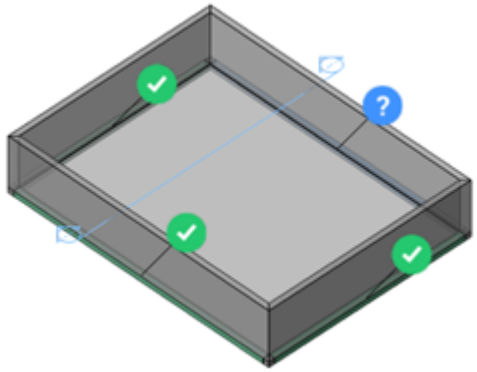
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

10 **BimPropagatePlanar** now scans the entire drawing for similar locations. When this is done, all sections are deactivated and the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

You are prompted:

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green checkmark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information.

In this case, it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the skirting board only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green checkmark.

11 Press **Enter** to complete the command.

Now, all four walls should have this particular connection with the floor slab, including the skirting board.

20.5.4 Propagate linear

20.5.4.1 Commands

BIMPROPAGATELINEAR

20.5.4.2 About

BIMPROPAGATELINEAR is one of the five Propagate flavors. It can be used to copy detailed connections between two or more linear solids. Common examples are column-beam connections or column-slab connections.

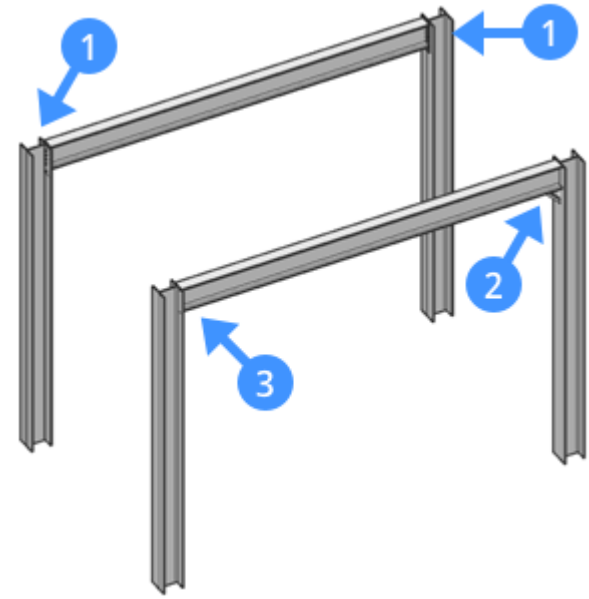
For a more in-depth explanation of the general principles of Propagate, read the Using Propagate article.

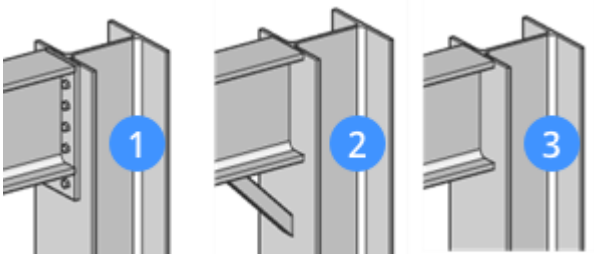
For more information about this command, visit the Command Reference article BIMPROPAGATELINEAR.

20.5.4.3 Procedure: propagating a column-beam connection

1 Consider the following situation of four columns and two beams. Two of the connections have been designed with a plate and bolts through the column flanges (1). One connection has been designed with a diagonal member for additional rotational stiffness (2). The last connection has not been

designed yet (3).





2 We can now use Propagate Linear to copy one of the details to the other locations. In this example we will copy connection 2 to all four locations.

3 We will launch BIMPROPAGATELINEAR.

4 **You are prompted:**

Select the linear or planar reference solids that form the connection. The connection detail will be copied to similar solids. [selection options (?)]:

At least one should be linear.

Select the column and the beam that are connected by the diagonal member, and press **Enter.**

5 **You are prompted:**

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the connection. [selection options (?)]:

Select the diagonal member and press **Enter**. A detail volume is created that completely encompasses the diagonal member and parts of the column and beam.

6 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain

Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

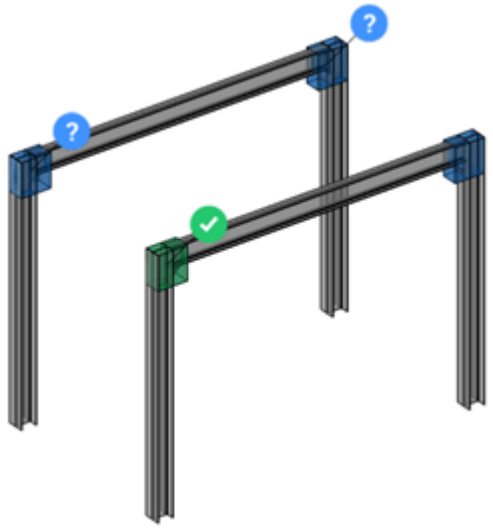
You can choose to propagate this detail as a block to keep the file smaller and more manageable, or as a copy. Inflating the detail is not necessary in this case. For more information on inflating, read the Using Propagate article. Choosing **No** will exit the command.

Press **Enter** to accept.

7 BIMPROPAGATELINEAR now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

8 **You are prompted:**

Accept or reject suggestions by clicking on the tick. Then [Apply/Cancel] <Apply>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Existing details detected. Suggestion will not be applied. Click to change**. We want to replace the bolted connection by this diagonal member, so we click both the blue widgets so they turn into green check marks.

9 If we now press **Enter**, all connections should be replaced by the diagonal member.

20.5.5 Propagate pattern

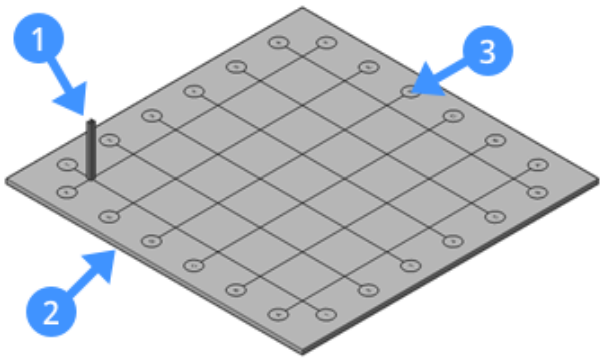
20.5.5.1 Commands

BIMPROPAGATEPATTERN

20.5.5.2 Procedure: propagating columns on top of a slab

1 Consider the following situation: a column (1) is placed on top of a slab (2). There is also a grid drawn

on top of this slab (3).



2 We want to place this column on specific axis intersections. We could manually copy the column but, Propagate allows us to do this faster.

3 Let's launch BIMPROPAGATEPATTERN.

4 **You are prompted:**

Select planar reference solid to which the detail is related. The detail will be copied to similar solids. [selection options (?)]:

Select the slab, and press **Enter.**

5 **You are prompted:**

Select detail objects (solids, block references, faces, edges,...) to be copied. [selection options (?)]:

Select the column, and press **Enter.**

6 A 3D detail volume is created.

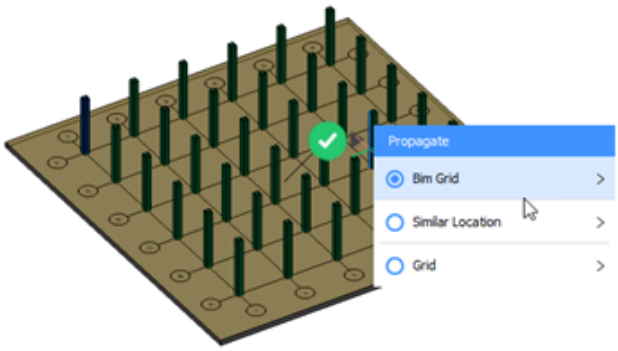
7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [yes, copy as Block/yes, plain Copy/No/Inflate first/Save detail/save detail to Library] <yes, copy as Block>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command.

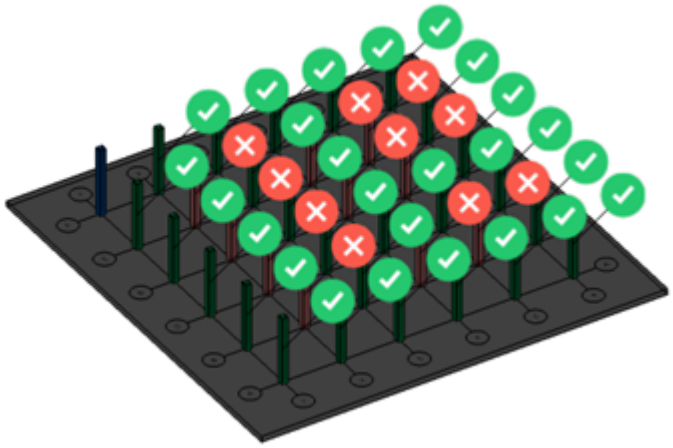
Press **Enter** to accept.

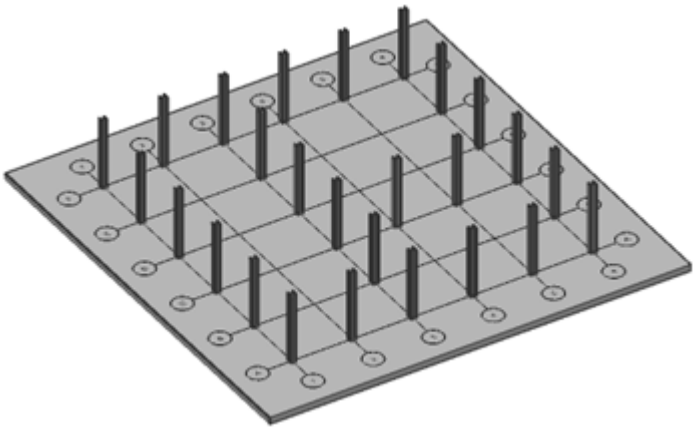
8 A widget appears. Hovering over this widget reveals a flyout menu with several options:



- **Bim Grid**: this is the grid that was drawn on top of the slab. Choosing this option and pressing Enter will copy this column to every axis intersection of this grid.
- **Similar Location**: this will copy the column to similar locations on this slab; in this case the four corners of the slab.
- **Grid**: this allows you to create your own grid of columns on top of this slab. You can choose to add or remove rows and columns. These rows and columns will always be evenly spaced.

9 We'll choose the BIM **Grid** option. Clicking this will expand the menu further; we now have the option to **Explode**. This will create a separate widget for each instance of this column, giving you more control over which suggestions you wish to apply (in this case which axis intersection you wish to place a column). We can now manually turn off suggestions:





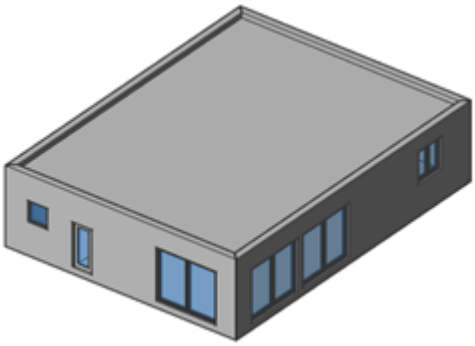
20.5.6 Propagate edges

20.5.6.1 Commands

BIMPROPAGATEEDGES

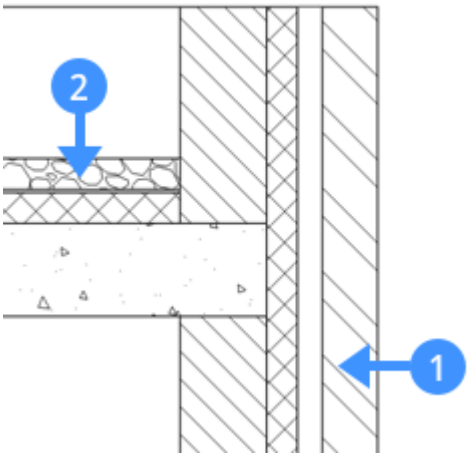
20.5.6.2 Procedure: propagating a wall cap

1 Consider the following situation of a one-story building with a flat roof.

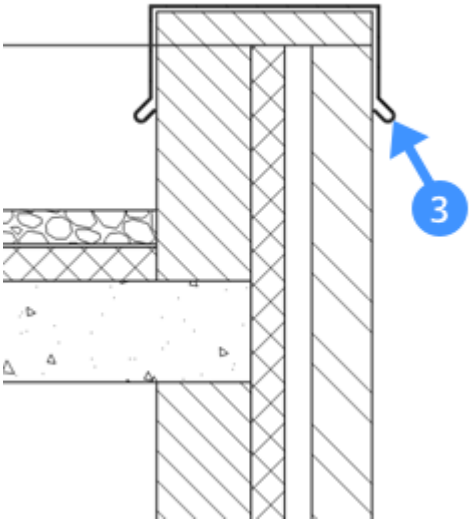


2 Let's create a vertical section plane to cut through our building and take a look at where the wall (1)

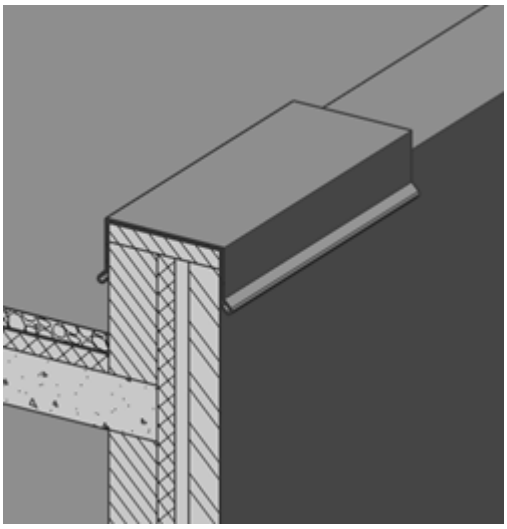
and the roof slab (2) come together.



3 We want to create a capping (3) on top of this wall to get the following result:



We can start by creating this locally on top of one wall, either by using direct modeling tools or inserting a block from somewhere else and exploding it. In our section it might look like this:



4 ow we can use Propagate to copy this detail over to all applicable locations. Let's deactivate the section and launch BIMPROPAGATEEDGES.

5 **You are prompted:**

Select planar reference solid to which the edge detail is related. The detail will be copied to edges of similar solids. [selection options (?)]:

Select the wall on top of which the capping was modeled, and press **Enter.**

6 **You are prompted:**

Select detail objects to be copied. These should be solids or faces that are parallel to an edge of the reference solid. [selection options (?)]:

Select the objects that make up the capping, and press **Enter.**

7 **You are prompted:**

Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

Inflating the detail is not necessary in this case. For more information on inflating, read the article on Propagate. Choosing **No** will exit the command. Press **Enter** to accept.

8 **You are prompted:**

Propagated to edges with similar orientation only? [Yes/No] <Yes>:

Choosing **Yes** will only apply this detail to edges with a similar orientation, in this case wall edges that point upwards. Press **Enter** to accept.

9 **You are prompted:**

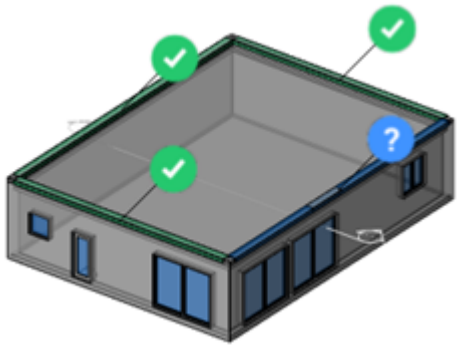
Choose location of detail relative to edge: [Outside reference solid/Inside reference solid] <Outside reference solid>:

This option controls whether the detail will be placed inside or outside (on top of) the base solids. When placed inside, the detail is subtracted from the base solids.

10 BIMPROPAGATEEDGES now scans the entire drawing for similar locations. When this is done, the

view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed:

11 **You are prompted:**Choose next action [Apply all/Cancel] <Apply all>:



A green check mark means that this suggestion will be applied when pressing Enter. A question mark means that it will not be applied due to one of several possible reasons. Hovering over the widget will reveal more information. In this case it says **Location of original detail. Click to replace original detail by adapted extrusion**. Because we modeled the wall cap only locally, we should replace the original detail by an adapted extrusion so that it covers the entire length of the wall. Clicking the widget will turn it into a green check mark.

12 Press **Enter** to complete the command.

Now, all four walls have this wall cap and the corner connections are created automatically.

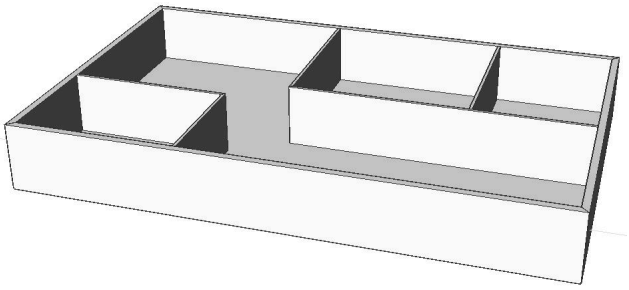
20.5.7 Propagate corner

20.5.7.1 Commands

BIMPROPAGATECORNER

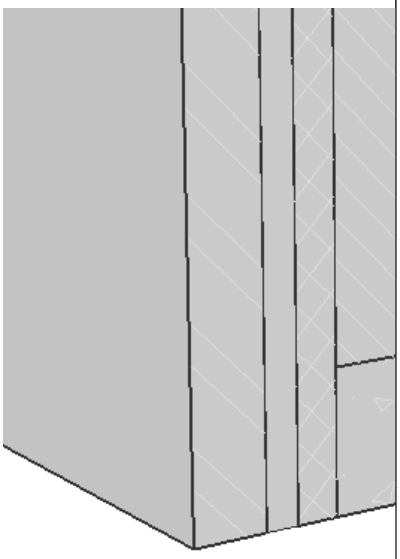
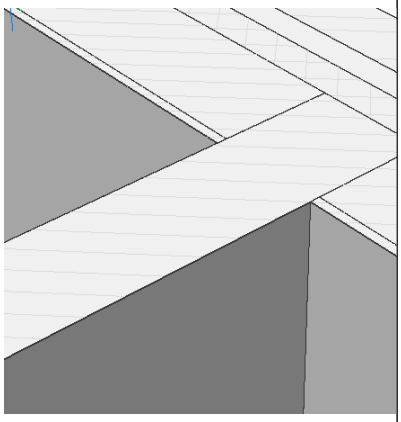
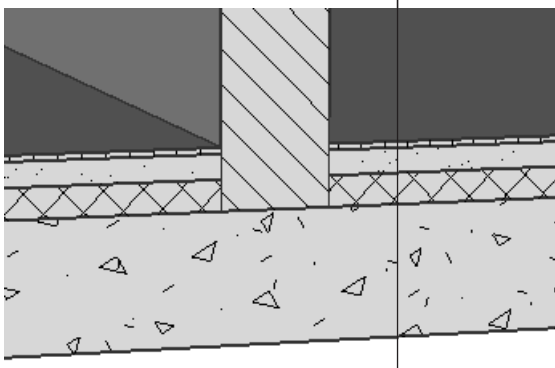
20.5.7.2 Setting the foundations

In this example the 4 exterior walls, 5 interior walls, and floor slab have already been assigned a composition.

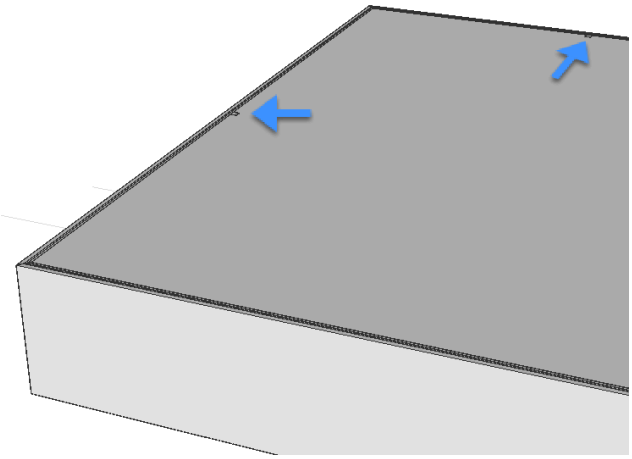


First, you must connect the 3 planar types of connections separately. For a detailed explanation of how to do this, read the article Propagate planar.

You should now have the following connections:

		
Exterior wall - Floor slab	Exterior wall - Interior wall	Interior wall - Floor slab

The nodes where the exterior wall, interior wall, and floor slab come together can't be correctly propagated using BIMPROPAGATEPLANAR. Instead, use BIMPROPAGATECORNER. BIMPROPAGATECORNER can propagate details connected to three planar base solids (i.e. corners).



20.5.7.3 Propagate corner

1 Launch BIMPROPAGATECORNER.

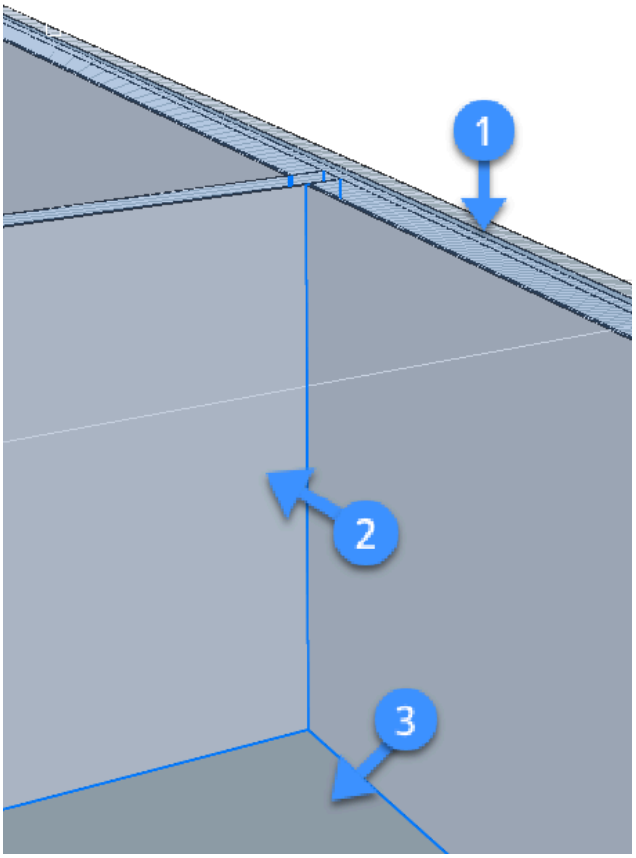
1 **You are prompted:**

Select at least three planar reference solids that form a 3D corner. The detail will be copied to similar

corners. [selection options (?)]:

2 Select the exterior wall (1), the interior wall (2), and the floor slab (3).

3 Press **Enter** to accept.



You are prompted:

Optional. Select detail objects (solids, block references, faces, edges,...) to be copied as part of the corner detail. [selection options (?)]:

If you have extra solids in your detail, i.e. an insulation block, you can select this as an extra detail object and it will be copied to the other nodes when using BIMPROPAGATECORNER.

4 Press **Enter** when ready.

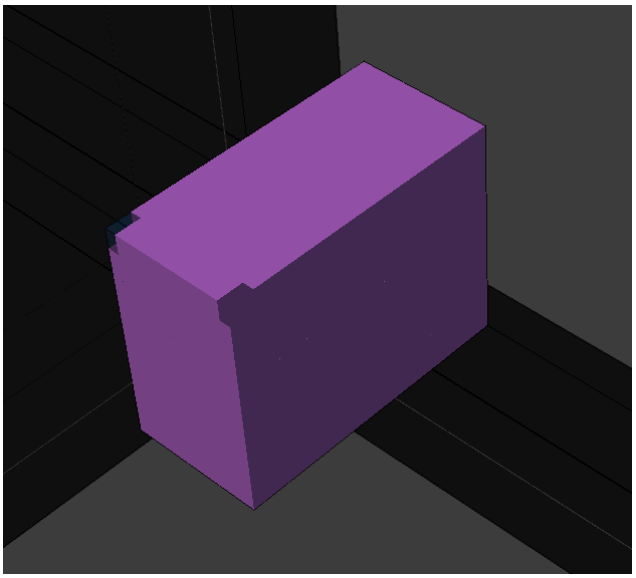
Nota: If you don't have any extra details, just press **Enter** to continue.

5 The volume that will be propagated is encased in a purple box.

6 **You are prompted:**

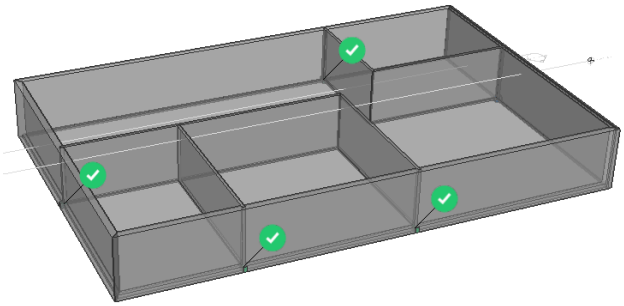
Blue: detail area to be copied. Purple: reference contact area. Accept? [Yes/No/Inflate first/Save detail/save detail to Library] <Yes>:

7 Press **Enter** to accept the detail.



Nota: BIMPROPAGATECORNER shows a 3D volume instead of a 2D section.

8 BIMPROPAGATECORNER now scans the entire drawing for similar locations. When this is done, the view is zoomed out to show all possible locations. Every suggestion is highlighted and a widget is displayed. A green checkmark means that this suggestion will be applied when pressing **Enter**.



You are prompted: Choose next action [Apply all/Cancel] <Apply all>:

9 Press **Enter** to complete the command.

The nodes where the exterior wall, interior wall, and floor slab come together should now be propagated correctly.

20.5.8 Parametrize details

20.5.8.1 Commands

BIMPARAMETRIZEDETAIL

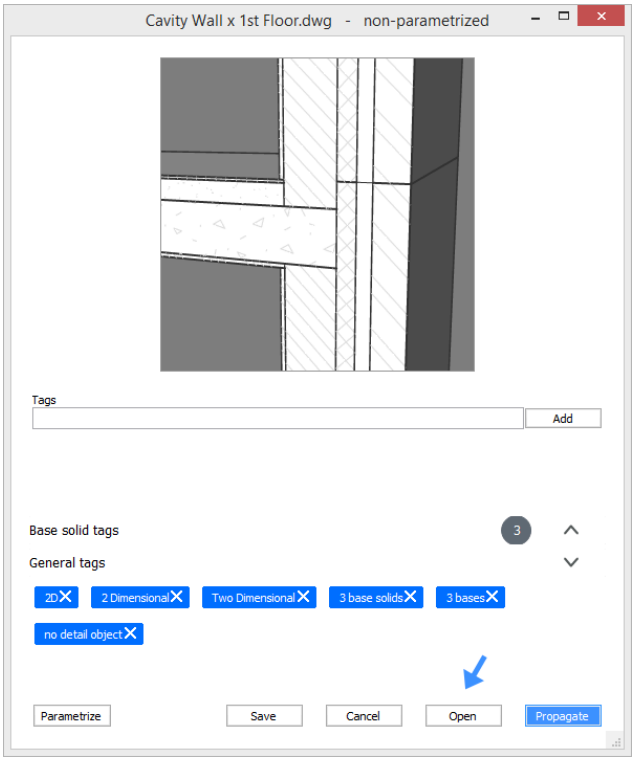
20.5.8.2 About BIMPARAMETRIZEDETAIL

Use the BIMPARAMETRIZEDETAIL command to automatically generate parameters for a saved detail. Standard parameters such as angles, offsets in different directions and tolerance parameters are created. These parameters mean you can propagate details to target situations similar to the example below.

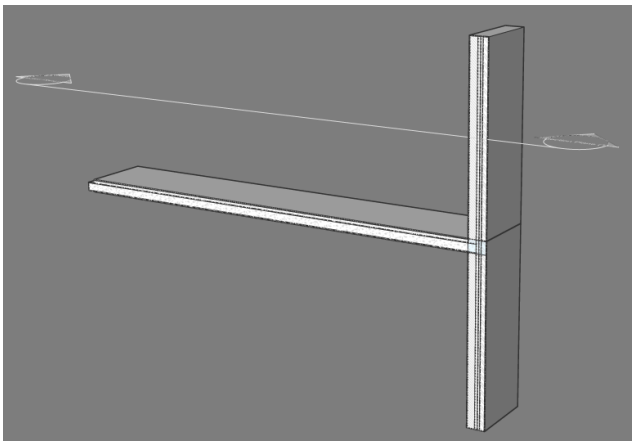
The automatically generated parameters form a framework that can control the position and shape of the reference solids. You can extend this framework adding custom constraints. You can evaluate the effect of these changes by animating the parameters from the **Mechanical Browser**. You can add custom constraints to link detail objects to the reference solids and to each other. You can also use the framework parameters in the expression of those constraints.

20.5.8.3 Parametrize and edit a wall-slab connection detail

1 To open the .dwg file of the detail you want to parametrize click on the detail in the **Details** panel. The **Detail** dialog box displays:



2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL. The detail .dwg file will open and the detail will be parametrized.



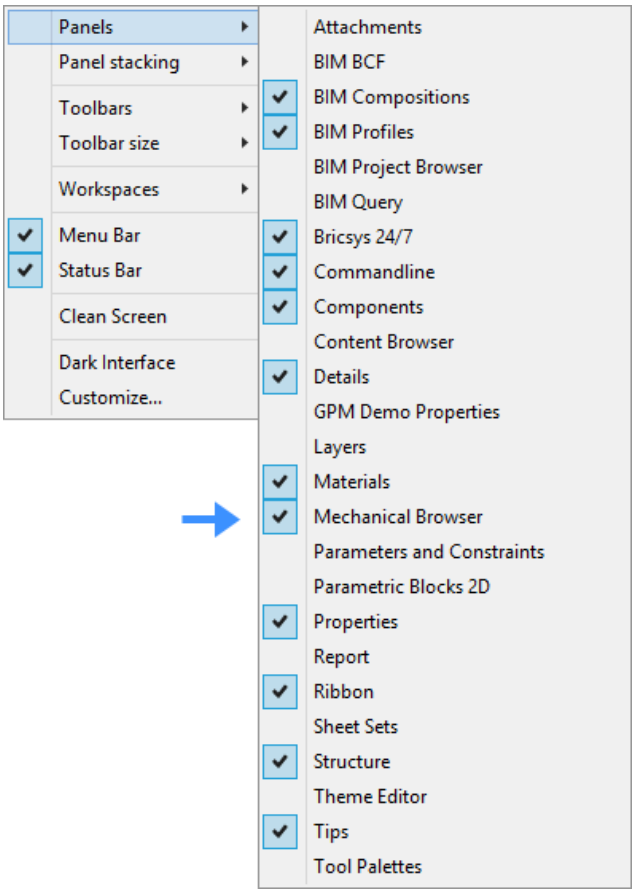
3 Open the **Mechanical Browser**.

4 Click the **Mechanical Browser** button on the right of the screen.

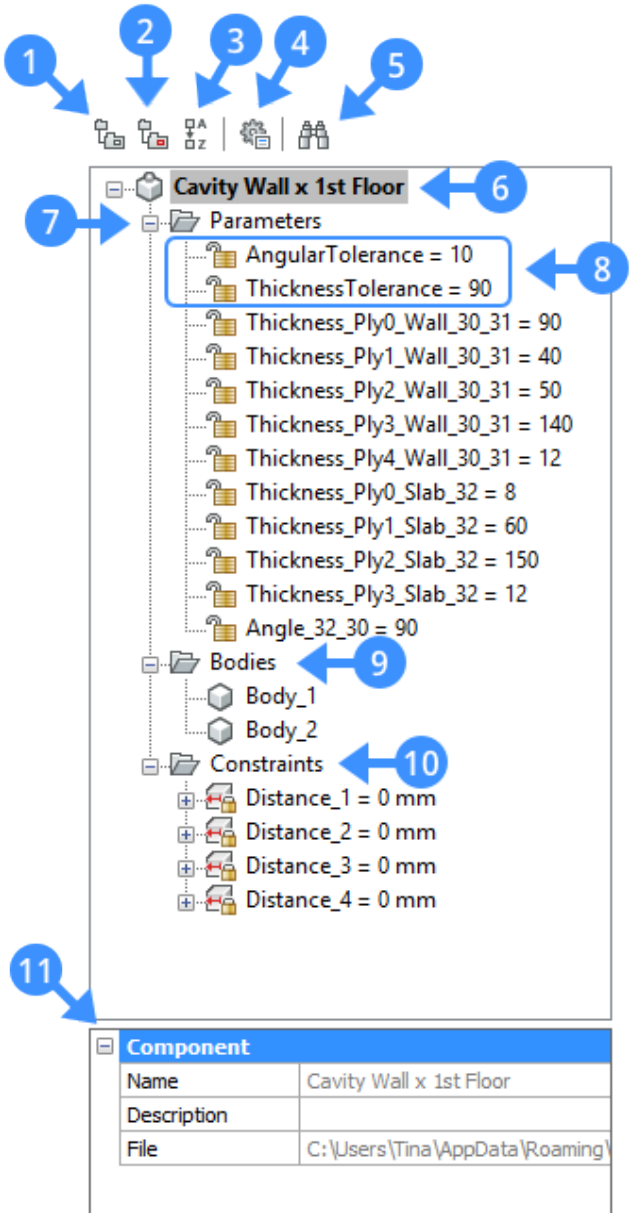
5 If the **Mechanical Browser** button isn't visible yet, move the cursor to the toolbar or ribbon panel and right-click. A context menu displays.

Expand the **Panels** option.

Select **Mechanical Browser** in the context menu.



6 Once you have clicked on the **Mechanical Browser** button, the **Mechanical Browser** displays:



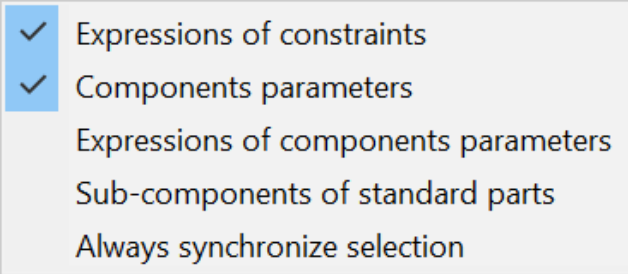
Nota: You can also use the MECHANICALBROWSEROPEN command.

(1) **Group by entity**: Group the parameters and constrains by entity.

(2) **Group by type**: Group the parameters and constrains by type.

(3) **Sort**: Sort alphabetically.

(4) **Settings**: An options menu will display:

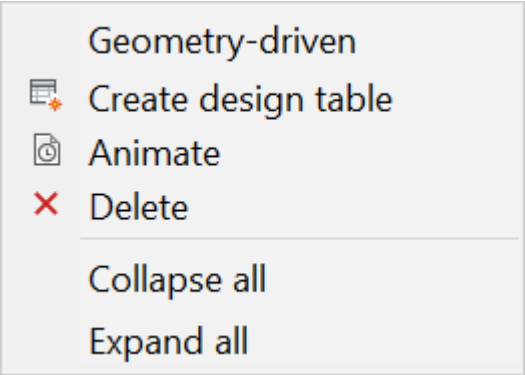


- **Expressions of constraints**:Controls whether the numeric value or the assigned parameter name displays.
- **Components parameters**:Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.
- **Expressions of components parameters**:Sub-component parameters control the visibility of sub-component parameters.
- **Sub-components of standard parts**:Parameters at properties add a parameters section for the selected instance to the **Mechanical Browser** properties.
- **Always synchronize selection**:When checked,theselectednodes in the browser mirror the selection in the document.

(5) **Show search** (Ctrl+F): Show or hides the search bar.

(6) The name of the parametrized detail.

(7) **Parameters**: Right-click on a parameter to open a context menu:



- **Geometry-driven**:If checked, it makes the parametergeometry-driven.
- **Create design table**: Creates a design table to drive parametric block parameters.
- **Animate**:Animates models by means of parameters.
- **Delete**: Deletes the selected parameter.
- **Collapse all**: Collapse the whole **Mechanical Browser**.
- **Expand all**: Expand the whole **Mechanical Browser**.

(8) **Tolerance Parameters**: The maximum and minimum variation of an angle, thickness or dimension from the original parameter value.

(9) **Bodies**: Display the bodies in the model.

(10) **Constraints**: The constraints you can delete, modify or add.

(11) **Component**: The properties of the detail file.

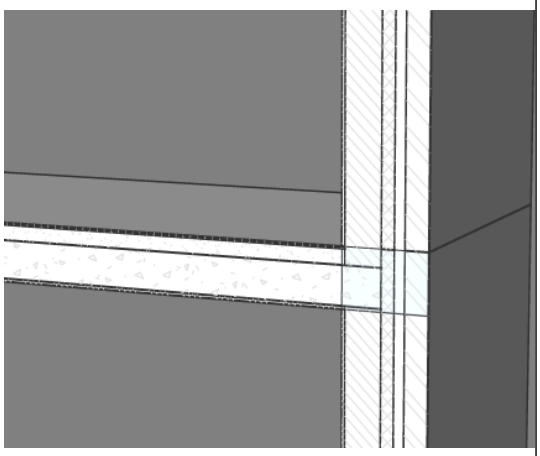
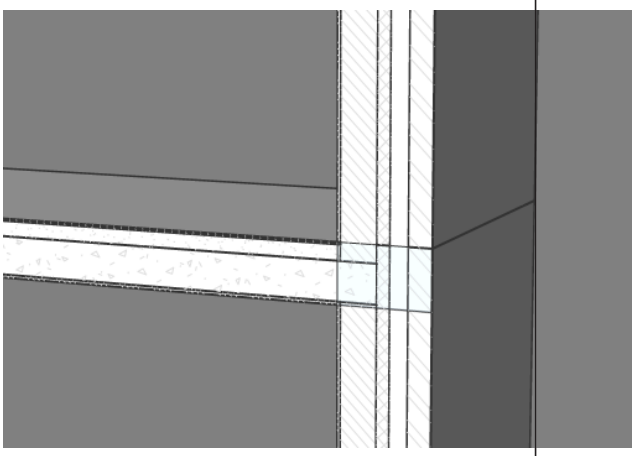
7 ***Nota:*** Some default constraints are generated that link the detail volume and the detail objects to the reference solids. You can delete and replace them with other constraints.

8 Edit the parameters.

9 The parameters that were generated have specific values to ensure the detail looks exactly as it was created. If you change these values, the detail will change.

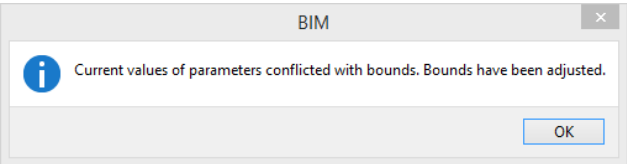
For example:

- Change the **Thickness_Ply1_Wall_30_31** parameter from 40 to 70 (in the **Expression** field). The new detail is visibly different in the model space:

Before	After
	

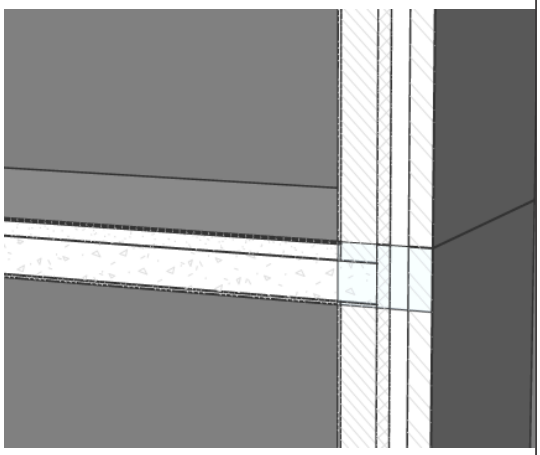
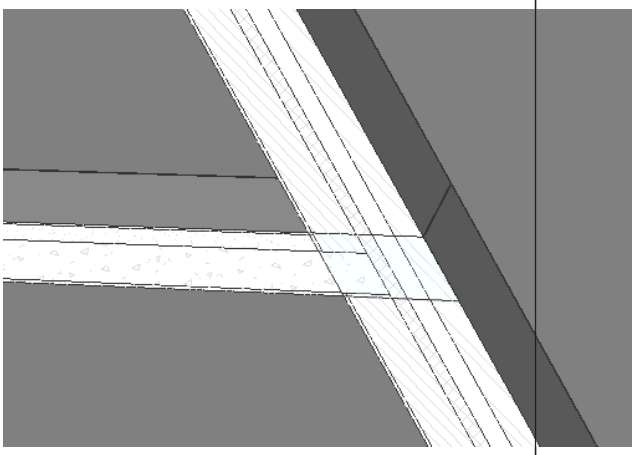
- Change the **Angle_32_30** parameter from 90 to 120. The new angle between the wall and the slab is different in the model space.

  A warning pops up:



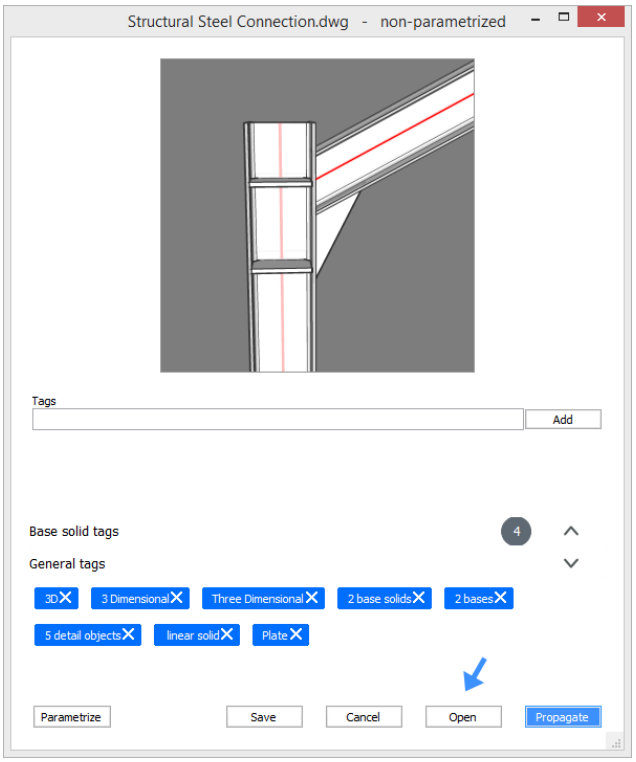
  This is because the **AngularTolerance** parameter was set to 10. The new **AngularTolerance** parameter automatically changes to 30 because you changed the **Angle_30_31** parameter by 30 degrees.

- Click **OK**

Before	After
	

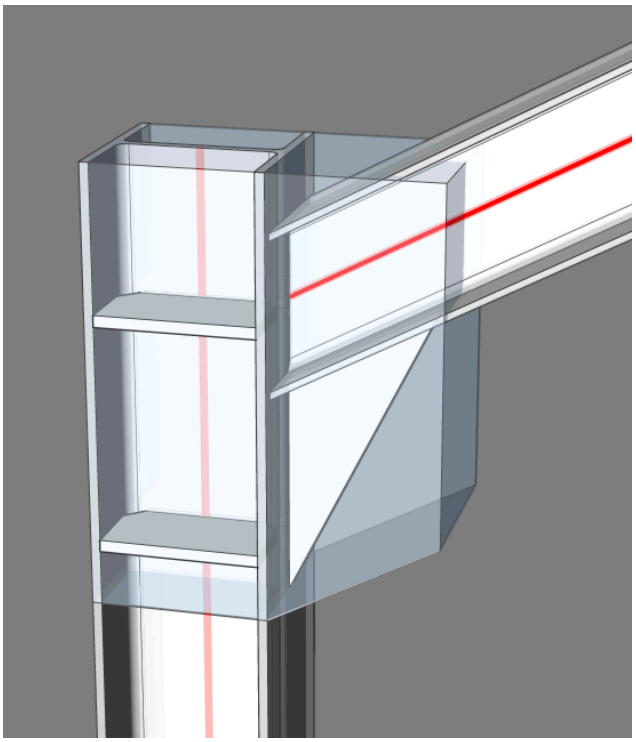
20.5.8.4 Parametrize and edit a structural steel connection

1 To open the .dwg file of the detail you want to parametrize, click on the detail in the **Details** panel. The **Detail** dialog box displays:

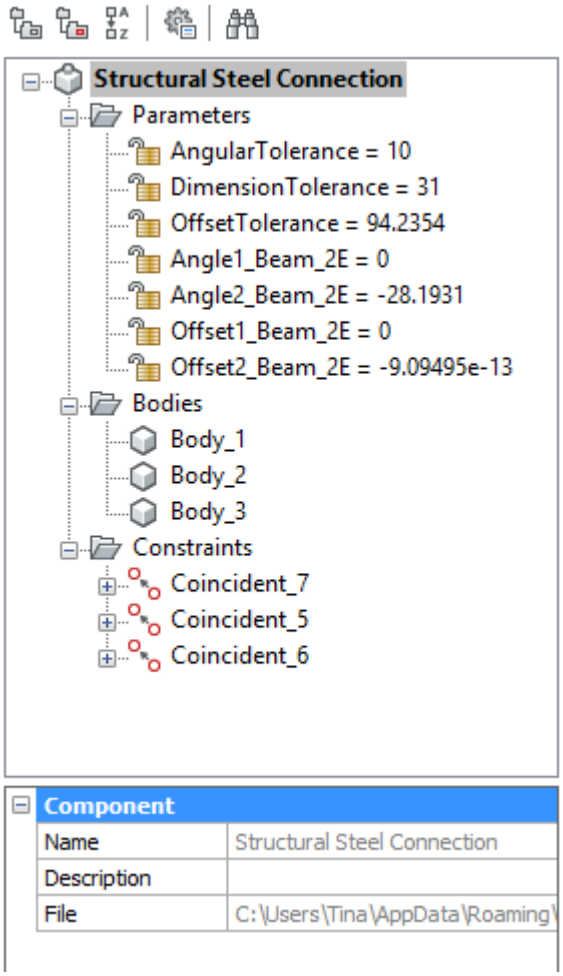


2 Click **Parametrize**, or click **Open** and run BIMPARAMETRIZEDETAIL.

The detail .dwg file will open and the detail will be parametrized.



3 Open the **Mechanical Browser**:



Nota: For connections between linear reference solids, only the framework parameters are automatically generated. You must manually add constraints to specify the behavior of detail objects in relation to the reference solids. In this example, three coincidence constraints have been manually added: two to make sure that the connection planes of the triangular steel plate maintain coincident with the flanges of the steel column and girder, and one to make sure that the end section of the girder maintains coincident with the column flange. You can add coincidence constraints before or after the parametrize tool. If you add the coincidence constraints after you can see how the reference solids change with new parameters. This makes it clear which constraints have to be added to make all detail objects move correctly.

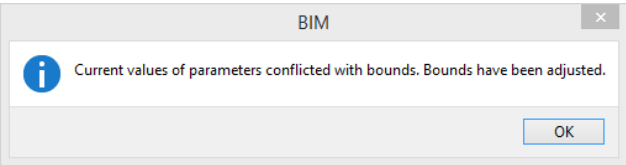
4 Edit parameters.

5 The parameters that were generated have specific values that make sure the detail looks exactly as it was created. If you change these values, the detail will change.

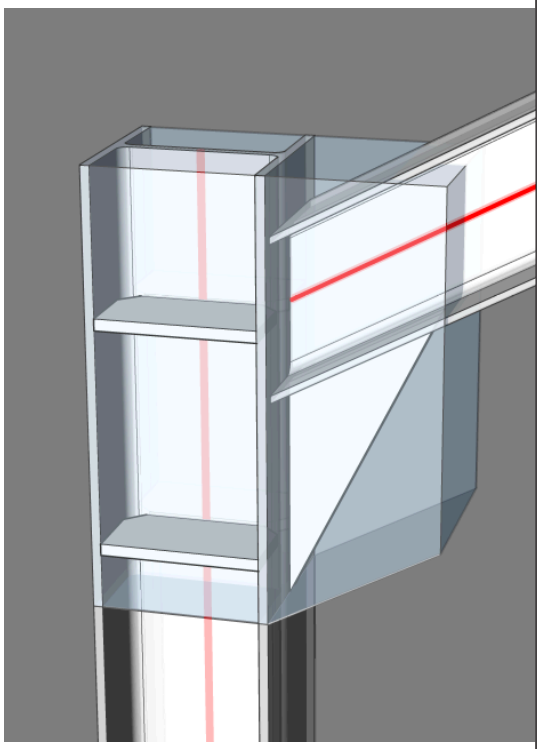
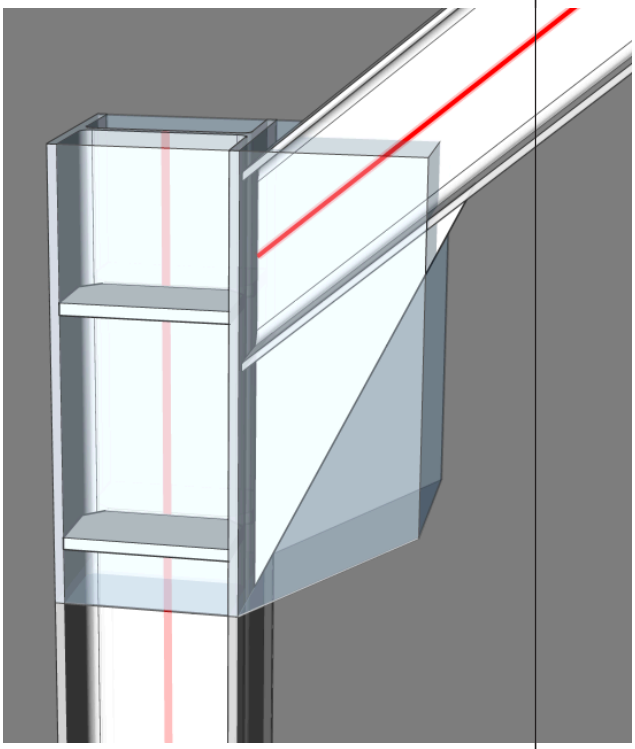
For example:

- Change the **Angle2_Beam_2E** from -28.1931013282 to -40. The column and the girder change in the model space.

  A warning pops up:

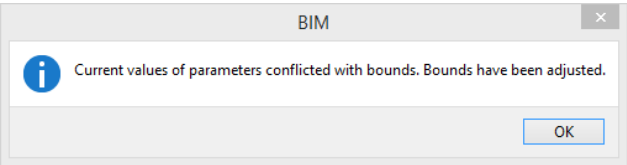


This is because the **AngularTolerance** parameter was set to 10. The **AngularTolerance** parameter automatically changed to 11.8069 because you have changed the **Angle2_Beam_2E** by 11.8069 degrees. Click **OK**.

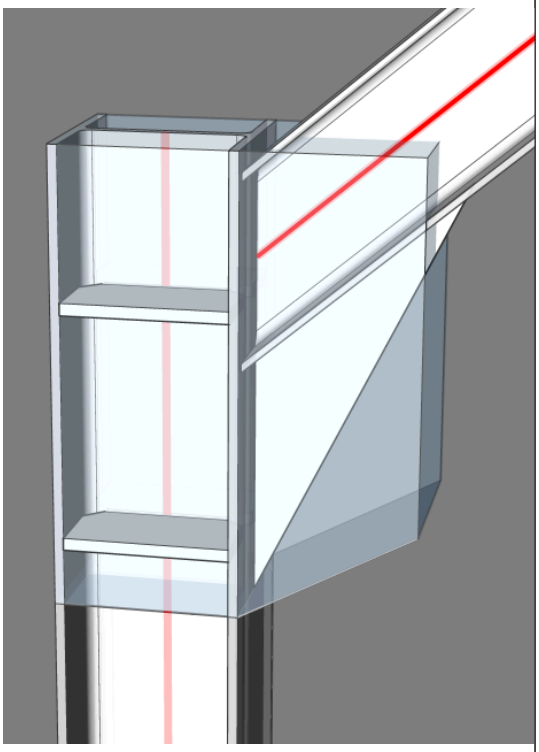
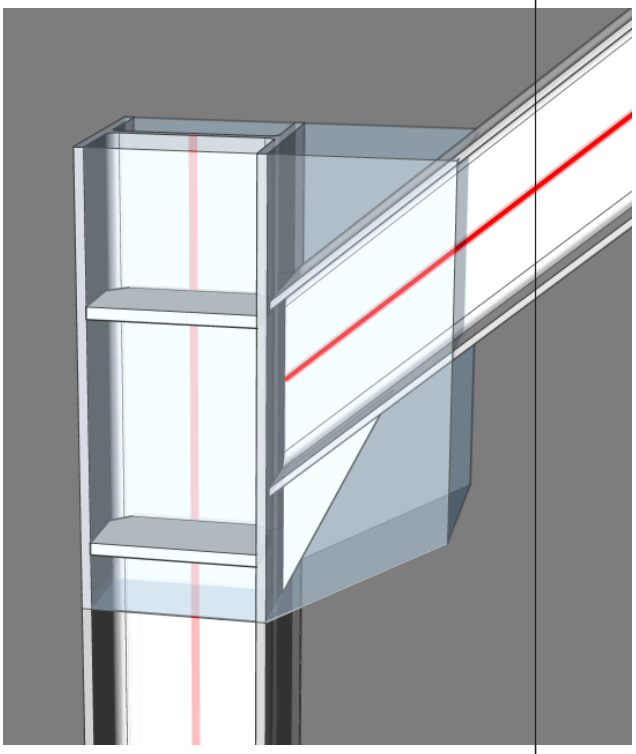
Before	After
	

- Change the **Offset1_Beam_2E** 0 to 200. You see the girder moves 200 mm downwards.

  A warning pops up:



This is because the **OffsetTolerance** parameter was set to 31. The **OffsetTolerance** parameter automatically changed to 200 because you changed the **Offset1_Beam_2E** parameter by 200 mm. Click **OK**.

Before	After
	

20.5.8.5 Propagate a detail in a project

There are different methods to propagate a detail:

- With the **Details** Panel: propagate a detail from the project on all similar connections in the project.
- With the BIMPROPAGATEFROMFILE command: propagate a detail from a file.

Fewer suggestions are given by the BIMPROPAGATE command for a detail that is not parametrized. This is because a detail that is not parametrized has fewer possible locations.

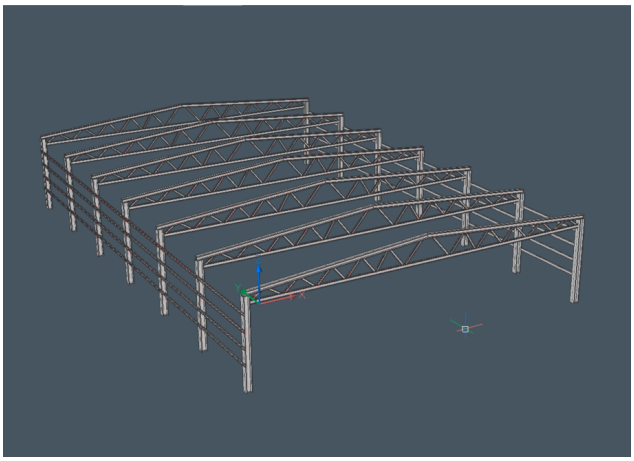
For example: if a non-parametrized detail of a wall-slab connection is propagated, BricsCAD BIM won't be able to propagate this detail to a wall-slab connection when the composition is different or a particular ply in the wall differs 10 mm from the corresponding ply in the wall of the original detail. To solve this problem parametrize the detail.

The BIMPROPAGATE command can adapt a detail with parameters to situations that differ from the original (eg: another corner, a smaller profile, etc.). You can decide how much the connections can differ from the original with the tolerance parameters.

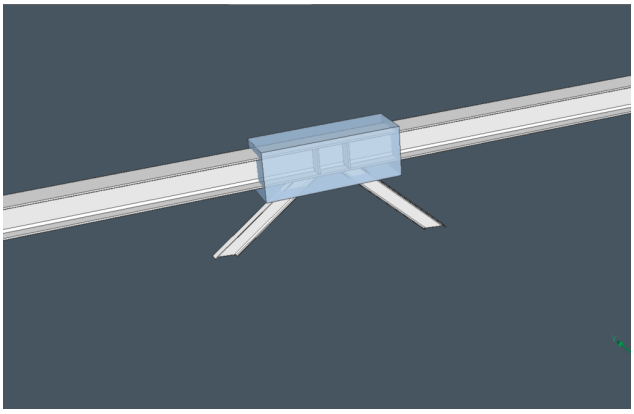
Different tolerance parameters:

- **Angular**: Sets the tolerance on angle parameters of the detail in degrees. Angles in the target situation cannot deviate more than this tolerance value from the original angles in the detail.
- **Dimension**: Sets the tolerance for the profile dimensions of linear reference solids in the detail. Profile widths and heights in the target situation cannot deviate more than this tolerance from the original widths and heights in the detail.
- **Offset**: Sets the tolerance for offset parameters of the detail. Offsets in the target situation cannot deviate more than this tolerance from the original offsets in the detail.
- **Thickness**: Sets the tolerance for the total thickness of planar reference solids in the detail. Thicknesses in the target situation cannot deviate more than this tolerance from the original thicknesses in the detail.

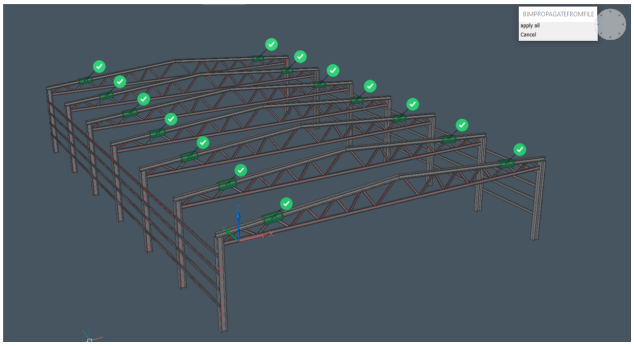
In the following project, there are a lot of similar connections present. However, the angles of the connections vary and the lower beam in the truss and has a different profile size than the upper beam.



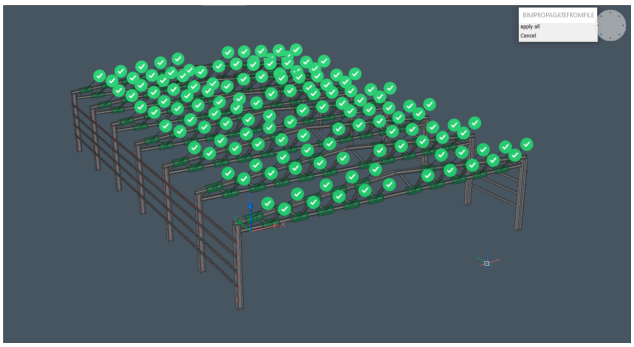
Propagate a detail in the project:

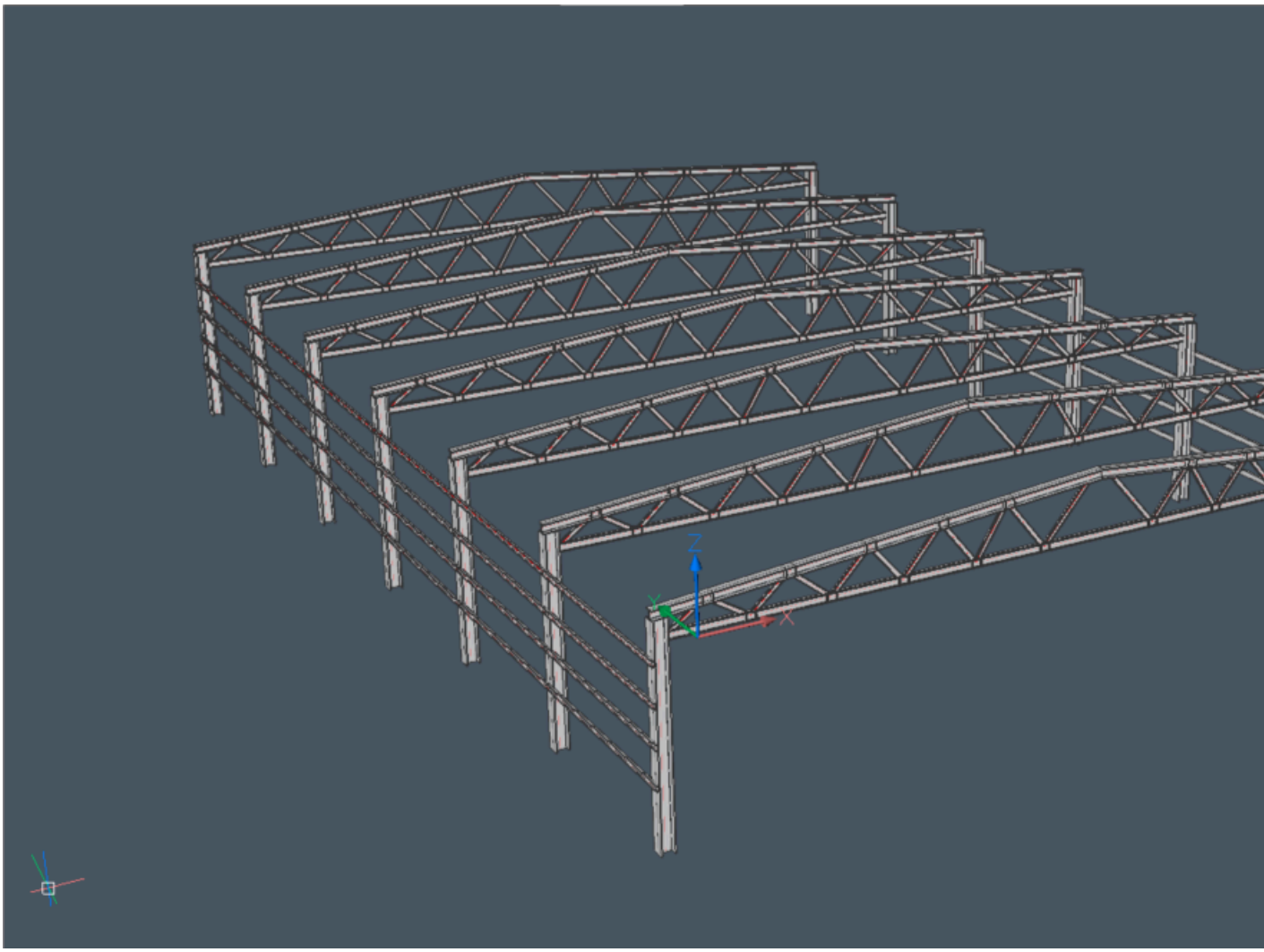


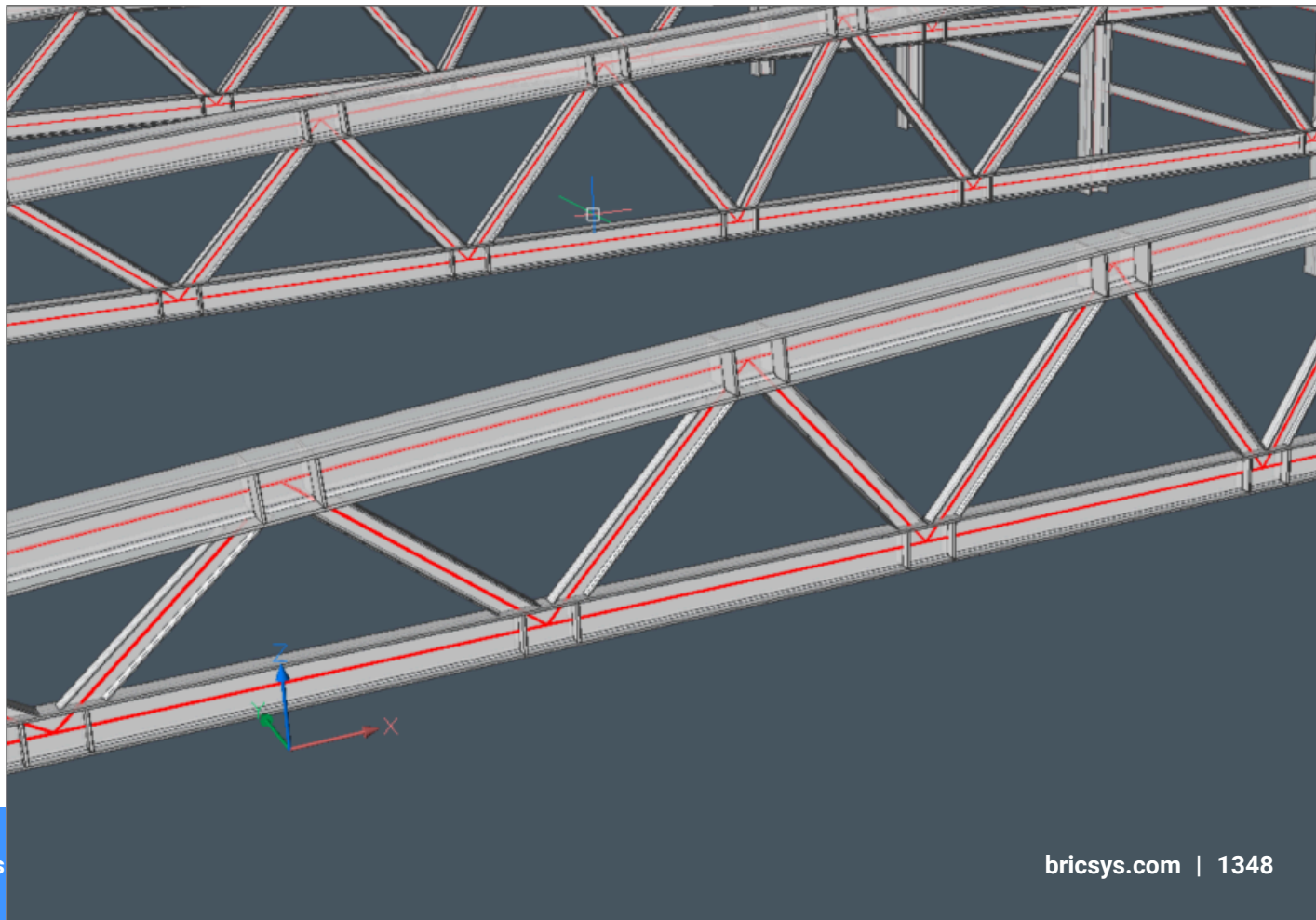
The **Mechanical Browser** displays the parameters. The **AngularTolerance** parameter is set to 0 degrees, the detail will only propagate to connections with exactly the same angle.



Change the **AngularTolerance** parameter to 45 degrees. The detail will propagate to all the similar connections. This is because none of the connections exceed the tolerance value.




Z
Y
X


Z
Y
X

20.5.9 Details panel

20.5.9.1 Commands

DETAILSPANELOPEN, DETAILSPANELCLOSE

20.5.9.2 Opening the details panel

Execute the DETAILSPANELOPEN command.

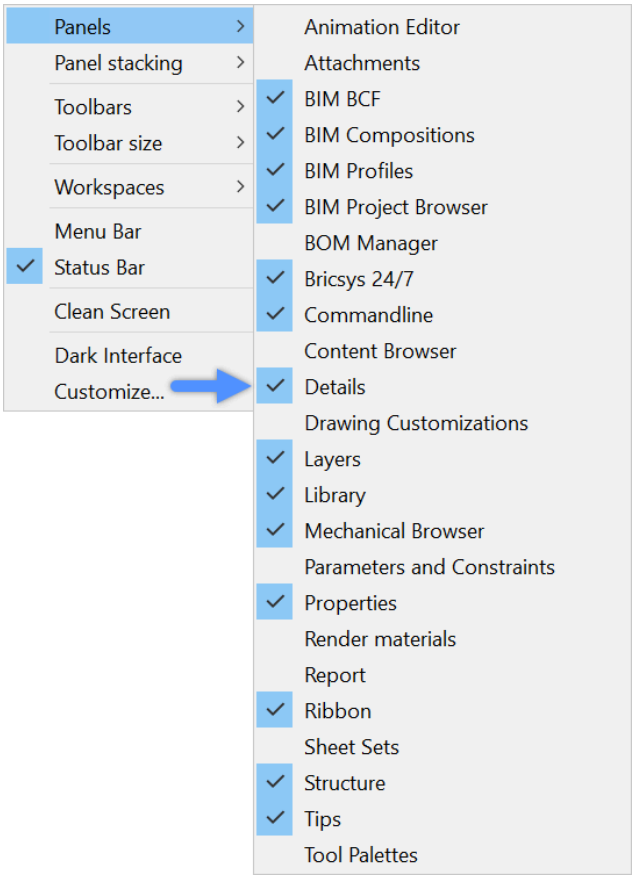
or

1 Place the cursor on a toolbar or ribbon panel and right-click.

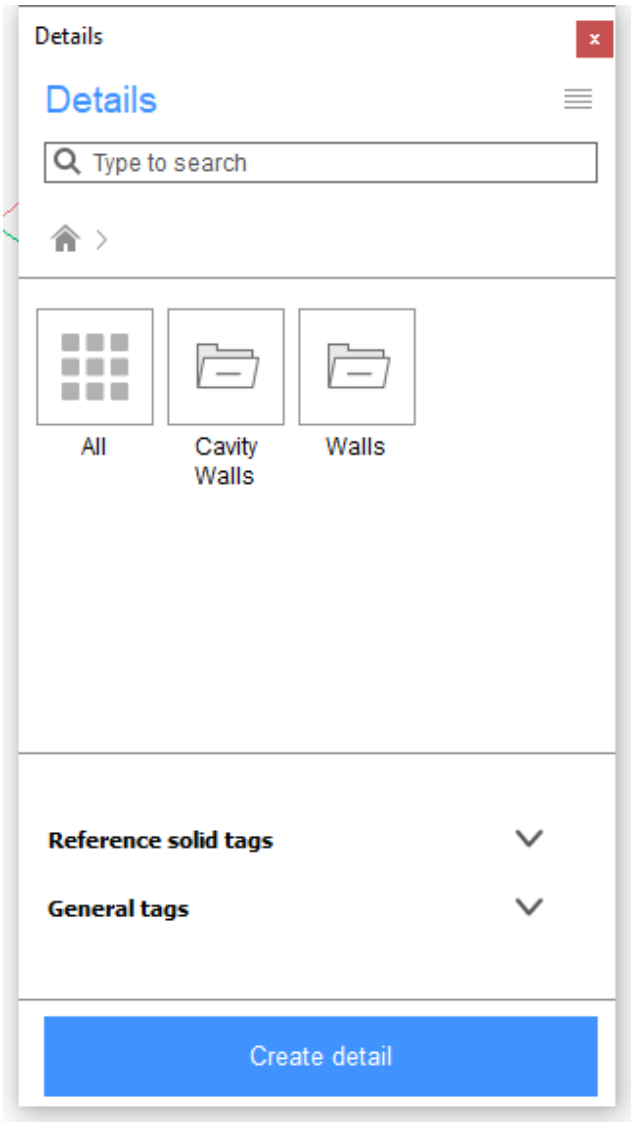
1 A context menu displays.

2 Expand the **Panels** option.

3 Select **Details** from the context menu.



The Details panel displays and the **Details** button ( ) is added to the collapsible panelset at the right-hand side of the BricsCAD application window.



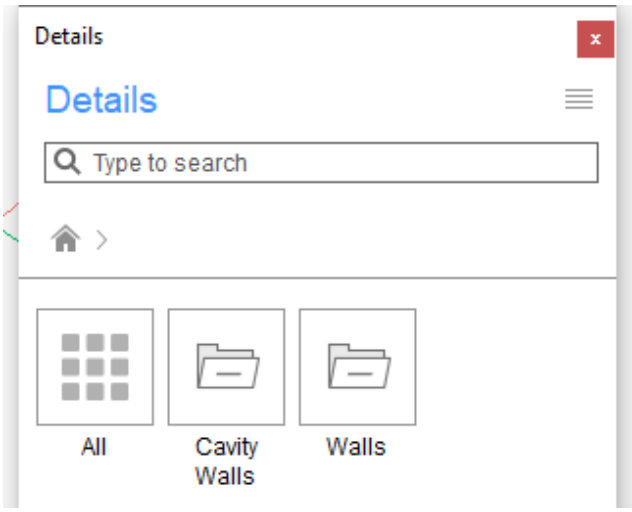
Click on the **Details** button ( ) to expand/collapse the **Details** panel.

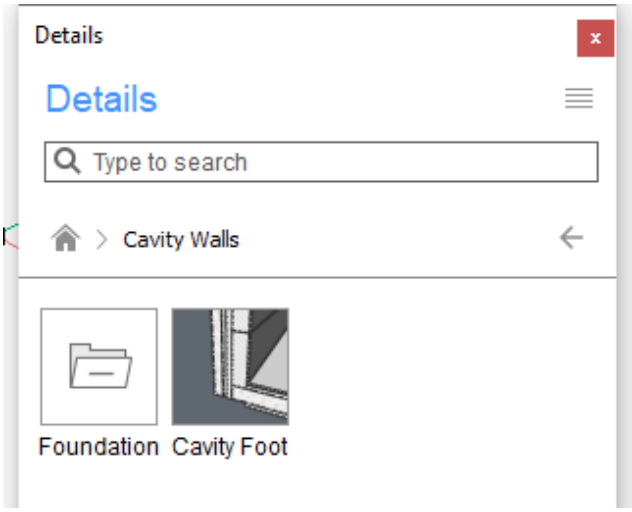
20.5.9.3 Saving details to the details panel

After you created a new detail, it's possible to save it to the **Details** Panel. This way you can use the same detail in other files as well. Use the command BIMCREATEDETAIL.

20.5.9.4 View a detail

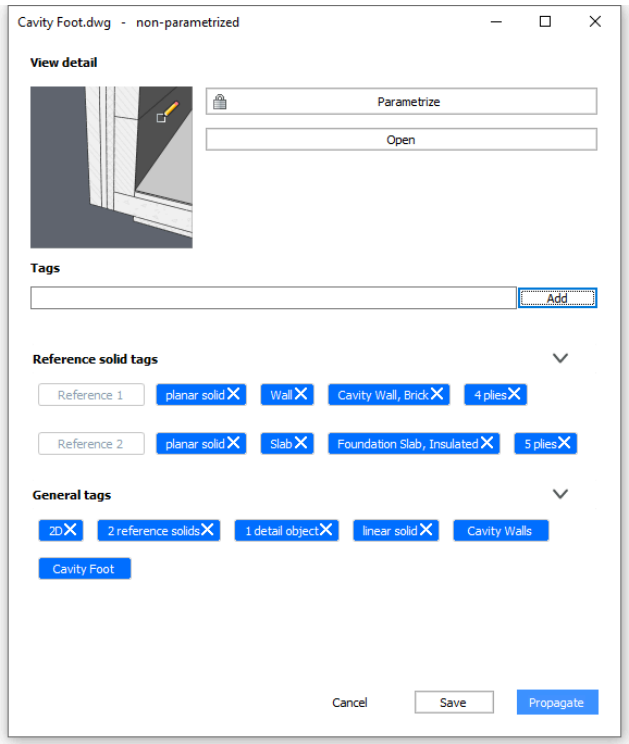
1 Click on the category of the detail you want to view. The details assigned to that category display.





2 Click on the detail you want to view.

3 The **Detail** dialog box displays:



- Edit the thumbnail image:
  - Click the **Pencil** icon ( ).

    The detail dwg opens.

    The CREATETHUMBNAIL command is launched.
  - Use view commands (zoom, pan, rotate) to adjust the view.
  - Choose **OK** or press Enter to save the thumbnail.
- **Parametrize**: Opens the .dwg detail file and executes the BIMPARAMETRIZEDETAIL command.
- **Open**: Opens the .dwg file of the detail.
- **Tags**: Add general tags.
- **Reference solid tags**: Click the down arrow to display the tags. You cannot add reference solid tags; you can only delete the existing tags.
- **General tags**: Click the down arrow to display the tags.
- **Save**: Saves the changes and closes the **Detail** dialog box.
- **Propagate**: Propagates the detail. See the BIMPROPAGATE command.
- **Cancel**: Closes the detail dialog box without saving.

Tags can be added to each detail. These are used to search for a specific detail in the library. There are two different tag categories:

- **Reference solid tags**: Tags that identify an individual reference solid from the detail.
- **General tags**: Tags that generally identify the detail.

During the BIMCREATEDETAIL command, tags are automatically generated for the detail. You can delete unrequired tags and add new tags. New tags can only be **General tags**.

You can add tags during the BIMCREATEDETAIL command or at any moment after the detail has already been created.

To add tags during the BIMCREATEDETAIL refer to the BIMCREATEDETAIL article. To add tags at another moment, see below.

Managing tags

1 In the **Details** panel, click on the detail you want to adjust.

The **Detail** dialog box displays.

2 Expand the **Reference solid tags** category.

You can only delete these tags, you cannot add new tags.

3 Click in the **Tags** field and type the name for the new tag, then click the **Add** button to add **General tags**.

The new tag is added to the **General tags** category in an orange box.

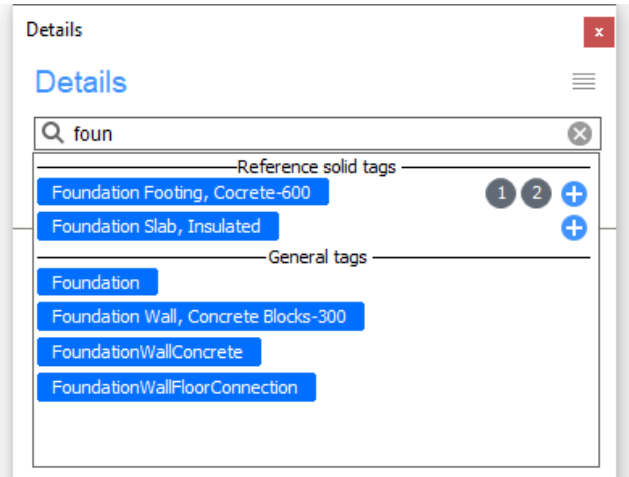
Repeat this procedure to add more tags.

4 Click the **Save** button to save the changes and close the **Detail** dialog box.

Searching details with tags

In the **Details** panel you can search the library for specific details using tags. You can add one or more tags at once to your search query to select all matching details.

1 Click the search bar in the **Details** panel and type in a search string. The matching tags display: **Reference solid tags** and the **General tags**.



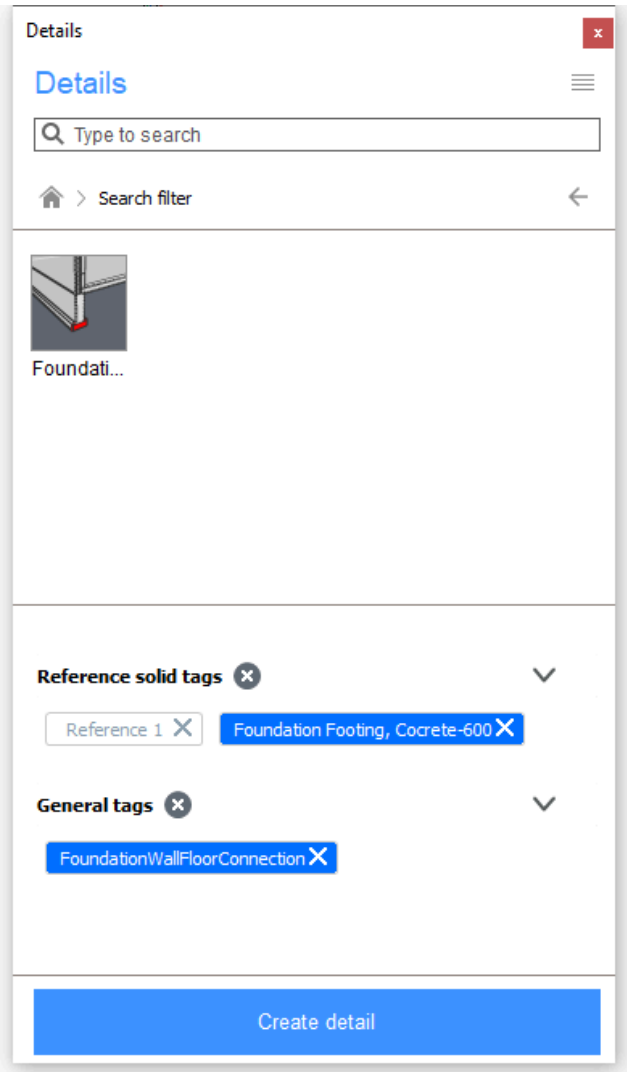
2 Do the following:

- **General tags**: Click the desired tag.
- **Reference solid tags**:
  - To add the tag to a reference solid that is not yet in your search query: click the plus sign next

to the required tag.

- Add the tag to a reference solid that is already in your search query: click the number of the required reference solids next to the tag.
- Once these tags have been added to your search query, they will appear at the bottom of the **Details** panel under the right tag category.

  The filtered search results display.



20.5.9.5 Editing a detail

1 Select the detail you want to edit.

2 Click the **Open** button.

  The .dwg file of the detail opens in a separate drawing tab.

3 Make the necessary adjustments to the detail.

4 Save the detail .dwg file.

5 The detail is updated in the **Details** panel.

6 The existing details won't be updated automatically. To replace the existing details by the new one, follow the steps below to propagate a detail.

20.5.9.6 To propagate a detail

You can propagate a detail to all of the similar connections in your project.

Do one of the following:

Drag and drop the detail from the library into the drawing.

or:

1 Select the detail in the library.

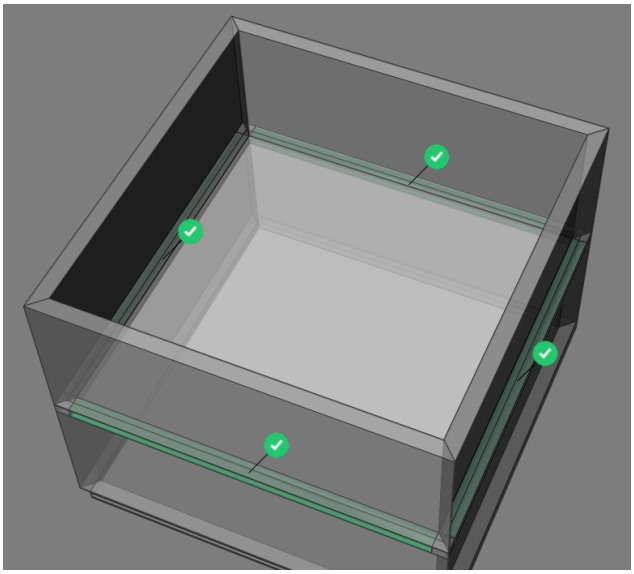
1 The **Detail** dialog box dispays.

2 Click the **Propagate** button.

To apply all suggestions, hit Enter.

To reject a suggestion click on the checkmark. A red cross icon replaces the checkmark. Click the icon again to accept the suggestion.

Press Enter to accept.



You can also propagate a detail in your project using the BIMPROPAGATEFROMFILE command. This command allows you to propagate detail files that are not in the library.

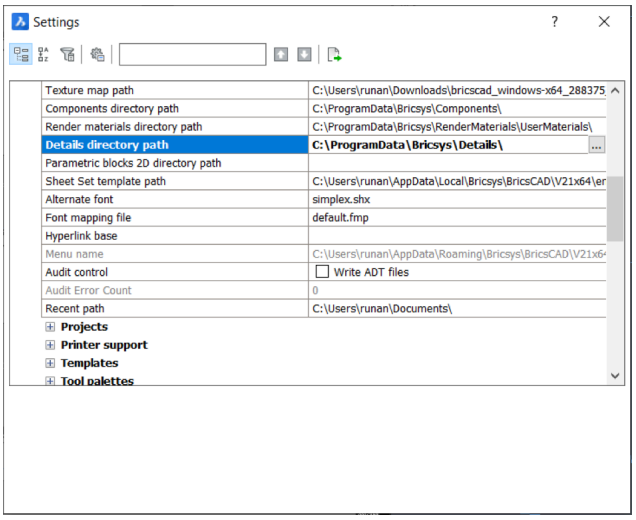
20.5.9.7 Sharing details

You can save details as a dwg-file and share the details folder/library. Save the detail drawings at the correct location which will then display in the details panel.

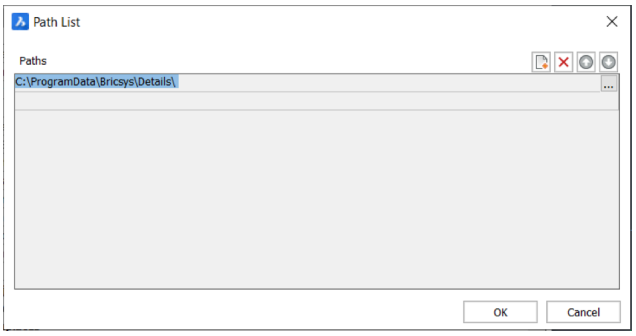
1 If you want to make a new detail to share, you can create a new detail first, as is explained above;

1 The following steps apply both to new details as well as existing details.

2 All the details you have created are automatically saved in the folder, and its subfolders, which is set through the DETAILSPATH system variable. By default, this is **C:\ProgramData\Bricsys\Details**.



Then click on the three dots next to the existing name in the **Path List** to change the location.

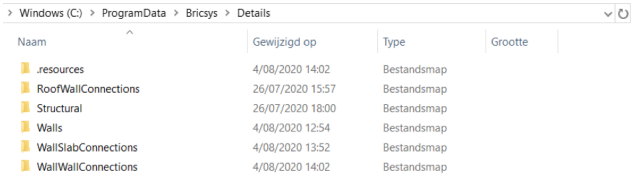


You can also create additional paths.

Click the **New** icon (), then click on the three dots of the new row to add a path for the **Details** library.

3 If you want to share a folder, go to the file explorer on your computer and follow the path to the details folder. If you haven't created any details yet, this folder only contains the folder '.resources'.

4 If you have created some details, you can find the same folders here as in your **Details** panel.



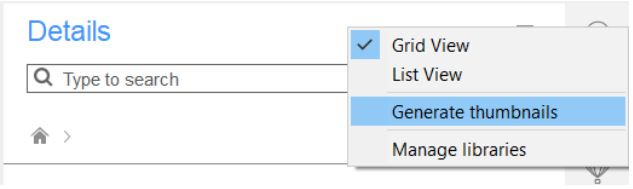
5 Copy the folder that you want to share and turn it into a zip file. This zip file can be shared with others.

6 If you receive a zip file you should unpack it and copy the folders containing the details to the folder *C:\ProgramData\Bricsys\Details*.

Nota: Make sure the details are saved in a folder that you place in the **Details** folder. If you just place the dwg files in your **Details** folder, they will not appear in your **Details** panel.

7 To load the new details in your library panel, go to the **Details** panel, open the menu and click **Generate thumbnails**.

The new details folders will appear in your **Details** panel.



20.5.10 Rhino/Grasshopper integration

20.5.10.1 Commands

RHINO, GRASSHOPPER, TOGRASSHOPPER

20.5.10.2 About

Rhinoceros3D is a 3D modeler developed by Robert McNeel & Associates of Seattle, Washington USA. Rhino is used to create, edit, analyze, document, render, animate and translate NURBS (curves, surfaces, solids, point clouds and polygon meshes).

Grasshopper is a visual programming language environment that runs within the Rhinoceros 3D computer-aided design application. It is tightly integrated with Rhino's 3D modeling tools.

20.5.10.3 Installing Rhino/Grasshopper connection

1 Download Rhino 7 from https://www.rhino3d.com/download.

Nota: Since the releases of BricsCAD V21 and Rhino 7, you can use the Rhino/Grasshopper Connection within BricsCAD with trial license keys of both BricsCAD BIM V21 and Rhino 7. Keys for older versions of Rhino will not work for this integration.

2 Download the Grasshopper-BricsCAD Connection from the BricsCAD Application Store: https://www.bricsys.com/applications/a/?rhino-grasshopper-connection-for-bricscad-a1353-al2360. The installer copies all required files to the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

Nota: The Grasshopper-BricsCAD Connection currently available from the Application Store will only work within BricsCAD BIM V21, not within older versions.

3 Restart BricsCAD V21.

4 To launch Grasshopper or Rhino, open and save a new drawing. Then use the buttons in the new tab that is displayed in the ribbon or launch with the Quad.

To launch a Grasshopper file from BricsCAD

1 Open an empty BricsCAD file.

2 Save it.

3 Open Grasshopper with the button in the new **Grasshopper** tab inside BricsCAD.

4 You can close the Rhino pop-up window. Rhino will continue to work in the background.

5 In the **File** tab of Grasshopper, choose **New document** to open a new and empty Grasshopper script.

6 Choose **Open document...** to open an existing Grasshopper script. You can use any script you made with Grasshopper in the past and open it in BricsCAD, as long as the input geometry from another program has been internalized. (To internalize data right-click the input geometry component and hit **Internalise data**.)

7 The geometry you created with the script should now pop-up in your BricsCAD model space, displayed in red. Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed.

8 If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

To link a Grasshopper file with a BricsCAD file

By default, the BricsCAD file that the Grasshopper document will link to is the one that was active when you launched Grasshopper from BricsCAD. If you want to link it to another BricsCAD file, you need to:

1 Make the target BricsCAD drawing the current drawing (i.e. the one you see in the model space window of your current session).

2 Link the Grasshopper script to this open drawing. You do that by clicking the Link icon in the bar just above the Grasshopper canvas that contains the script that you are trying to link.

To launch a sample file

The sample files are provided with the Grasshopper-BricsCAD Connection. They can show you what is possible with the Connection. They are stored in the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

1 Open up a sample .dwg file by opening a .dwg from the 'Program Files\Bricsys\Grasshopper-BricsCAD Connection' folder.

2 You will get a warning that the file is read-only. This happens because the document is located in a secured folder. Click **Yes** to open as read-only.

Optional: If you don't want this warning, or if you want to edit the file, copy-paste the sample files into one of your own folders and open them from there instead.

3 From inside Grasshopper, choose **File** > **Open Document**. Go to the 'Program Files\Bricsys \Grasshopper-BricsCAD Connection' folder again.

4 Choose the .ghx-file with the same name as the .dwg file.

5 The sample file now opens and you should see red preview geometry in your BricsCAD model space.

6 Make sure you don't close the **Grasshopper** window, otherwise the preview geometry will disappear, as the Grasshopper document is closed. If your Grasshopper script is open and you still don't see the preview geometry, it probably means your Grasshopper file is linked to the wrong BricsCAD document. To link it to the correct one, see 'To link a Grasshopper file with a BricsCAD file'.

7 You can play around with the input sliders in the Grasshopper script.

8 If you are happy with your model and would like to convert the preview geometry into real BricsCAD geometry, you can 'bake' the geometry into BricsCAD. You do that by selecting all the **Bake Building Element** components (Shift-clicking them all) and pressing the **Bake** button .

9 A pop-up window appears, just click **OK** to accept.

10 The geometry is now converted into real BricsCAD geometry, even with BIM data attached to it.

11 You won't be able to save your changes to your files, as they were read-only. So, discard your changes or save the files as new ones in a different location.

20.5.10.4 Customizing the Rhino/Grasshopper connection

The code for the Rhino/Grasshopper Connection is 100% open-source and is available at https://github.com/Bricsys/rhino.inside-bricscad. This allows you to fully customize your Connector by coding to your heart's content (assuming, of course, that you know how to code).

20.5.10.5 Uninstall the Rhino/Grasshopper connection

1 Launch **Add or Remove Programs** in Windows.

2 Search for **Grasshopper-BricsCAD connection**.

3 Hit **Uninstall**.

20.5.10.6 How to draw a box in Rhino and bake into BricsCAD via Grasshopper

1 Open a new BricsCAD file.

2 Save the file.

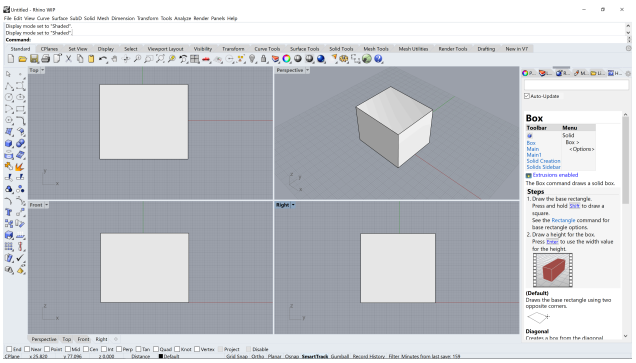
3 In BricsCAD, click on **Grasshopper** > **Rhino**.

4 Draw a Rhino box by typing BOX in the Rhino Command line. (This works in the same way as the BricsCAD BOX command).

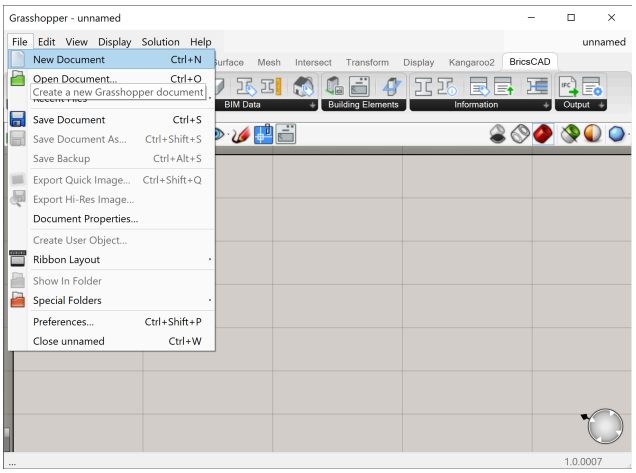
5 Click for the first corner of the base.

6 Click for the other corner of the base.

7 Click to define the height of the box.



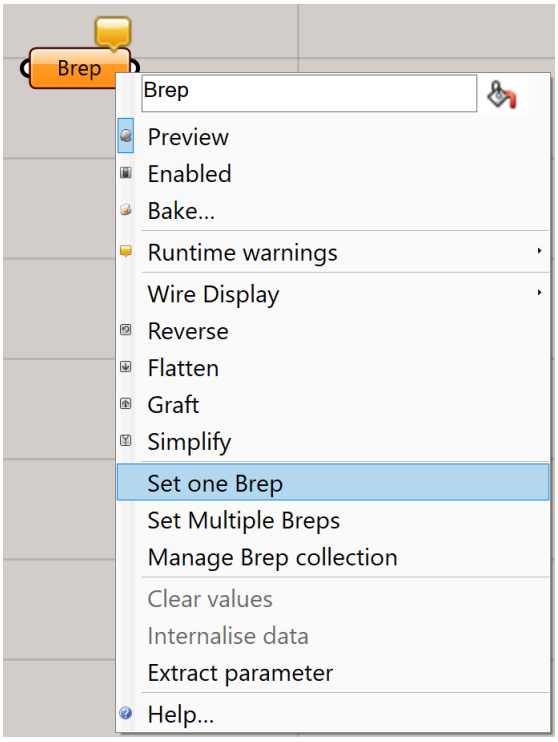
8 To open a new Grasshopper script, inside the Grasshopper environment, click on **GrasshopperGrasshopper** then click on **File** > **New Document**.



9 Save the file.

10 Drag and drop the **Params** > **Geometry** > **Brep component** onto the Grasshopper canvas.

11 Right-click on the component and select **Set one Brep**.



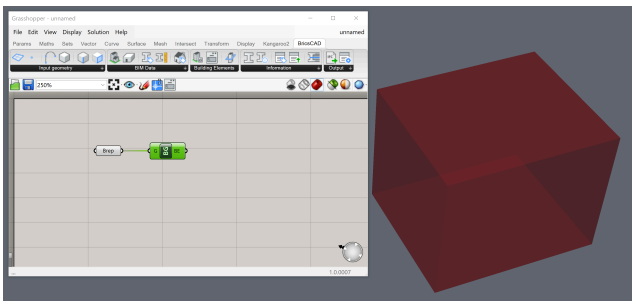
12 Go to the Rhino canvas and select your cube.

13 The Brep component will change color from orange to grey. If you close the Rhino window and return to the **Grasshopper** window, a preview of the cube will display in the BricsCAD model space.

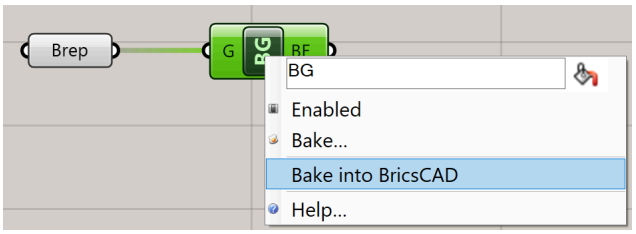
14 Drag and drop the **BricsCAD** > **Building Element** > **Bake Geometry component** onto the Grasshopper canvas.

15 To link the two components, click and drag from the Brep component's right dot to the BricsCAD Bake

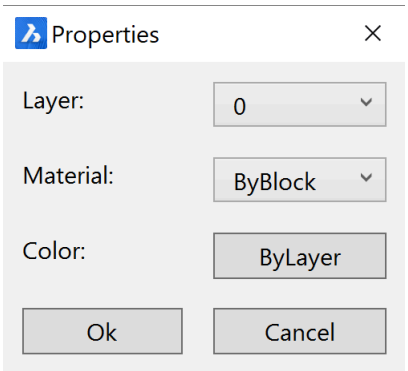
component's left dot.



16 Right-click onto the **Bake Geometry component** and choose **Bake into BricsCAD**.



The following dialog displays. Here you can choose the destination **Layer**, **Material** and **Color**:

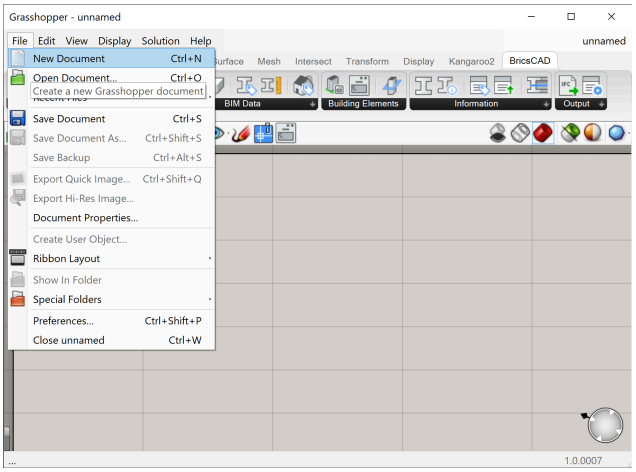


17 Click **OK**. The box is baked as a solid into BricsCAD.

20.5.10.7 How to draw a box in Grasshopper from a BricsCAD rectangle

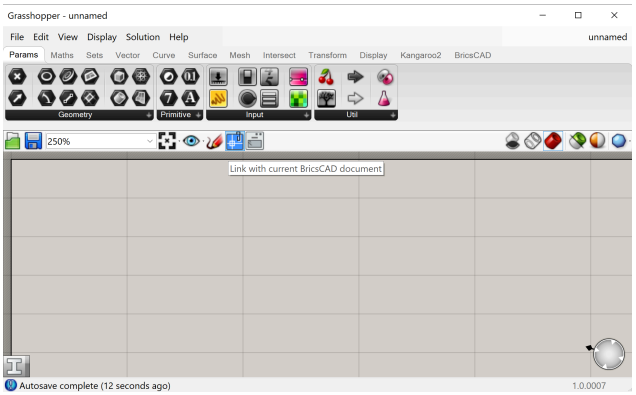
1 To open a new Grasshopper script, click on **Grasshopper** > **Grasshopper** and clicking on **File** > **New**

Document inside the Grasshopper environment. Save the file.



2 Open a new BricsCAD file in a **meter template** and save it.

3 To link the new script to the open .dwg file hit the **Link** button in the **Canvas** Toolbar.



4 Go to the BricsCAD model space and draw a rectangle.

Nota: Use a polyline to create irregular geometry.

5 Drag and drop the **BricsCAD** > **Input Geometry** > **Curve component** onto the Grasshopper canvas.

6 Right-click onto the **Curve component** and choose **Set One BricsCAD Curve**.

7 Go to the BricsCAD model space and select the rectangle.

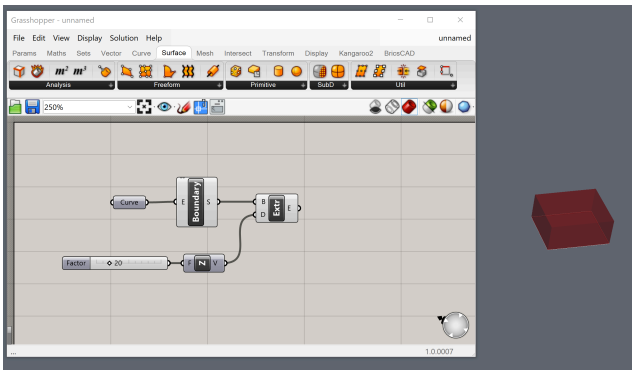
Nota: Save time with the Quad: select the curve in BricsCAD and then select **ToGrasshopper** under the **Grasshopper** tab of the Quad. A Grasshopper component appears in the upper left corner of your Grasshopper canvas.

8 Link the Curve component to a **Surface** > **Freeform** > **Boundary Surfaces component**.

9 Link this to a **Surface** > **Freeform** > **Extrude component**.

10 In the Direction input of the Extrude component link a **Vector** > **Vector** > **Unit Z component** with **Params** > **Input** > **Number Slider component**. Set this to the height of your box, 17 for example.

Nota: You can edit the range and preciseness of the slider by right-clicking the component and selecting **Edit...**. A dialog box pops up where you can set the digits and min and max value of the slider.



11 You now have drawn a box in Grasshopper, using BricsCAD geometry as an input.

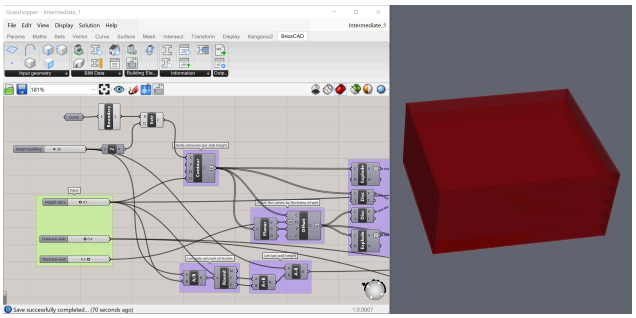
20.5.10.8 How to split up a box into different floors using Grasshopper

1 Add the 'Start_1.ghx'-code to your previous Grasshopper script.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_1.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.
   - Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_1).
   - You then get a drop-down of all the active Grasshopper scripts, so open the one you saved in 'How to draw a box in Grasshopper from a BricsCAD rectangle'.
   - Now click on the canvas and hit **Ctrl+V** to paste the 'Start_1.ghx'-code.

2 Right-click to rename the number slider from the previous procedure to **Height building**.

3 Link this number slider to input A from the Division component in the **Compute amount of stories** group and to input A from the Subtraction component in the **Get last wall height** group.

4 Link your extrusion to the Shape input of the Contour component.



5 Do some cleaning up of the file:
   - Drag the green **Input** group to the front of your canvas.
   - Add the **Height building** slider to the group by selecting the component and then right-clicking on the group and choosing **Add to group**.
   - Put off the preview for your Boundary and Extrusion components: select the components (Shift-click them) and right-click on an empty space of your canvas and choose **Preview Off**.

6 **(Optional)** For intermediate files that already have these changes, use 'Intermediate_1.ghx' and 'Intermediate_1.dwg'. (Located in the zip file at the bottom of this page.)

7 The script you just made allows you to:
   - Divide the extrusion per slab height, using the Contour component.
   - **Offset** the curves by the thickness of the walls.
   - Make the bottom surfaces of the walls (the surfaces are split into lower floor walls and roof walls).
   - Compute the last wall height.
   - **Extrude** the surfaces into roof, slabs, and walls.
   - Adapt the height of the building, the height of the stories, and the thickness of the slabs and walls, by using the sliders in the green **Input** group.

8 Add 3 **BricsCAD** > **Building Element** > **Bake Building Element components** to the end of your Grasshopper script.

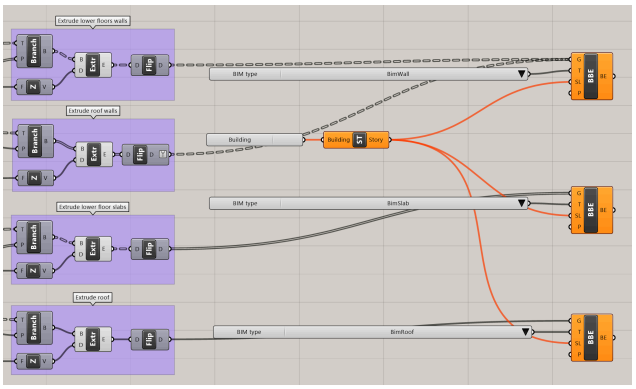
9 Attach the extruded elements (the Flip components) to the Geometry input nodes of the **Bake Building Element** components:
   - Connect the extruded lower floor walls and extruded roof walls to one **Bake Building Element** component. (Hold Shift while connecting multiple nodes to one input node).
   - Connect the extruded lower floor slabs to another **Bake Building Element** component.
   - Connect the extruded roof to the remaining **Bake Building Element** component.

10 Add the following components to your canvas from the **BricsCAD** > **BIM Data** category: 1 **Buildings**, 1 **Stories**, and 3 **BIM Types components**.

11 Attach Buildings to the Building input of the Stories component and attach this to the Spatial Location input of all the **Bake Building Element** components.

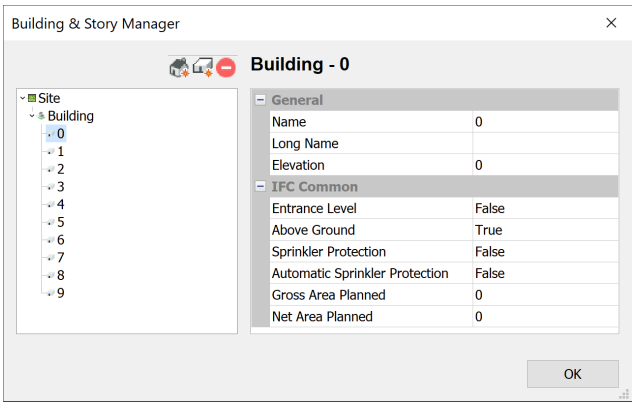
12 Attach BIM Types (set respectively to **BimWall**, **BimSlab**, and **BimRoof** in the selection menu) to the Element Type input of the **Bake Building Element** components of the extruded walls, the slabs, and the roof respectively.

13 The following warning will display: **Input parameter Building failed to collect data**. This is because, when no buildings are defined in the BricsCAD drawing, Grasshopper fails to collect input.

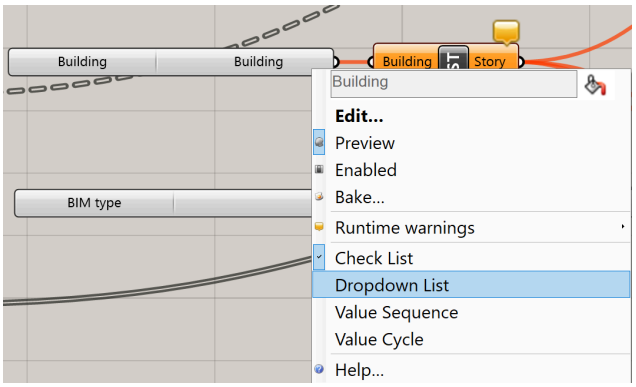


14 To make spatial locations, for the component to detect, go to BricsCAD and click on **Home** > **Classify** > **Spatial Locations**.

15 Add a building and a few stories (more stories than you want to divide your building into).



16 Go to Grasshopper and right-click the Buildings component and put it onto **Drop-down List** instead of **Check List**.

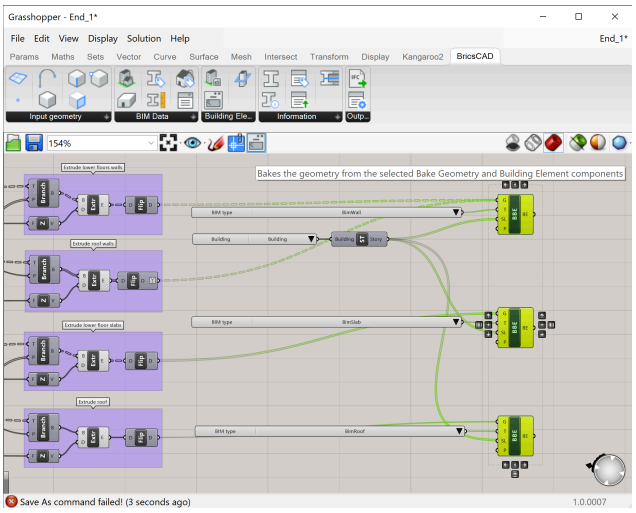


17 Select the newly made building in the drop-down list.

18 Shift-select all the **Bake Building Element** components and choose the **Bake** option from the **Canvas** Toolbar. Click **OK** to accept the **Layer**, **Material**, and **Color**.

19 The drawing will now have different floors.

Nota: Check the properties to see that the geometry is correctly classified as **Wall**, **Slab** or **Roof**. These elements will also have a spatial location assigned to them.

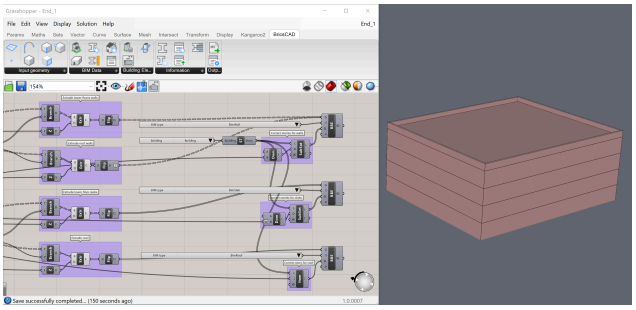


20 The last input geometry will be baked onto all remaining stories as there are now more stories than geometry. To correct this, select a correct subset of the stories to assign. Bake the geometry in BricsCAD and the stories will be correct. No 'duplicate' geometry will be baked.

21 You can see how it's done by opening the 'End_1.dwg' and then opening the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

22 You have now successfully created a building.

23 If you want to see the result of the baked building, check the file 'End_1_Baked.dwg' or bake it yourself.



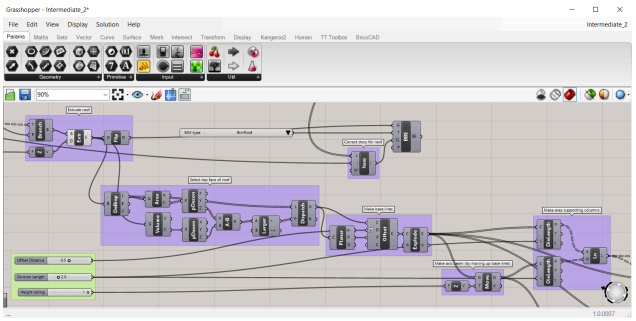
20.5.10.9 How to add a railing to the top of the building

We will continue with the building made in the previous procedure. So open the 'End_1.dwg' and then open the Grasshopper file 'End_1.ghx' that is linked to the 'End_1.dwg'.

1 Add the 'Start_2.ghx'-code to the Grasshopper canvas.
   - You do that by downloading the zip file at the bottom of this page and extracting its content.
   - Then open the 'Start_2.ghx' file by going to **File** > **Open Document...** in your **Grasshopper** window.
   - You can now select all of the Grasshopper code by hitting **Ctrl+A** on the keyboard and copy it using **Ctrl+C**.

- Then switch back to the document you were working on in the previous procedure by going to the upper right corner of the **Grasshopper** window and clicking on the title of the current document (Start_2).
- You then get a drop-down of all the active Grasshopper scripts, so open 'End_1.ghx'.
- Now click on the canvas and hit **Ctrl+V** to paste the 'Start_2.ghx'-code.

2 Link the **Extrude Roof** group's end component to the Brep input of the Deconstruct Brep component of the **Select top face of roof** group.



3 The script does the following:
- The **Select top face of roof** group will select the top face of the roof.
- The remaining script will take that face and use it to make the baselines for the railing.
- It will then make the axes of the supporting columns and beams.
- It will also calculate half of the height of the beam profile.
- There is still a profile required, however, to complete this step and the following one. We will, therefore, create a profile from step 6 onward.
- With that profile and the axes, the beams and supporting columns are made as extrusions.

4 The intermediate result is found in the 'Intermediate_2.ghx' and 'Intermediate_2.dwg' files. (Located in the zip file at the bottom of this page.)

5 **(Optional)** Skip the **Select top face of roof** group and replace it with a **BricsCAD** > **Input Geometry** > **Face component**. Set it to the top face of the roof slab in your BricsCAD baked building. (Do this by right-clicking on the component, click on **Set one BricsCAD face** and select the face in the BricsCAD drawing). Connect this Face component with a **Params** > **Geometry** > **Surface component** and connect this surface to the correct inputs.

6 Go to **BricsCAD** > **BIM Data**.

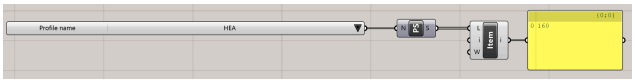
Drag the **Profile Names** and the **Profile Sizes** components onto the canvas.

7 Select **HEA** from the drop-down list in **Profile Names**. This drop-down list displays all the names of the profiles in the **Profile Library** of BricsCAD.

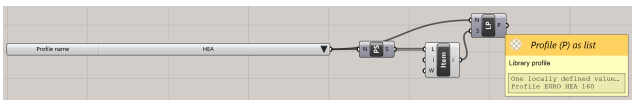
8 Link the component to the input **ProfileName** of the **Profile Sizes** component.

9 Link the **Profile Sizes** component to the input node **List** in a **Sets** > **List** > **List Item component**, where i is set to '4' (right-click on the **i** input and **Set integer** to **4**). This **List Item** lets you pick one size from the list with all available profile sizes for the **Profile Name** linked to it.

Nota: If you link a **Params** > **Input** > **Panel component** to the Profile Sizes output, you will see that the index 4 stands for size 160.

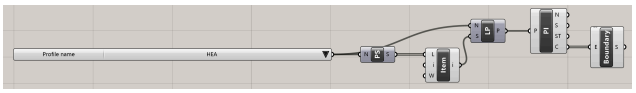


Link the **Profile Names** and **List Item** components to the **BricsCAD** > **Information** > **Library Profile component**, to get an HEA 160 profile as an output.



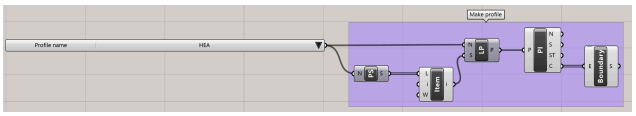
10 To get the curves that make up this profile, attach a **BricsCAD** > **Information** > **Profile Info component** to the **Library Profile**. One of the outputs created is the **Profile Curves** as a tree.

11 Attach a **Surface** > **Freeform** > **Boundary Surfaces component** to the **Profile Curves** output node. This is to later extrude the surface as a solid, instead of the lines as a surface.

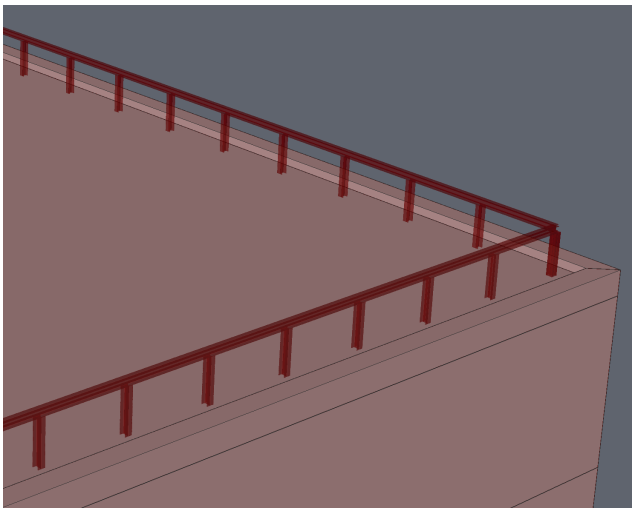


12 To group together the components just created, select them (with a selection box) and right-click on **Group**.

13 **(Optional)** To rename the group, right-click on the purple rectangle. Rename the group in the upper text box, e.g. to **Make profile**.



14 The Boundary Surfaces component should now be linked to the Content input of the Bounding Box component of the **Calculate half of height of beam** group. The Boundary Surfaces component should also be linked to the two Profile inputs of the Extrude components in the groups **Make supporting columns** and **Make beams**. The extrusions should now show you a railing with HEA 160 beams and columns.



15 To bake those beams and columns into BricsCAD attach two **BricsCAD** > **Building Elements** > **Bake Building Element components** to the Grasshopper canvas.

16 Do the following:
   - Set the **Geometry** input to the extrusions.
   - Set the **Element Type** input to a **BricsCAD** > **BIM DataBricsCAD > BIM Data > BIM Types component**, which is set to **BimColumn** for the column extrusions and to **BimBeam** for the beam extrusions.
   - Set the **Spatial Location** input to the same as the roof.
   - Set the **Profiles** input to the **Library Profiles** component from the **Make profile** group.

17 You now successfully created a railing on top of your building. (The full script to do this is in the 'End_2.ghx' file at the bottom of this page, which is linked to the 'End_2.dwg'.)

18 Bake into BricsCAD. The profiles are now set to the one in the Grasshopper script. You can still modify the connections of the beams in the corners by using L-Connect in BricsCAD. (To see the finished building with the railing on top of it, check out the 'End_2_baked.dwg' file at the bottom of this page.)

20.5.10.1 Procedure: add information to a baked model

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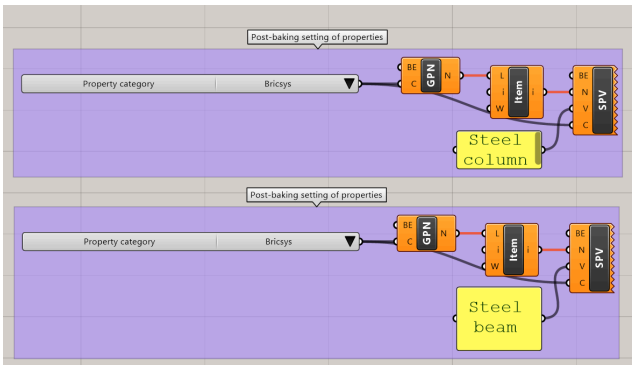
We will continue with the building made in the previous procedure. So, open the 'End_2.dwg' and then open the Grasshopper file 'End_2.ghx' that is linked to the 'End_2.dwg'.

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**.

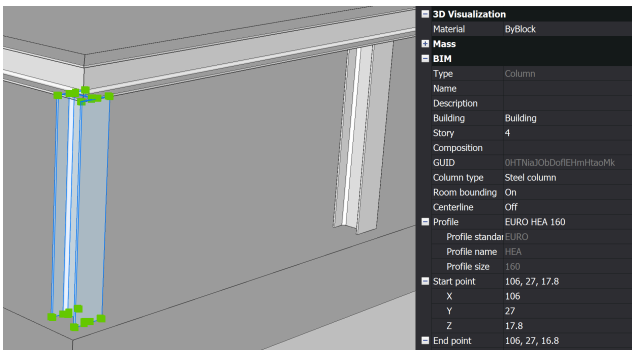
3 To make the script work, bake the two **Bake Building Element components** that make up the railing.

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 Collaborazione al Progetto

20.6.1 BIM collaboration format (BCF) panel

20.6.1.1 About BCF

The BIM Collaboration Format (BCF) is an open file format used to exchange textual comments, images, and more on top of an IFC model. This allows better communication between different parties in the BIM process.

BCF makes it possible to link information such as a comment, a screenshot, a list of involved objects, and a camera viewpoint to an issue. An issue can be anything ranging from a small change in the design to clashes between different disciplines in the model. These issues can then either be saved as .bcf files or they can be managed on the cloud, using any sort of BCF cloud management system.

The advantage of BCF is that it allows communication over different software packages (e.g. a BCF issue created in Solibri can be opened in BricsCAD BIM), and that it enables easy navigation through a model based on issues.

20.6.1.2 BCF files

Issues can be saved in standalone files. This is generally a more cumbersome workflow than working with a cloud system but it can be useful in case you are working offline. A .bcf file can then be imported in any BCF manager (e.g. BricsCAD BIM) to read the issues and see where in the model it applies.

Note that a .bcf file cannot be created in BricsCAD BIM. For this, you will need access to another service e.g. Solibri Model Checker or BIMcollab ZOOM. This software allows you to create a .bcf file based on one or more .ifc files. You can then create issues containing:

- Comments
- Screenshots or other images
- A list of involved objects, based on GUID (Global Unique IDs) to distinguish them unambiguously
- A camera position

20.6.1.3 Using the cloud

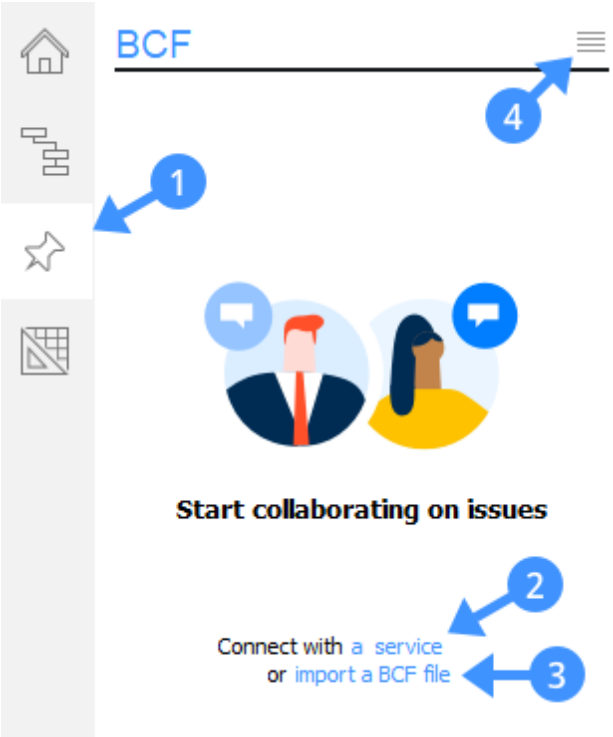
It is generally easier to use cloud services to manage BCF issues. Currently, BricsCAD BIM supports connection to three such services: **BIMcollab**, **BimSync**and **BIMtrack**. If you wish to connect to these services from within BricsCAD BIM you will need an account of these services first. These allow you to create issues and manage them in real-time from anywhere.

Note that it is not possible to **create** issues within BricsCAD BIM. The BCF Panel allows you to connect to these services from within BricsCAD BIM, log into your account and add comments, screenshots or change statuses of existing issues.

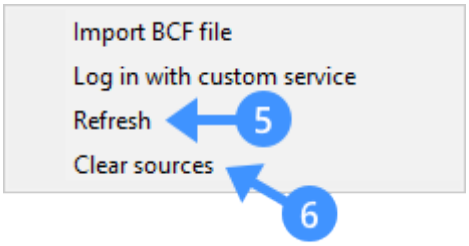
20.6.1.4 The BCF panel

The **BCF** panel can be opened by clicking the board pin icon on the left of your screen (1). If the icon is not there, right-click an empty space in the ribbon and under **Panels**, enable the BIM **BCF** panel.

Using the panel, you can either **Connect with a service** (2) or **import a BCF file** (3), as stated above. The hamburger menu at the top (4) gives you the same options, as well as refreshing the panel and clearing the sources.



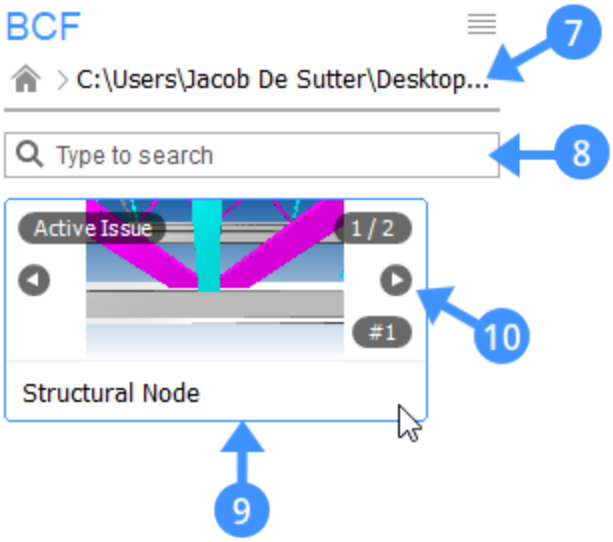
Refreshing the panel (5) will make sure you see the latest updates in case you are working on the cloud. **Clearing the sources** (6) will disconnect you with the cloud service in case you are online or close the .bcf file in case you are working on an imported .bcf file.



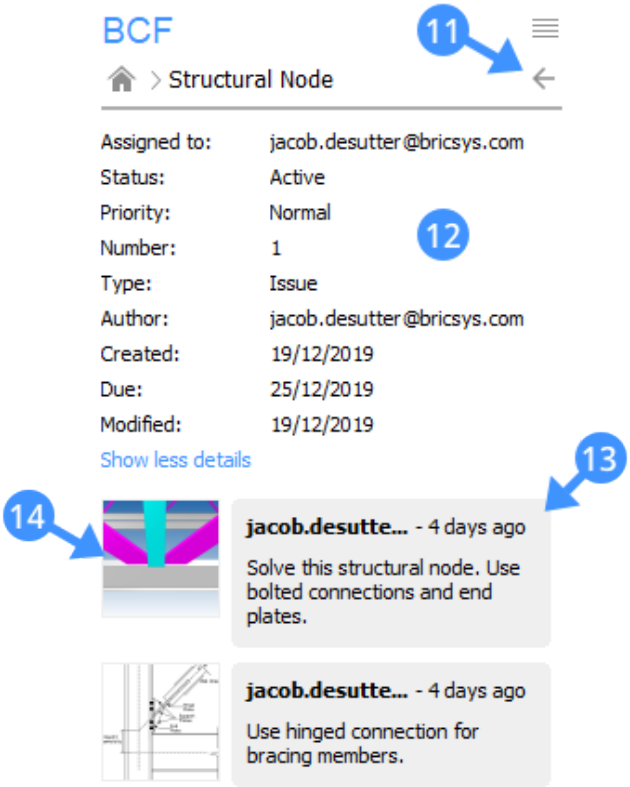
20.6.1.5 Procedure: working with the BCF panel

The following example will be shown with the files attached at the end of this article. If you want to follow along, open **West_str.dwg** and import **Structural_node.bcf**.

When importing a .bcf file you will see the path of the currently opened file (7), a search bar that can be used to filter on issues (8) and a list of issues. In the case of this imported .bcf file, there is only one issue (9) with two associated views (10).



Click on the issue to open it. When opening an issue, you will see a button to return to the issue overview (11), some more detailed information about the issue such as status, creation date and so on (12) and different comments on this issue (13). If this comment has an associated camera position, then clicking the thumbnail image (14) will take the camera in the current drawing to this position. This is useful if you want to know where exactly the issue is and you don't want to manually search through the model.

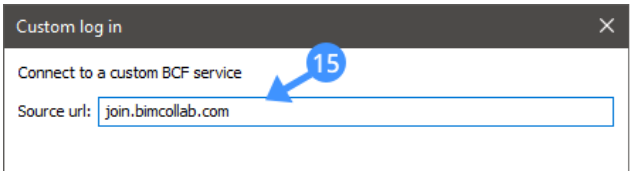


When using a cloud service, you are presented with some more options, such as setting statuses and adding comments and screenshots.

To connect to a cloud service, click the **Connect with a service** button (2). You are prompted to enter a **Source URL** (15). The correct Source URL's for the respective services can be found in the table below.

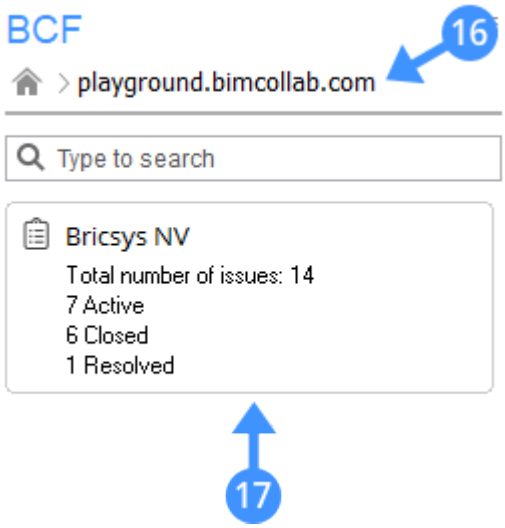
Cloud service	Source URL
BIMcollab	*<space name>*.bimcollab.com *
BimSync	bcf.bimsync.com
BIMtrack	bcfrestapi.bimtrackapp.co

* Note that the Source URL for BIMcollab includes the name of the space you bought. If you are using a free account, use *join.bimcollab.com* instead.

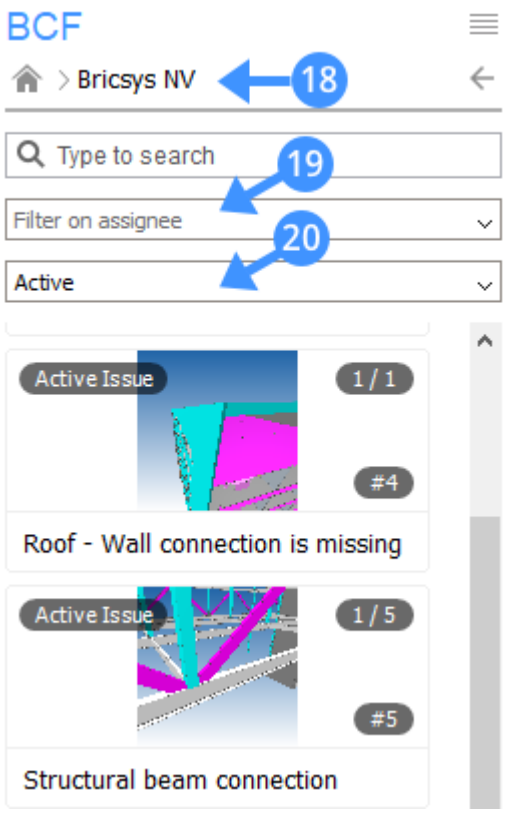


The following examples will be shown on an internal cloud space so it will not be possible to follow along with these steps exactly. It is possible however to create a free account on, for example, BIMcollab. Clicking the **Log In** button should take you to a login page of the chosen service.

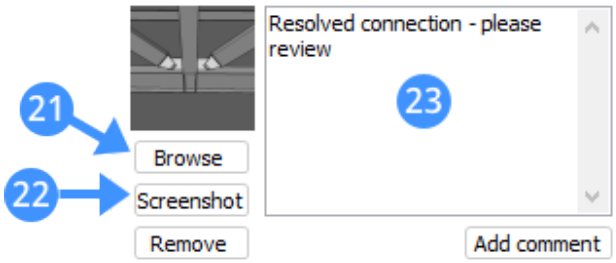
After logging in, the source URL should be shown (16), along with a list of projects that you are involved in (17). In this example, only one project is shown with 14 issues. Left-clicking the project will open it and show a list of the issues, similar to what we saw when importing the .bcf file.



At the top of the panel, the current project is displayed (18). Note that now it is also possible to filter the issues on assignee (19) and status (20). This makes it easy to quickly find the issues that are relevant to you.



Left-clicking an issue will open it and you will see similar options as before: detailed information about the issue (12) and comments on this issue (13). However, note that at the bottom it is also possible to add bitmap images (21) or a screenshot from model space (22). You can add some textual comments as well (23).



Structural_node.bcf

West_Str.dwg

20.6.2 Bricsys 24/7 panel

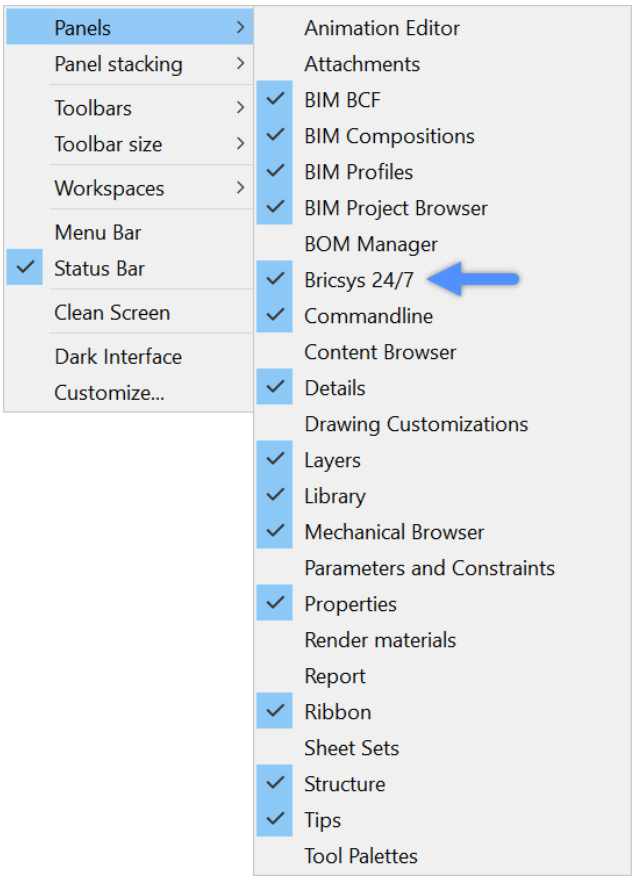
20.6.2.1 About Bricsys 24/7

Bricsys 24/7 helps team members to find and share their drawings. Bricsys 24/7 makes project management, a collaboration between project teams, cloud computing security, and task automation possible. A big advantage of this cloud-based service is that BricsCAD is directly linked to it by the Bricsys 24/7 Panel.

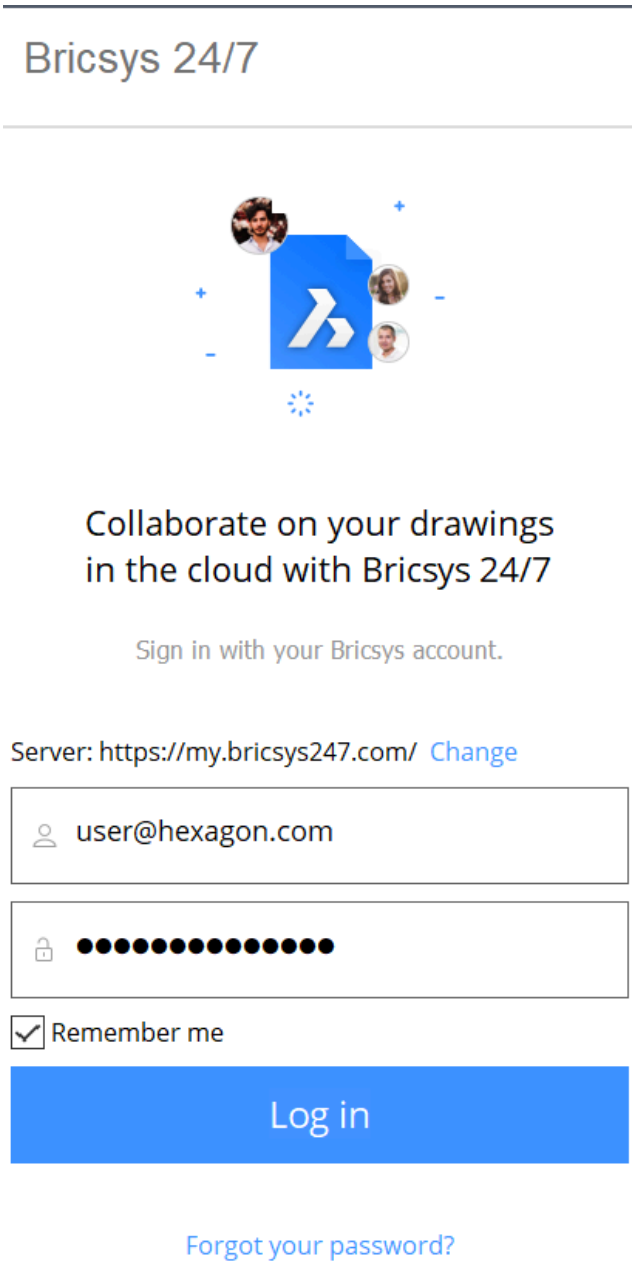
20.6.2.2 Use the Bricsys 24/7 panel in BricsCAD BIM

1 Open the **Bricsys 24/7** panel by clicking on the Bricsys 24/7 icon in the **Tool** panel on the right-hand side of the workspace.

Nota: If you do not find this icon in the **Tool** panel, right-click in the **Tool** panel and check **Bricsys 24/7** in the drop-down menu.



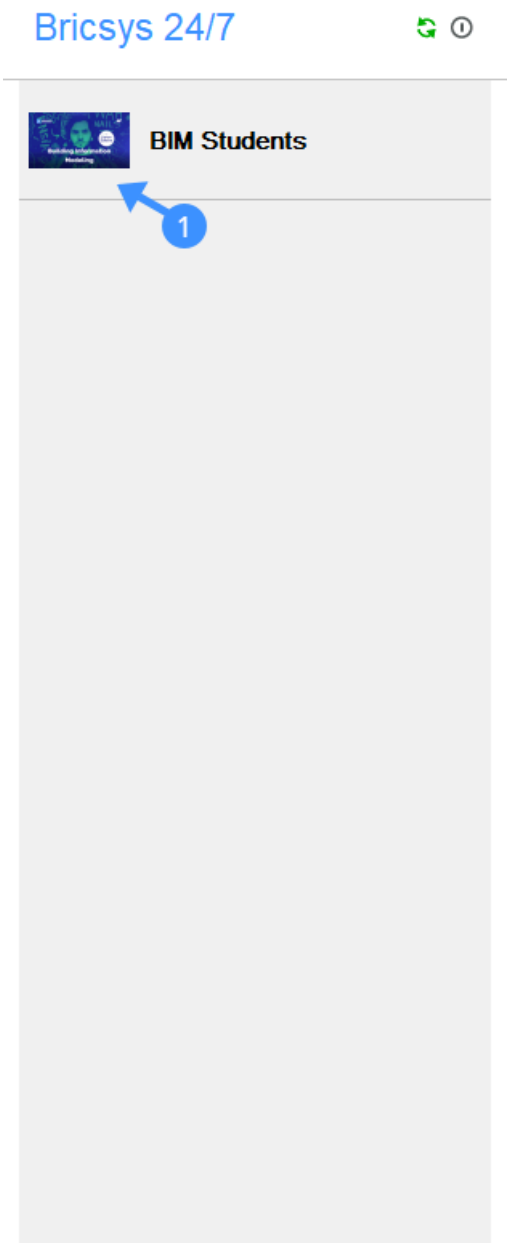
If you are logged out of your 24/7 account a screen will appear:

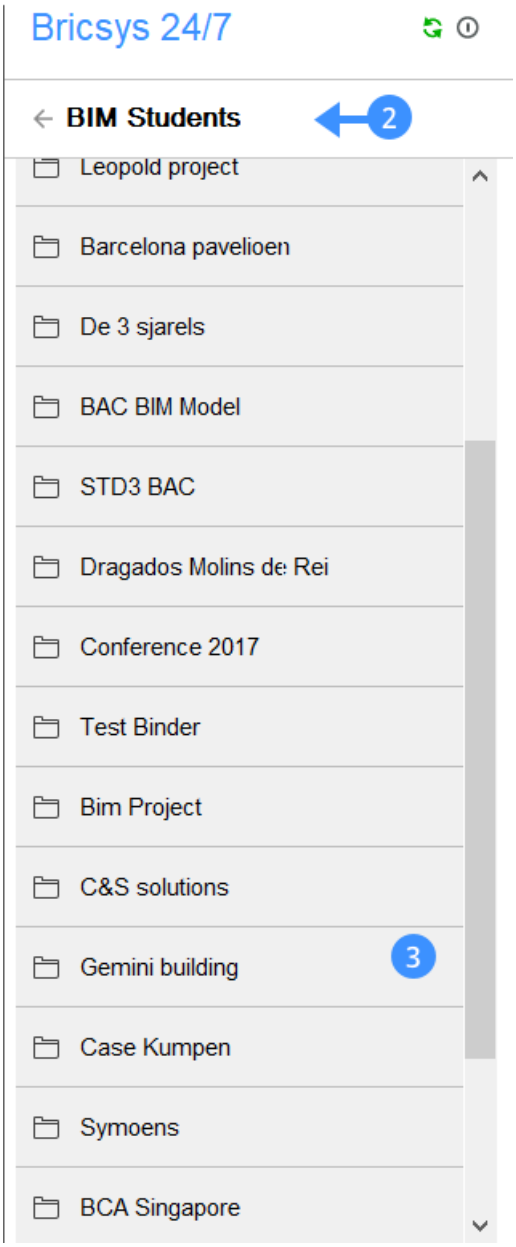


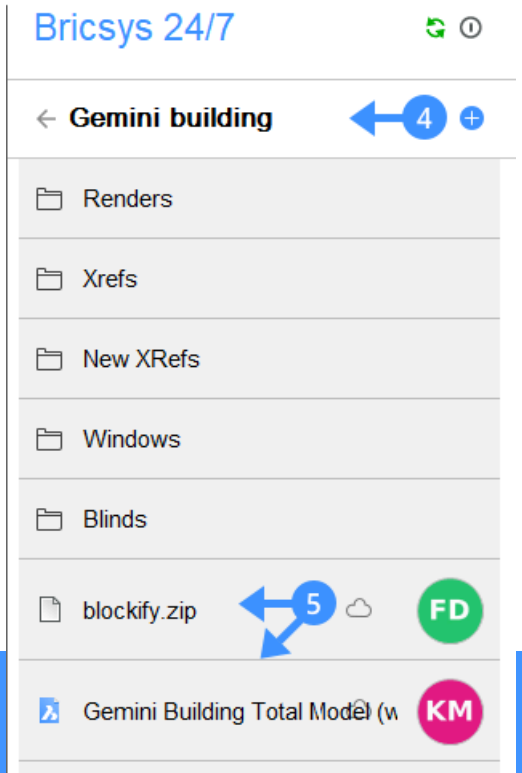
2 Enter your email address and password and press the **Log in** button.

3 (Option) Tick the **Remember Me** checkbox to store username and password.

4 Once logged on you will see an overview of your 'in the cloud' Projects, Folders, and Documents.

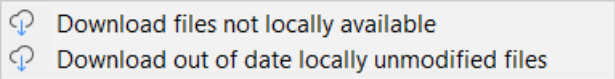

Bricsys 24/7
BIM Students
1


Bricsys 24/7
BIM Students
2
Leopold project
Barcelona pavelioen
De 3 sjarels
BAC BIM Model
STD3 BAC
Dragados Molins de Rei
Conference 2017
Test Binder
Bim Project
C&S solutions
Gemini building
3
Case Kumpen
Symoens
BCA Singapore


Bricsys 24/7
Gemini building
4
Renders
Xrefs
New XRefs
Windows
Blinds
blockify.zip
5
FD
Gemini Building Total Model (w
KM

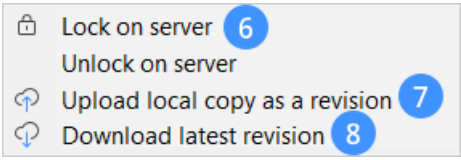
(1)	**project**	The first thing you see on the 24/7 Panel when logged on, are all the projects you are assigned to.
(2)		At the top of the panel you see your project. You can go back to the overview of projects by clicking on it.
(3)	**folder**	When you open a project, a list of folders is shown.
(4)		At the top of the panel you see the folder you're in. You can go back to the overview of by clicking it.
(5)	**documents**	In a folder you find documents. On the right-hand icons are displayed: JW 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. JW The document is locked by someone, another person cannot change it.

Right-click on a folder to get a menu with 2 possible actions:



You can download a BIM model from your Bricsys 24/7 project to a local folder.

5 Right-click on a document to get a menu with possible document actions:



(6)	Lock the document so no changes can be made.
(7)	Upload your drawing as a revision of the original drawing.
(8)	Download the latest revision.

20.6.3 IFC import and export in BricsCAD

20.6.3.1 Commands

XREF, XATTACH, XOPEN, XCLIP

20.6.3.2 About

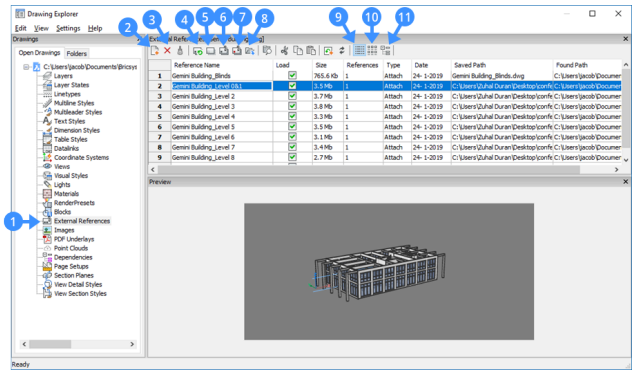
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models.

Nota: You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading.

20.6.3.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.

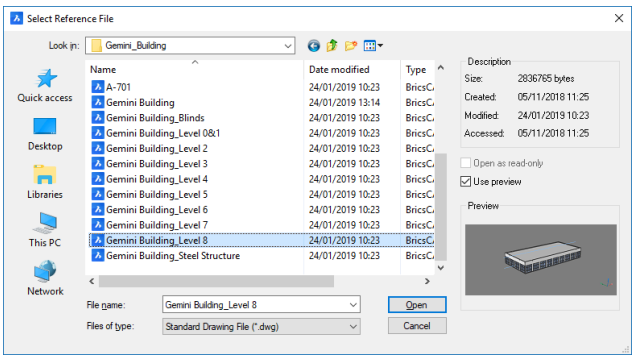


External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model.
Unload XRef (5)	Unloads the selected XRef. This hides the XRef from the view.
Bind XRef (6)	Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer.
Insert XRef (7)	Converts XRefs into blocks.
Open XRef (8)	Opens the referenced drawing.
Detail view (9)	Displays the details of XRefs.
Icon view (10)	Displays icons of the XRefs.
Tree view (11)	Displays the relationship between XRefs in a tree-like view.

2 To attach externally referenced models, click the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

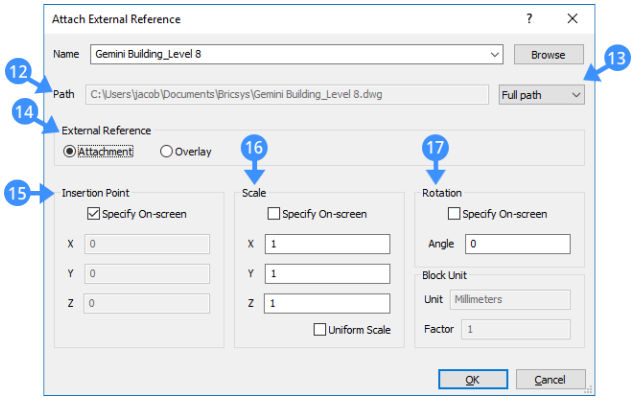
The **Select Reference File** dialog box displays.

The following illustration shows the **Select Reference File** dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open**.



Next, the **Attach External Reference** dialog box displays.

The following illustration shows the **Attach External Reference** dialog box. The table below presents each option of the **Attach External Reference** dialog box. From this dialog box, select the options you want to apply, and then click **OK**.



Path field (12)	Displays the full path of the selected drawing.
Path type (13)	Includes **Full path**, **Relative path**, and **No path** options. **Full path**: When you move your drawing containing an XRef to another location, BricsCAD will still search for the XRef in the previous location and it will not find this XRef. **Relative path**: When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one.

External Reference type (14)	Includes **Attachment** and **Overlay** reference types. **Attachment**: Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef "X" and this contains nested XRefs "Y" and "Z". When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing. **Overlay**: Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references.
Insertion Point (15)	Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields.
Scale (16)	Allows the user to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field.
Rotation (17)	Allows the user to specify the Rotation angle on the screen or in the **Angle** field.

Note that the **BIM XReference** option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step.

The following illustration shows the BIM category in the **properties** panel after attaching the model that has been classified as an XRef.

3 To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the **Drawing Explorer** dialog box.

The selected XRefs are detached.

Nota: Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

4 Do one of the following to unload and reload an existing XRef.

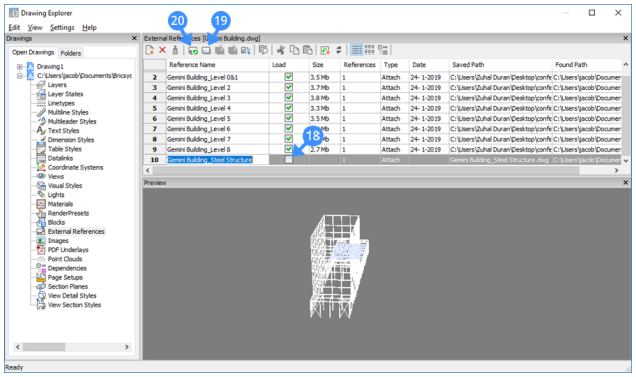
- Click the **Load** (18) column of a loaded XRef.

  This XRef is unloaded in your main model.

  To reload the XRef check the load box of this XRef.

- Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar.
- Select the unloaded XRef, then click Reload XRef (20) tool button.

  The following illustration shows the **Load** column (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.



5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **Attach** to **Overlay** or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6 Click **X** to close the dialog box. Notice that the changes that have been made appear after you close the **Drawing Explorer**.

20.6.4 Import and underlay Autodesk® Revit® files in BricsCAD

20.6.4.1 Commands

IMPORT

20.6.4.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

20.6.4.3 About Revit import

The Revit import function allows you to import .RVT files into BricsCAD as 3D solids or as a geometry underlay for referencing purposes. However, any project data and model metadata, i.e. Constraints, Wall Type structure assembly, materials etc., from the Revit model are not imported.

You can also choose to import selected geometries by 3D Views, by Family Categories, by Levels or by View Worksets, depending on how the Revit project file has been set out.

If you import the RVT file as a solid, only the geometry will be imported with no BIM metadata attached to it. The solids can later be classified and edited with direct modelling tools in BricsCAD.

If you import the RVT file as an underlay, almost like how an XREF is used, the BricsCAD model performs faster and smoother generally. This process also takes a shorter time than importing as a solid. However, the underlay model becomes an OdRvtEntity, like a non-editable, singular block. This method is particularly

useful if you are looking to import the Revit model only for reference and coordination purposes, i.e. importing a Revit structural model to look for clashes with your architectural / MEP model within BricsCAD.

BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the IFC Import and Export in BricsCAD article.

20.6.4.4 Procedure: importing a Revit file as a solid in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **I** and Press Enter to select Import.

5 **You are prompted:** Choose elements [**A**ll/by **V**iews/by **C**ategories/by **L**evels/by **W**ork**S**ets]:

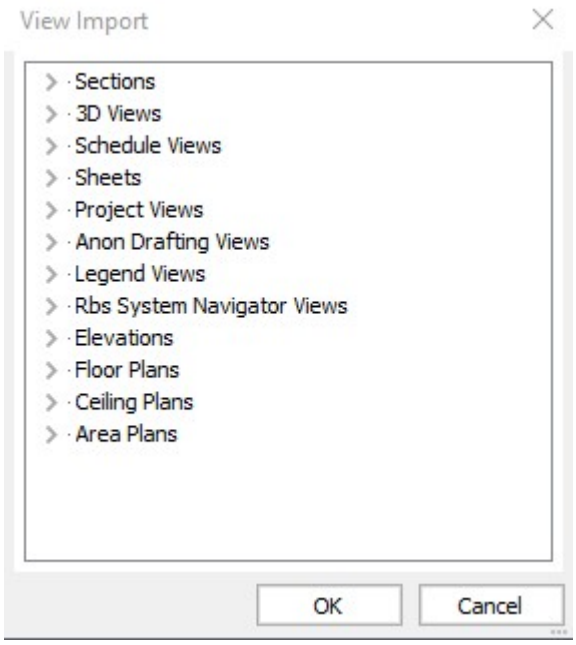
There are different filters to import a file. Select the filter you want to use by clicking into your desired option in the command-line or by hitting their respective letters in bold.

- **All**

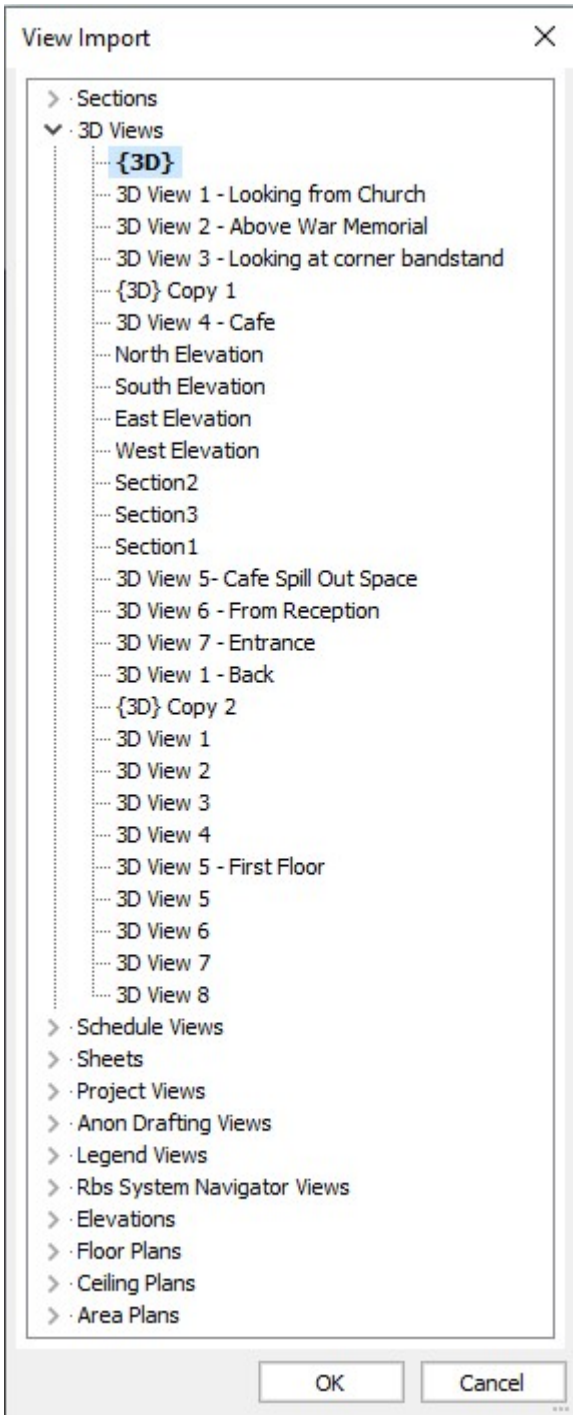
  All elements present within the RVT file will be imported.

- **By Views**

  Select a view from the **View Import** dialog box and the elements will be imported based on the view's Visibility/Graphics Overrides settings in Revit. The views are structured based on their View Types.



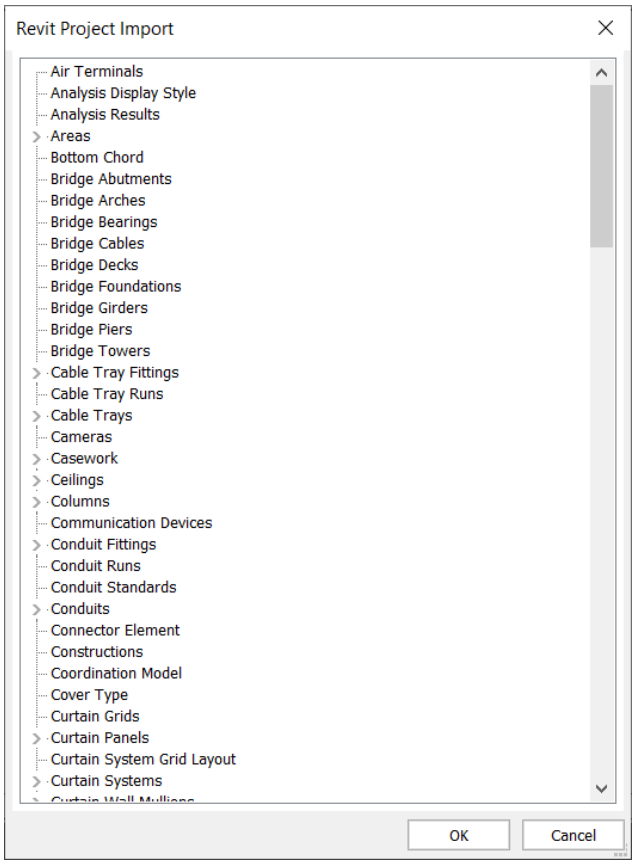
You can expand any relevant or applicable View Type category for the specific views. It could look like this, if the **3D Views** category is expanded:



Nota: Importing elements via a 2D view, ie. Floor Plans etc., only brings elements that are made visible in the Visibility/Graphic Overrides in the Revit model settings for that particular view. It does not necessarily bring elements hosted onto its corresponding level, if they are made hidden on the plan view.

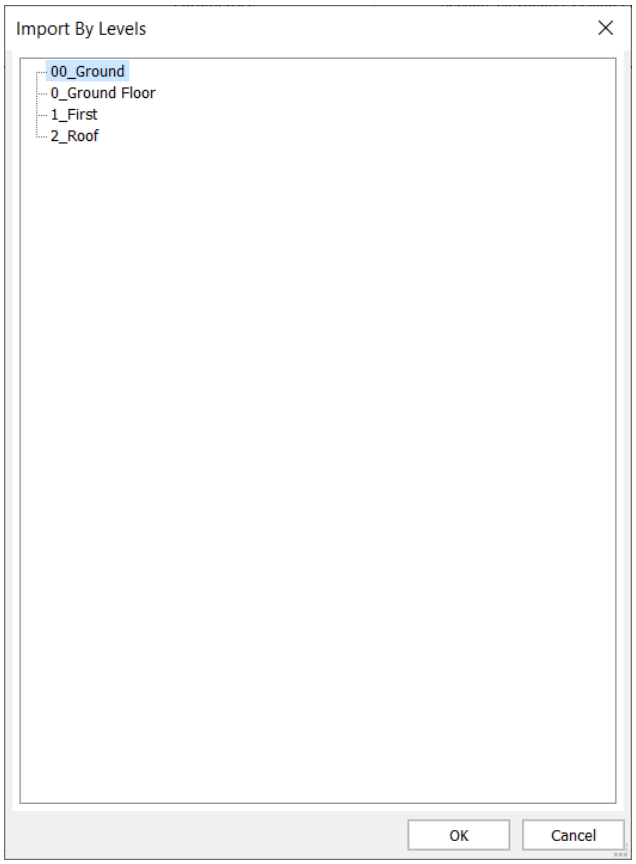
- **By Categories**

  You can select one or multiple built-in Categories and Element Types from the **Revit Project Import** dialog box. Elements of those chosen categories and element types will be imported.



- **By Levels**

  You can select one or multiple levels from the **Import By Levels** dialog box. Elements hosted to the chosen levels will be imported.

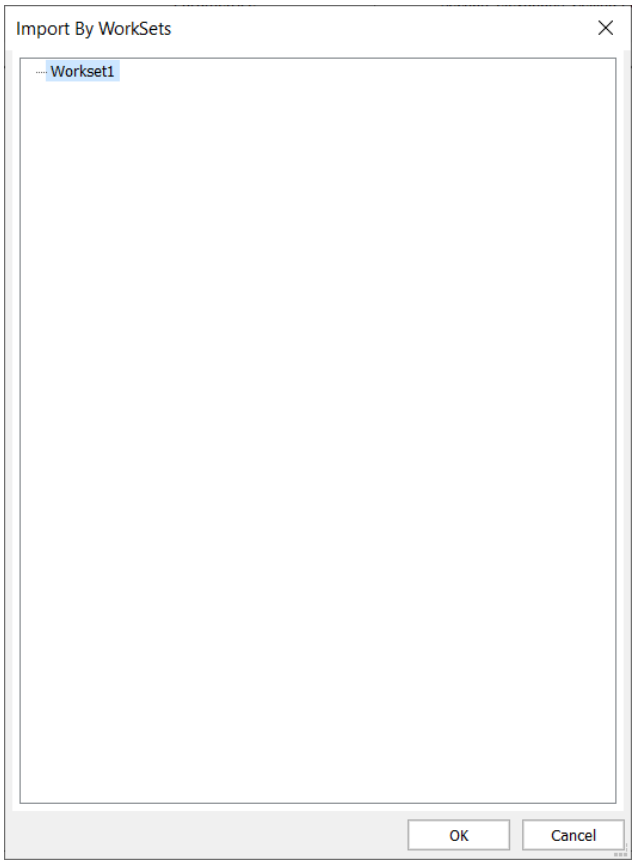


- **By WorkSets**

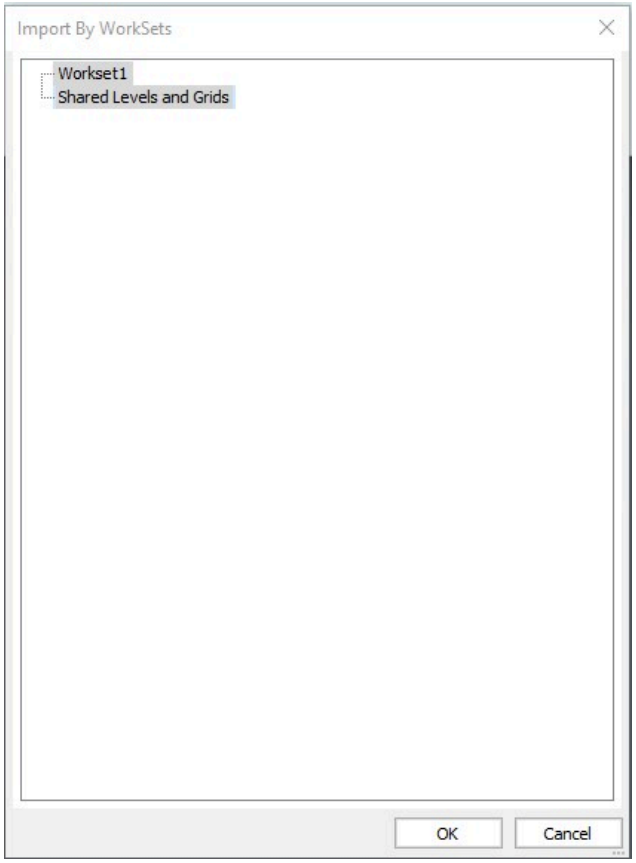
  You can select one or multiple worksets from the **Import By Worksets** dialog box.

  This method of importing works best on a workshared Revit model. It is recommended you import a Central Model to ensure you are importing the latest information in the worksets. Detaching the Central Model is optional.

  If you import a non-workshared Revit model, only **Workset1** is available.



If you import a workshared Revit model, either Central or Local, the User-Created worksets within the model will be available in the dialog. The elements which belong to this workset will be imported.



Nota: The **Shared Levels** and Grids and **Workset1** worksets are created in Revit by default when Worksharing is enabled and can be managed by the Revit user.

6 If you have chosen to import by view, you do not have to specify the level of detail. The Detail Level of the selected view set in Revit will be used.

7 For the rest of the other options,

8 **you are prompted:** Import Elements with Detail Level [Fine/Medium/Coarse]:

Select the level of detail you want. Type **F** in the command-line for fine, **M** for medium and **C** for coarse and press Enter.

Nota: When selecting the level of detail, make sure you are aware of how the incoming geometries have been set out in terms of Detail Levels within the Family Elements Visibility Settings in Revit.

9 Select the View, Categories, Levels or WorkSets you want to import and click **OK**. The RVT file will be imported as 3D solid geometries.

10 Now, you can classify the solid manually or automatically and/or further edit it with direct modeling tools in BricsCAD.

Nota: Some of Revit's system families, such as walls and floors etc., are imported into BricsCAD as 3D solids, where you can further manipulate their geometries directly. This also applies to Model-In-Place components and Element Types, such as Fascias, that can be defined geometrically by a Profile.

However, other system families designed with in-built constraints, i.e. stairs and curtain walls etc.

are imported into BricsCAD as separate Block References. For eg., Curtain Panecl and Curtain Wall Mullion Types, which make up a Curtain Wall Type in Revit, will exist as separate Block References. This also applies to all other loadable families, like furniture, placed within the Revit model. For more information, please visit the article on Working with Blocks.

20.6.4.5 Procedure: importing a Revit file as an underlay in BricsCAD

1 Open a new BricsCAD file.

2 Type **Import** in the command-line and press Enter **or** click the BricsCAD icon in the ribbon and select **Import** to launch the Import command.

3 The **Import File** dialog box pops-up. Select Revit Project (*.rvt) in **Files of type**, select the file you want to import from your folder and click **Open**.

4 **You are prompted:** Choose import mode as [Import/Underlay]:

Type **U** and Press Enter to select **Underlay**.

5 Follow Steps 5-7 of the 'Procedure: importing a Revit file as a solid in BricsCAD' above.

6 **You are prompted**: Deviation value:

By default, the deviation value is 0.001. Press Enter to accept **or** type a new value and press Enter.

Nota: The deviation value defines the accuracy of the imported model. A lower value will result in a finer geometry, but it will take longer time to load.

7 The RVT file is now imported as a non-editable OdRvtEntity.

20.6.4.6 Procedure: importing a Revit family in BricsCAD

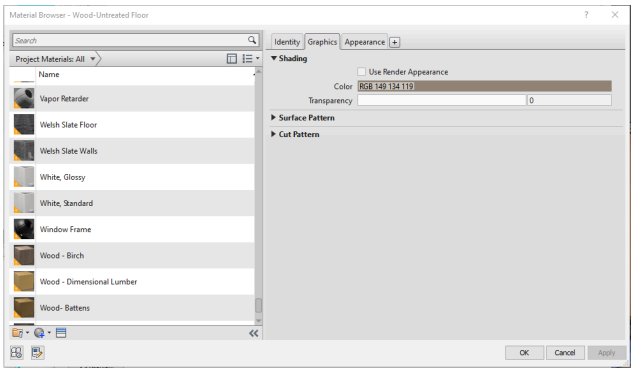
Please refer to the article **Importing Revit RFA Families in BricsCAD** for more information.

20.6.4.7 Texture and graphics of the imported geometries

Although project information and element metadata in a Revit model cannot be brought into BricsCAD (just yet), there are some aspects of the imported 3D solids, i.e. Material Color, Transparency, which can be brought over and therefore correspond to the original graphics set in the Revit.

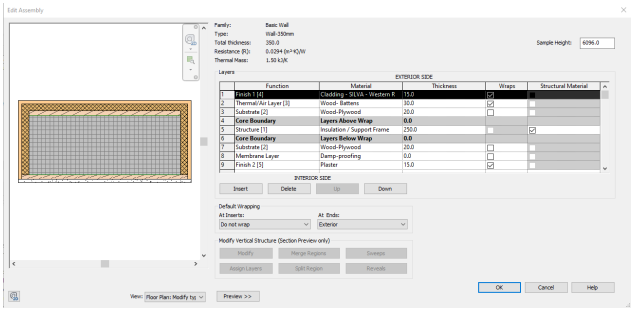
To retain the consistency of the graphical information during the import process, ensure your RGB color and transparency settings set in Revit's Material Browser are set to your desired values.

In the example below, the values taken will be **RGB 149 134 119** and **0** (%) for the colors and transparency values respectively.



If the imported element is a Revit System Family with the ability to define its structure assembly, i.e. Wall Type, and is made up of various layers of materials or functions, BricsCAD applies the color template values from the outermost materials (on both the Exterior and Interior sides) accordingly on to the corresponding geometry's faces.

Let's take an example from the illustration below, the values will be derived from the materials assigned on Layers 1 and 9, which are **Cladding...** and **Plaster** respectively.



Otherwise, BricsCAD applies the color value, and the transparency settings if applicable, to the 3D solid replacing the respective parts of the Revit Family. This also applies to those Loadable Families and Components, having loaded into the imported Revit project.

20.6.5 Importing Autodesk® Revit® families in BricsCAD

20.6.5.1 Commands

IMPORT, BMINSERT

20.6.5.2 Disclaimer

Unless otherwise indicated, the word "Revit" used in this article refers to the Revit® software and its peripherals by Autodesk®.

20.6.5.3 Important notification before getting started

RFA import is a work-in-progress. See the BMINSERT command.

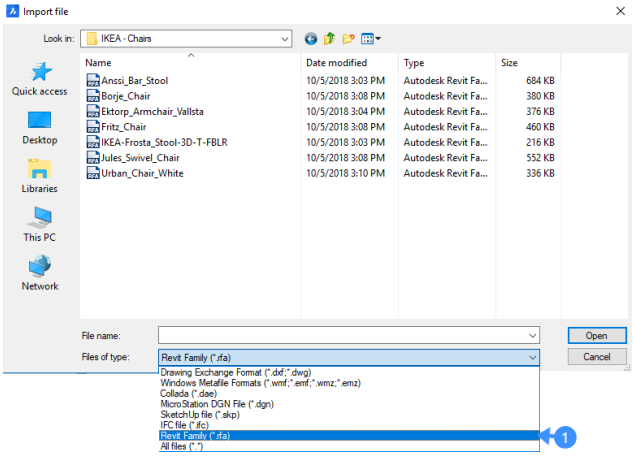
BricsCAD V21 currently supports the import of **Revit 2015-2020** files only. If you would like to import a model created in a later version of Revit, please consider importing the model into BricsCAD as an IFC file. For more information, please visit the **IFC import and export in BricsCAD** topic.

20.6.5.4 Importing RFA family files

1 Launch the Import command.

1 The **Import file** dialog box displays.

2 Select **Revit Family (*.rfa file)** (1) in the **Files of type** list.



3 Select the .rfa file you want to import.

4 Click **Open** to import the Revit Family.

When the RFA family file is imported the geometry from this file is included in BricsCAD as separate 3D solids.

20.6.5.5 Procedure: inserting RFA family files

Nota:

- When DUCS = ON (Dynamic UCS) the RFA entity aligns with the 3D solid face under the cursor.
- When DYN = ON (Dynamic Dimensions) the distances from the nearest parallel edge on the face of the 3D solid display.

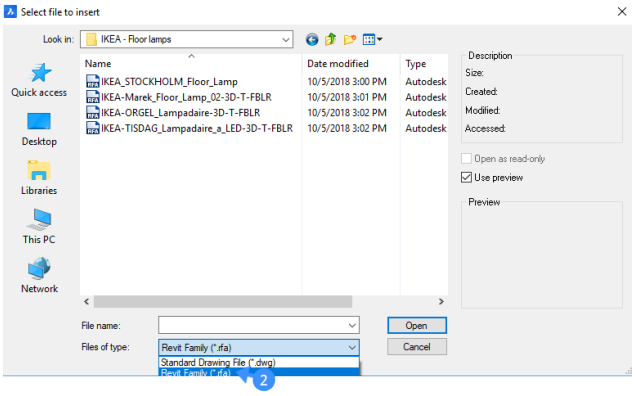
Using BmInsert

1 Launch the BMINSERT command.

1 A **File** dialog box appears.

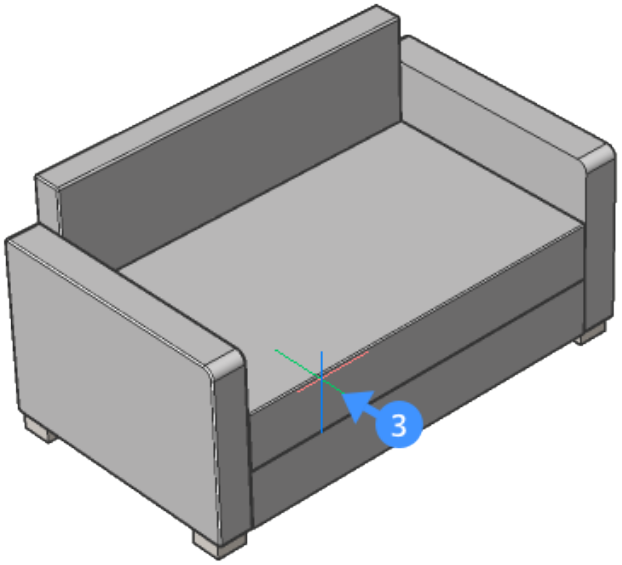
2 Browse to the folder where your .rfa family files are saved.

3 Select **Revit Family (*.rfa file)** (2) in the **Files of type** list.

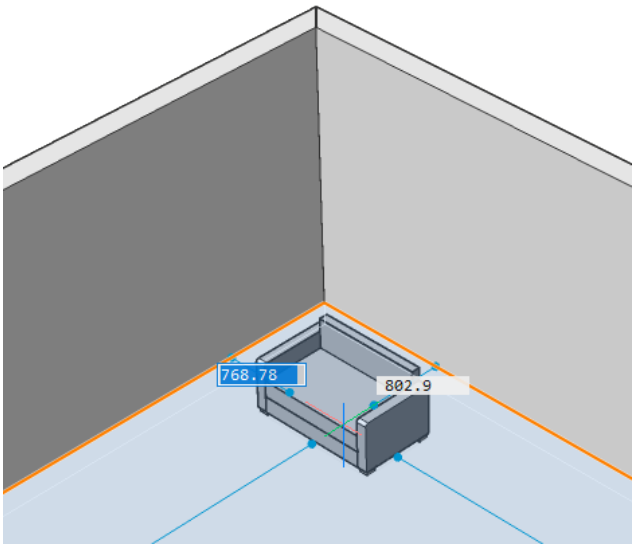


4 Select the .rfa file you want to import.

5 The selected file is attached to the cursor (3).



The following illustration shows inserting a Revit family sofa when both DYN and DUCS are on.

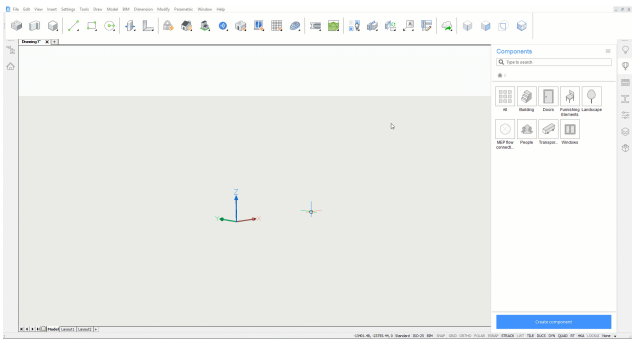


6 Do the following to place the selected RFA family file using the dynamic dimension fields:
   - Hit the Tab key to select one of the dynamic dimension fields, then type a value and hit the Tab key to lock the dimension. The locked dimension will turn red.
   - Repeat the previous step to lock another dynamic dimension field.
   - Optionally hit the Tab key to adjust the dynamic dimensions.
   - Press Enter to accept the current position.

Using the Library panel

1 To insert a RFA family folder in the **Library** panel, add the RFA root folder path to the Library directory path in the **Settings** dialog box. The content of the root folder and all subfolders will be available in the **Library** panel.

   The following animated gif shows how to add the RFA root folder path to the components directory path.



   The RFA family files are ready to insert in the model space.

   If necessary, choose to **Generate Thumbnails** in the hamburger menu on the **Library** panel.

2 Drag and drop the RFA file from the **Library** panel into your model space.

20.6.6 Interference checking

20.6.6.1 Commands

INTERFERE

20.6.6.2 About interference checking

The **Interference Checking** tool is used during the modeling phase to find out conflicts in the model by comparing 3D solid models of building elements. Interference occurs when two 3D entities overlap in 3D space. In BricsCAD BIM, the Interfere tool helps you to find intersections between the entities that you have selected. You can also do the interference checking for all elements in your model at a time. Optionally, the tool creates interference solids after running the command. The newly created solids are placed on their own layer, as specified by the **InterfereLayer** variable. Running an interference check on existing models allows you to prevent the conflicts in your entire model.

For more information about this command, visit the Command Reference article INTERFERE.

20.6.6.3 Procedure: performing an interference check

1 Select the desired 3D solids to be checked for interference, then launch the INTERFERE command.

Nota: Multiple solids can be selected using the selection box or in the Structure Browser, depending on whether the selected entities appear when selected in the tree.

The first set of entities is defined.

The Command line displays how many entities in total are set for interference checking e.g. Entities in set: 20.

Note that the Nested Selection option allows you to select 3D solid models that are inside blocks and external references (XRef) and once they are selected you can compare them against other entities in the selection set.

2 Select the 3D solids in the model to add them to the second set of entities.

3 The entities in the first set are compared with the entities in the second set.

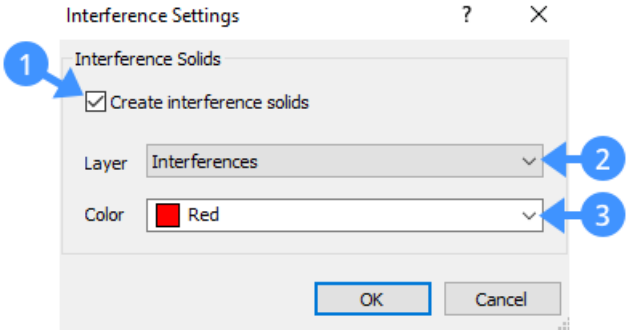
Nota: If you press the **Enter** key without selecting any solids for defining the second set of entities, the entities in the first set will be compared to each other.

4 Press **Enter** to run the interference check.

5 The interference volume is created for each collision and it is shown in a solid model.

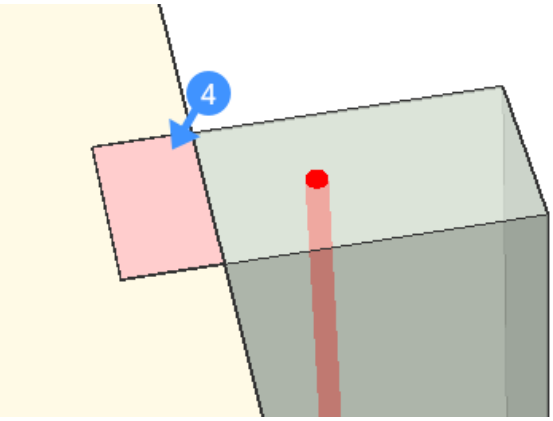
Nota: The solids of the interference volume are displayed in red since they are placed on layer **Interferences**. The default settings on the Interference Checking tool makes it easy to find the intersecting solids in the model. This setting can be changed using the prompt menu or the Command line. To open it, click the **Settings** option in the prompt menu or type "**S**" while the interfere command is still active. The **Interference Settings** dialog box displays. The dialog box allows you to configure interference settings, which are stored in variables.

The following illustration shows the Interference Settings dialog box. The table below presents each component of the dialog box.



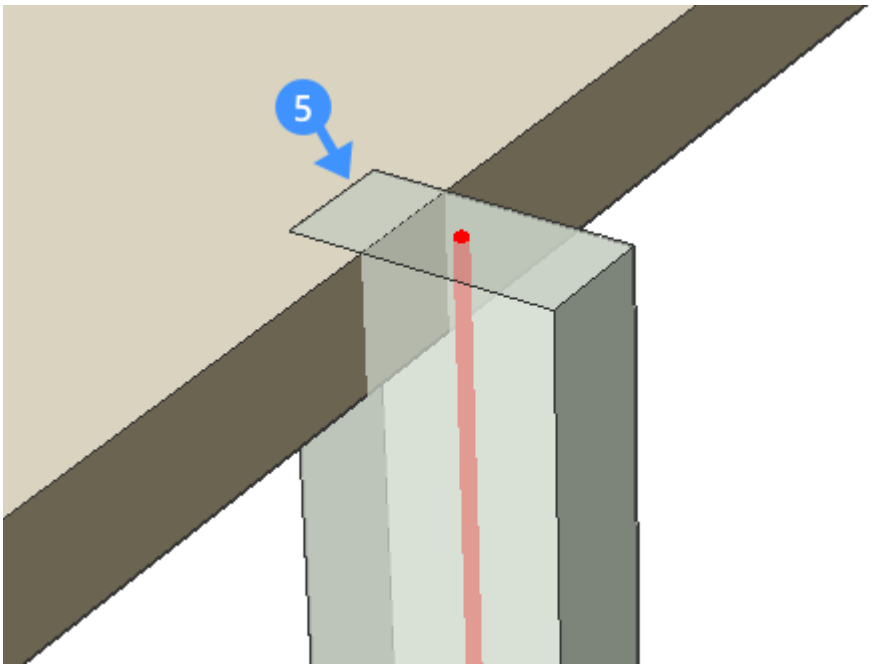
Create interference solids box (1)	Determines whether a solid or region of the interference is created. Check to create a solid of the interference volume or region of the interference area.
Layer (2)	Shows the current layer for interferences.
Color (3)	Shows the current layer color.

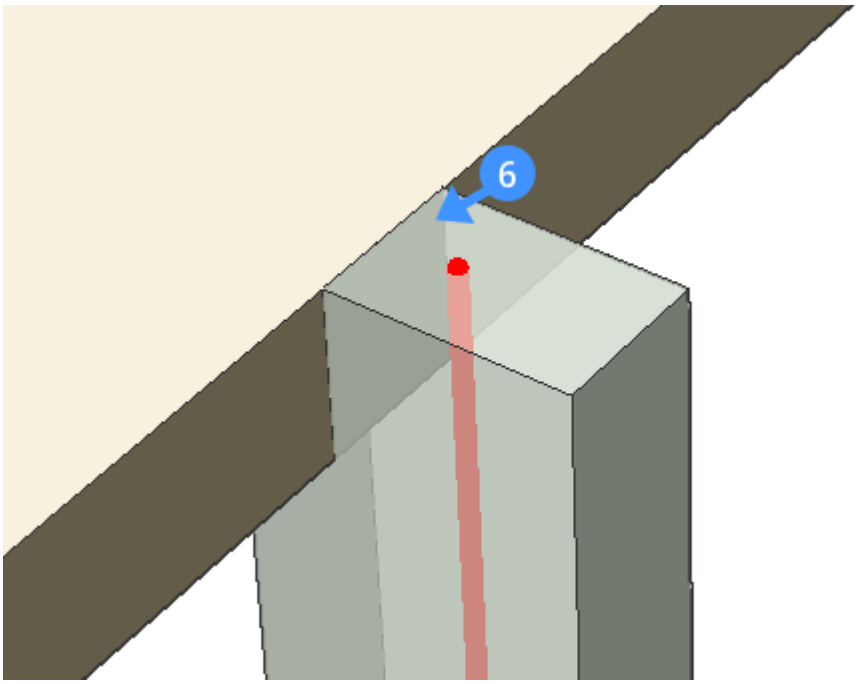
The following illustration shows the detected collision (4) between the floor slab and column.



6 To remove the undesired intersections, turn off the **Interferences** layer, and use one of the 3D solid modifying tools e.g. **BIM Drag** or **Subtract** on the interfering solid. This process will let you make the necessary corrections in your model.

In the following illustration, the created solid of the interference volume is shown (5). In the second image, the intersection is fixed by subtracting the slab geometry from the column.





7 Run the INTERFERE command a second time to check whether all interferences are fixed.

8 If there are no collisions left, the Command line reports that "**Solids do not interfere**".

20.6.7 Using Xrefs

20.6.7.1 Commands

XREF, XATTACH, XOPEN, XCLIP

20.6.7.2 About

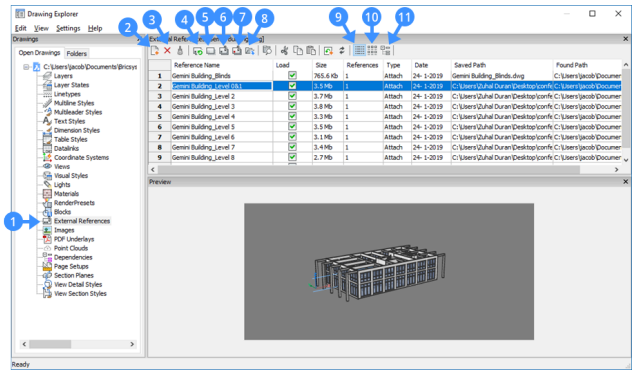
Using external references or Xrefs allows you to insert any model in the current BIM model. Instead of copying the model, only a reference to the source model is saved in the master model. You can attach multiple models to a single master model without increasing the size of the current model significantly. Further advantages of using the Xrefs are that the source models can be edited simultaneously by different members of the design team, and the master model can always be updated to show the latest version of each of the source models.

Nota: You can distinguish between XRef entities and entities in the current drawing using the XDWGFADECTL system variable. This system variable allows to fade the display of XRefs. You can give the values between 0 (no fading) and 90 to change the default value of the fading.

20.6.7.3 Procedure: managing XRefs in the current drawing

1 Open the **Drawing Explorer** dialog box and click the **External References** section.

The XRef preview pane appears, allowing the user to attach a model or manage the existing XRefs on the list. The following illustration shows the XRef preview pane.



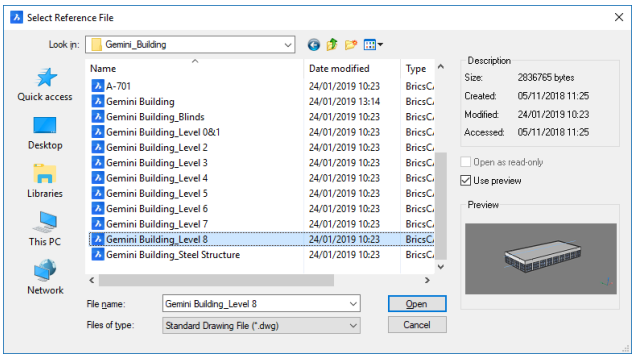
External References(1)	Shows the External References section.
Attach XRef (2)	Attaches a model as an external reference.
Detach XRef (3)	Removes the selected XRef completely from the current model.
Reload XRef (4)	Reloads the selected XRef. This is useful when the source models have been changed, and you want to see the updated versions in the master model.
Unload XRef (5)	Unloads the selected XRef. This hides the XRef from the view.

Bind XRef (6)	Merges the source models permanently to the current drawing. Merging the XRef to the current drawing makes the XRef part of the drawing. References to the XRef disappear from the Drawing Explorer.
Insert XRef (7)	Converts XRefs into blocks.
Open XRef (8)	Opens the referenced drawing.
Detail view (9)	Displays the details of XRefs.
Icon view (10)	Displays icons of the XRefs.
Tree view (11)	Displays the relationship between XRefs in a tree-like view.

2 To attach externally referenced models, click the **Attach XRef** tool button on the **Drawing Explorer** dialog box.

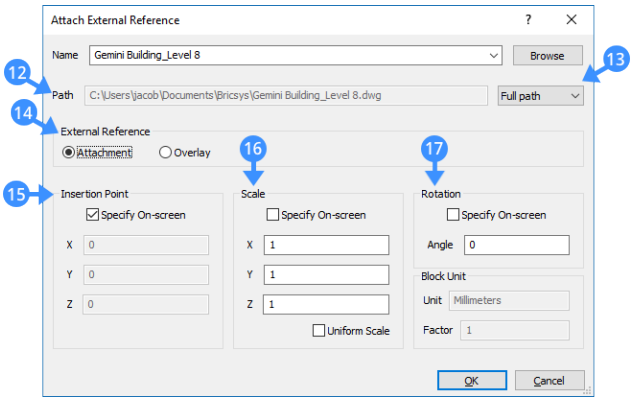
The **Select Reference File** dialog box displays.

The following illustration shows the **Select Reference File** dialog box. From this dialog box, select the DWG file you wish to attach in your main model, and then click **Open**.



Next, the **Attach External Reference** dialog box displays.

The following illustration shows the **Attach External Reference** dialog box. The table below presents each option of the **Attach External Reference** dialog box. From this dialog box, select the options you want to apply, and then click **OK**.



Path field (12)	Displays the full path of the selected drawing.
Path type (13)	Includes **Full path**, **Relative path**, and **No path** options. **Full path**: When you move your drawing containing an XRef to another location, BricsCAD will still search for the XRef in the previous location and it will not find this XRef. **Relative path**: When you move your drawing containing an XRef to another location, the new file path will be created in the new location. The file folder will set up correctly using the same folder structure as the first one.
External Reference type (14)	Includes **Attachment** and **Overlay** reference types. **Attachment**: Attaches all XRefs, including nested XRefs. For example, you have the first level of XRef “X” and this contains nested XRefs “Y” and “Z”. When the attachment option is selected the first level of XRef and its nested XRefs Y and Z will be attached to the drawing. **Overlay**: Attaches only the first level of xref (X), nested xrefs (Y, Z) will be not loaded. This can also be used to prevent circular references.
Insertion Point (15)	Allows the user to specify the Insertion point on the screen or in the X-, Y-, Z- fields.
Scale (16)	Allows the user to specify the Scale on the screen or in the X-, Y-, Z- fields. Check the **Uniform Scale** option to specify the scale in the X- field.

Rotation (17)	Allows the user to specify the Rotation angle on the screen or in the **Angle** field.

Note that the **BIM XReference** option from BIM Classify allows you to classify the model as an external reference. The identification of information will not explicitly be represented in the current model or in the project database. Such information appears after attaching the model to another project. To do so, follow the above step.

The following illustration shows the BIM category in the **properties** panel after attaching the model that has been classified as an XRef.

BIM	
Type	BIM XReference
Name	
Description	
Building	
Story	

3 To detach an existing XRef, select the XRef you wish to detach in your main model, and then click the **Detach XRef** tool button on the **Drawing Explorer** dialog box.

The selected XRefs are detached.

Nota: Press and hold the CTRL key to select multiple XRefs or to unselect a selected XRef.

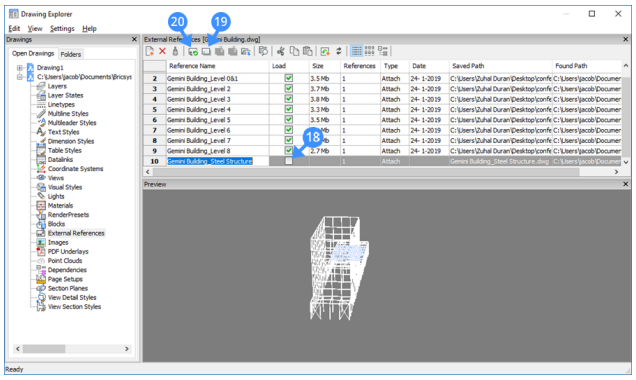
4 Do one of the following to unload and reload an existing XRef.

- Click the **Load** (18) column of a loaded XRef.

  This XRef is unloaded in your main model.

  To reload the XRef check the load box of this XRef.
- Select the XRef, then click the Unload XRef (19) tool button on the Details toolbar.
- Select the unloaded XRef, then click Reload XRef (20) tool button.

  The following illustration shows the **Load** column (18), Unload XRef (19) and Reload XRef (20) tool buttons on the **Drawing Explorer**.



5 To modify the attachment type, click the **Type** column of an existing XRef and switch from **Attach** to **Overlay** or vice versa. The same process can be followed for modifying the **Saved path** of an XRef and **Found path** of an overlay.

6 Click **X** to close the dialog box. Notice that the changes that have been made appear after you close the **Drawing Explorer**.

20.7 Documentazione della progettazione

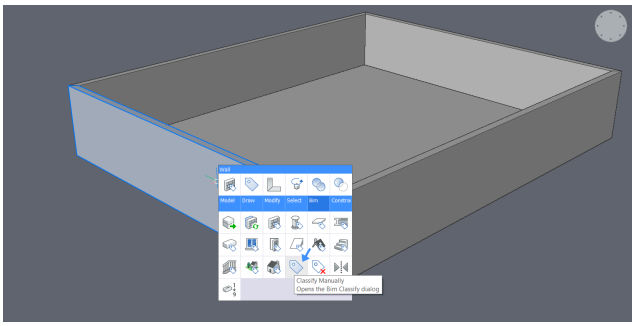
20.7.1 Annotations in 3D

20.7.1.1 Commands

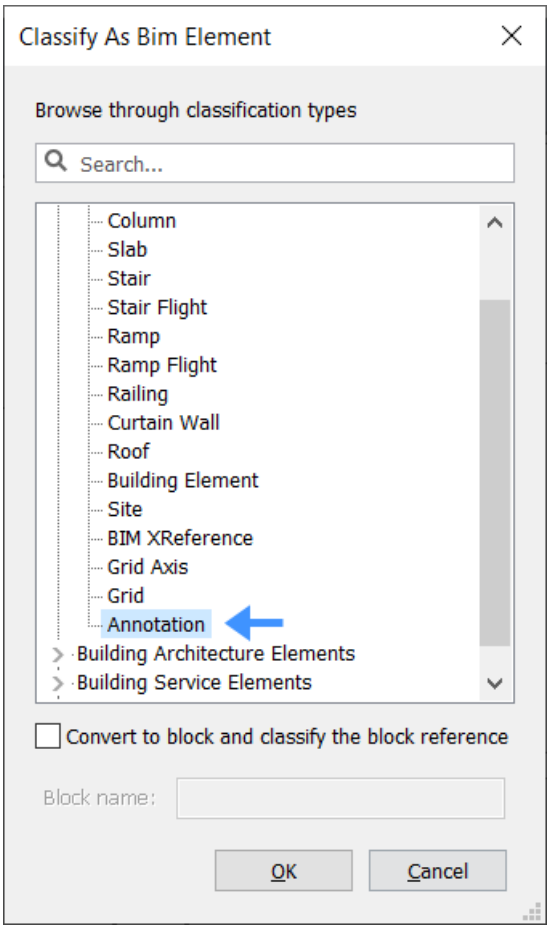
BIMTAG, BIMCLASSIFY

20.7.1.2 BIM classify as an annotation

1 Select a solid and select **Classify Manually** in the Quad or classify as other after launch the BIMCLASSIFY in the Command line:



2 Choose to classify as **Annotation** in the folder of **Building Core Elements** and click **OK**:



3 When you open the Properties panel of the selected solid you can change the **Name**, **Description**, **Building** and **Story**.

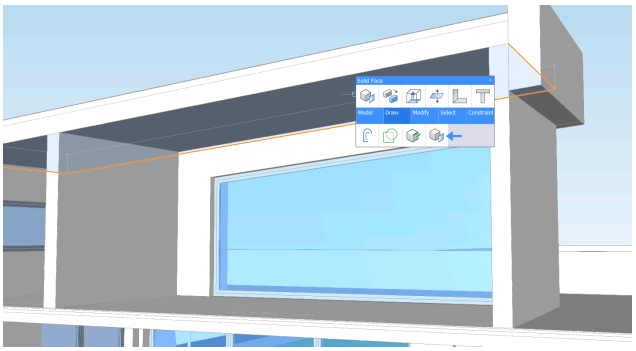
Nota: The property GUID cannot be changed because it has a unique, automatically generated string.

3D Solid	
General	
Handle	B4
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
3D Visualization	
Material	ByLayer
Mass	
BIM	
Type	Annotation
Name	
Description	
Building	Building
Story	Floor 0
GUID	3p42tSCm10YQfj1YI0xPEO

20.7.1.3 Show added information of a finishing material

In this procedure, you will add design information to a concrete roof structure.

1 Use the EXTRACTFACES command under **Draw** in the quad, to extract the face of the **Roof** entity. A region is now created.



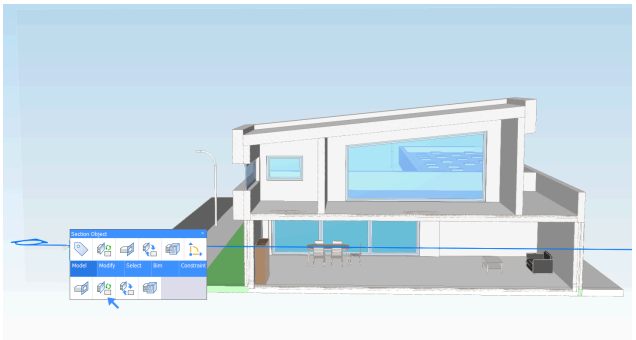
2 Classify the region as an **Annotation** as seen above.

3 Enter the desired text in the **Description** field (found in the Properties panel):

Nota: By default, the BIMTag references the **Description** property.

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

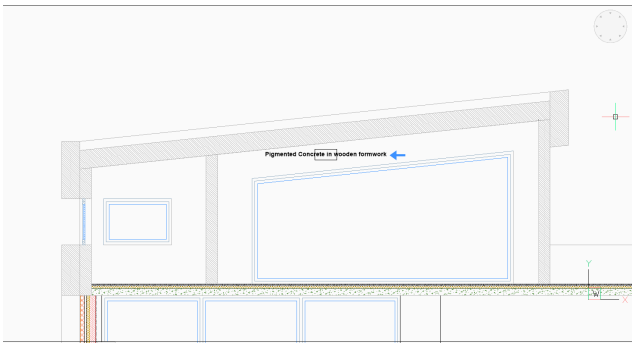
4 Make a section through the region (with BIMSECTION command) and select **Generate Section**() from the Quad:



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

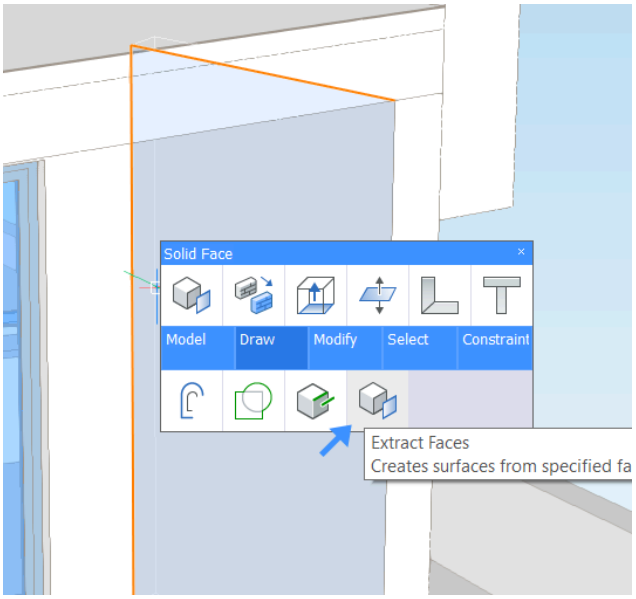
This procedure will teach you how to display other properties of an entity classified as an **Annotation** object.

Nota: The annotation tag pulls information from the **Description** property by default.

For more information about tagging, see the article BIM Tags.

1 Select the face you want to extract the information from.

2 Select **Extract Faces** from the Quad. A region is now created.



3 Classify the region as an **Annotation** as above.

4 Make a section through the solid (with BIMSECTION command) and select **Generate Section** ( ) from the Quad.

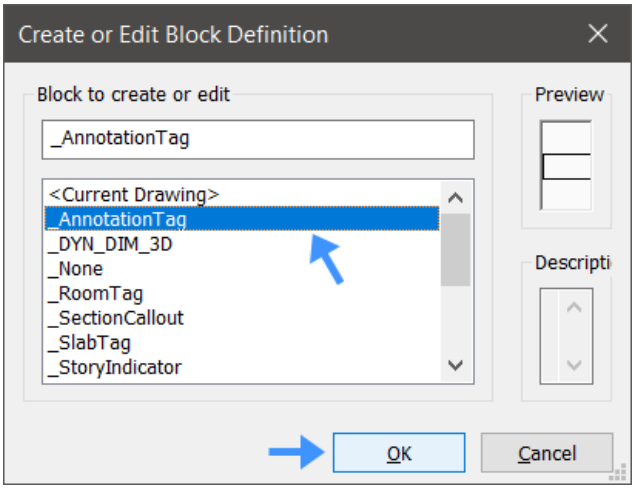
5 Open the generated section drawing.

6 Type BIMTAG in the Command line OR select **Tag Automatically** in the **Annotate** tab of the Ribbon.

Nota: The tag appears empty because it refers to the **Description** property by default.

7 Launch the BEDIT command.

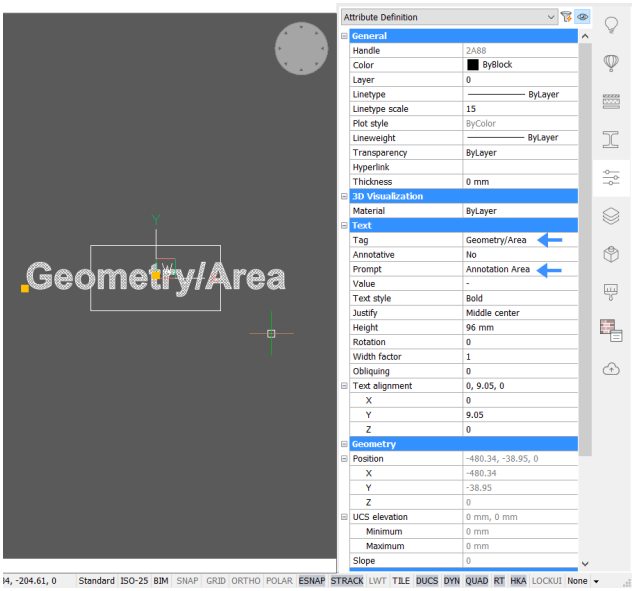
8 Select **_AnnotationTag** and click **OK**.



9 Select the attribute definition, in the properties panel.

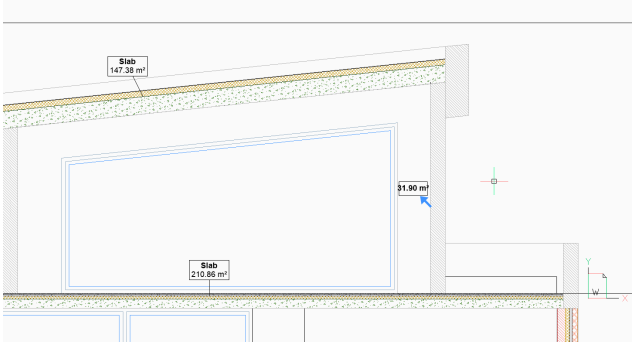
10 Edit the **Tag** property, change it to **Geometry/Area**.

11 **(Optional)** You can also change the **Prompt** property.



12 Enter BCLOSE in the Command line and click **Save**.

13 Select the viewport and pick **Generate Section** ( ) from the Quad. The area of the region is now displayed in the tag.



20.7.2 BIM sections

20.7.2.1 Commands

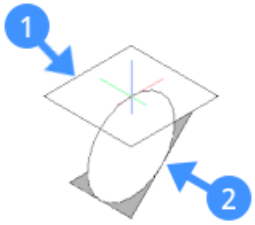
BIMSECTION

20.7.2.2 Defining a section

In BricsCAD BIM, the **Define Section** tool allows you to create BIM section entities. The BIM section allows you to section a BIM model by a plane and thus view the interior details.

For more information about this command, visit the Command Reference article BIMSECTION.

When defining a BIM section, the section tracker is attached to the cursor.



The section plane (1) lies in the XY-plane of the current coordinate system (WCS or UCS). The view direction (2) is in the negative direction of the current coordinate system.

Nota:

- If Dynamic UCS (DUCS) is ON, the section plane aligns to the face of the 3D solid under the cursor. Hit the TAB key to select an obscured face.
- You can press the Shift key to lock a highlighted plane, allowing you to specify the definition point for the BIM section entity outside the face of the solid.
- The section line is parallel to the X-axis of the coordinate system (WCS, UCS or DUCS).

General procedure to create a BIM section

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Do one of the following:
   - To create a plan section, click a point anywhere outside the model.
   - Hover the cursor over the face of a 3D solid, parallel to the section plane you want to create.
   - The section tracker aligns to the face under the cursor, on the condition that DUCS is active. Left-click.
   - **(Optional)** Hit the Shift key to lock the highlighted plane. This allows you to start from a point outside the selected 3D solid face. Next, left-click.

3 The initial section plane displays dynamically and the 3D model is clipped accordingly.

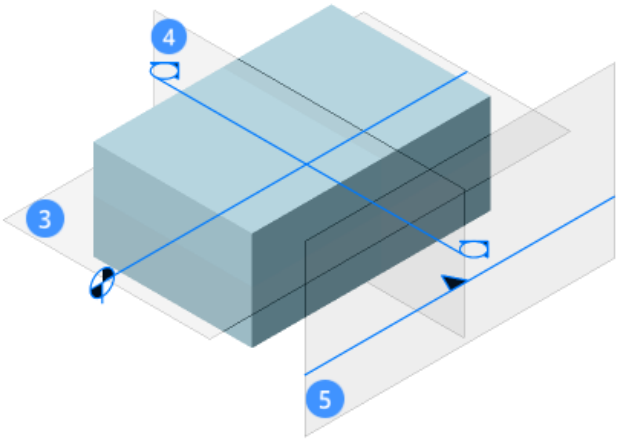
4 You are prompted: Specify distance:

   Do one of the following:

   - Type a value in the dynamic dimension field to offset the section from the initial position.
   - Left-click when the section plane is at the location you want it to be.

   The BIM Section entity is defined.

   The Section Type depends on the direction of the section plane (horizontal or vertical) and whether it intersects with the model.



Plan (3)	Displays horizontal section plane.
Section (4)	Displays a vertical section that cuts through the model.
Elevation (5)	Displays the exterior elevation view.

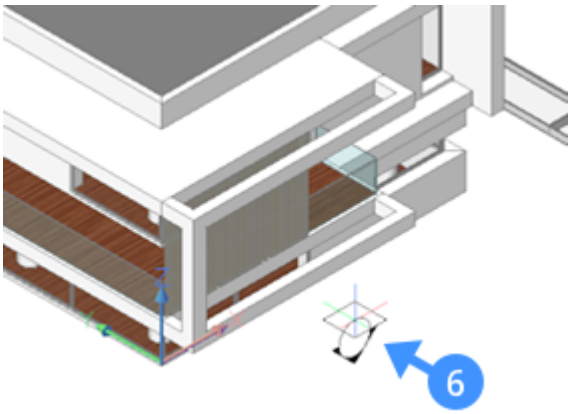
Creating a plan section

Nota: A plan section is best defined in the WCS (World Coordinate System).

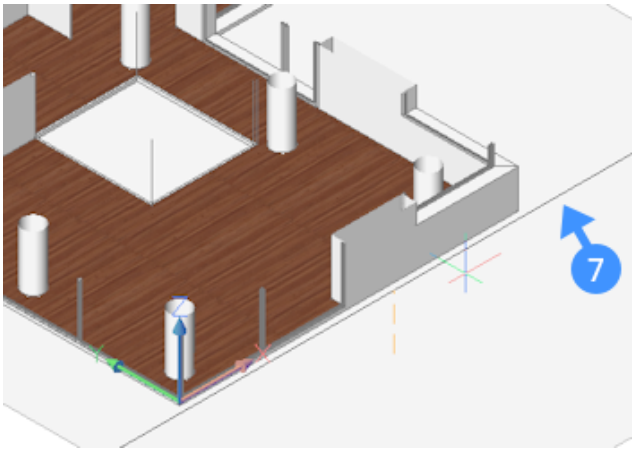
1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

   You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker near the 3D model and click (6).



The initial section plane lies in the XY-plane (Z = 0) of the current coordinate system (e.g. the WCS). The section plane displays dynamically and the 3D model is clipped accordingly (7).



You are prompted: Specify distance:

3 If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

4 Do one of the following:
   - Move the cursor and click when the section plane is at the desired height.
   - Use an Entity Snap to specify a point.
   - Type a value to define the offset from the initial position of the section plane.

The plan section is defined.

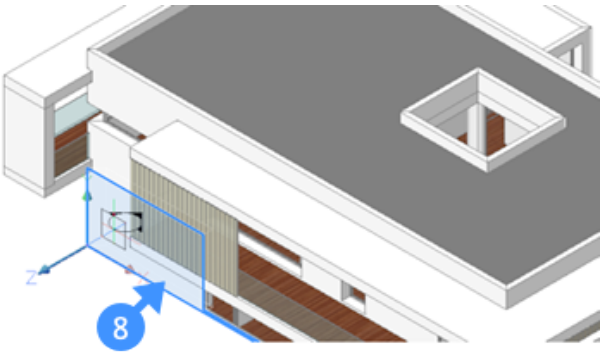
Creating a cross section

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

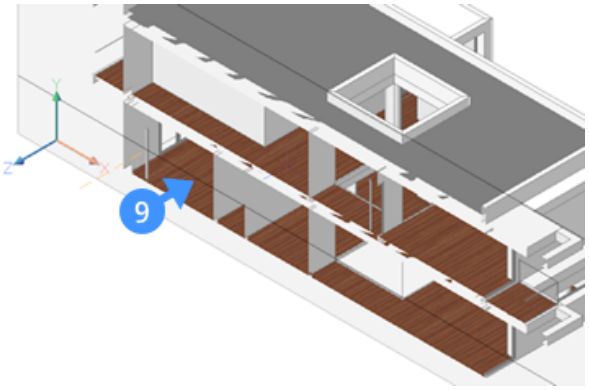
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker over a 3D solid face (8) parallel to the cross-section to be defined.



3 Click when the face highlights and move the section tracker inside the 3D model.

4 The section plane displays dynamically and the 3D model is clipped accordingly (9).



You are prompted: Specify distance:

If Dynamic Dimension is ON, the current distance displays in a dynamic dimension field.

5 Click when the section plane is at the desired location or specify a distance.

6 The cross-section is defined.

7 **(Optional)** Click the section entity and hold down the left mouse key a little longer to display the **Manipulator**. Drag an axis of the manipulator to move the section.

8 **(Optional)** Click the Arrow grip to flip the view direction.

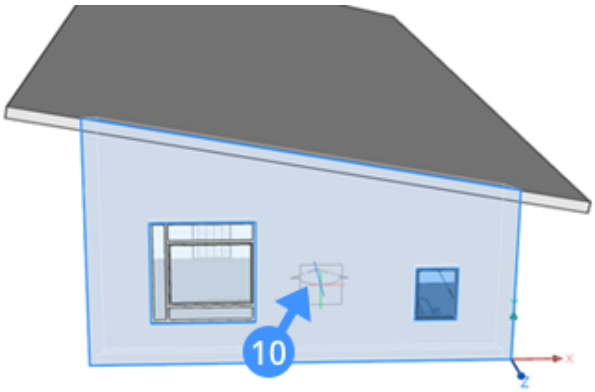
Creating an elevation

Make sure Dynamic UCS (DUCS) is enabled.

1 Choose **Define Section** in the **Model** tab of the **No Selection Quad**.

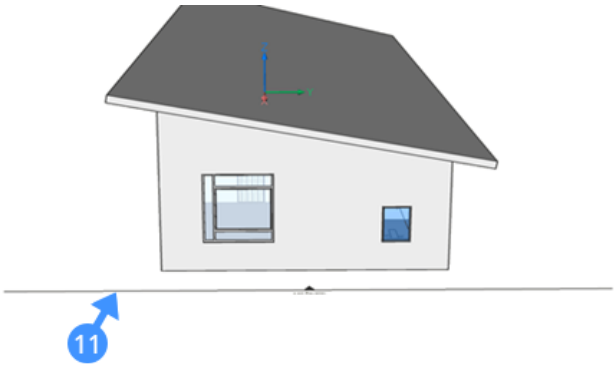
You are prompted: Select a point to place section or [Detail/Interior/Scale/Reflected ceiling]:

2 Place the section tracker over a 3D solid face (10) which is parallel to the elevation to be defined and click when face highlights.



3 You are prompted: Specify distance:

4 Move the section tracker outside the 3D model and click to define the elevation (11).



Vertical sections outside the model (no geometry is sectioned) get the Elevation section type automatically.

20.7.2.3 Editing the default properties of a BIM section

The default properties of a BIM section entity are copied from the section entities in _SectionSettings.dwg in the **BIM/Sections** subfolder of the **Support** folder.

1 Select the BIM section in the drawing or the **Structure panel**.

The section properties display in the Properties panel.

2 Define the properties of the BIM section entity.

General	
Handle	14F2
Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
History	Current
3D Visualization	
Material	ByLayer
Section Entity	
Name	Floor 0
State	Plane
Live section	No
Clip display	No
Use lids	Yes
Plane transparency	70
Plane color	9
Destination file	Plans
Geometry	
Elevation	120 cm
Top plane	15.86 m
Bottom plane	15.86 m
Vertex	1
Position	-4122.5, 7000, 1200
X	-4122.5
Y	7000
Z	1200
UCS Elevation	899.57 mm, 139.957 cm
Minimum	899.57 mm
Maximum	139.957 cm
BIM	
Name	
Description	
Section type	Plan
Callout position	Start
Initial scale	1:50
Layer state	
Apply layer state	Off
Project section	On
Generate tags	On
Entity property sets	

Section Entity:

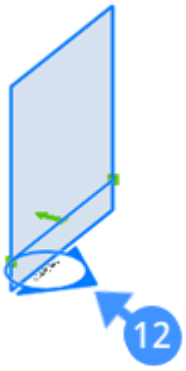
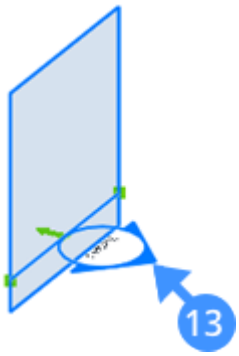
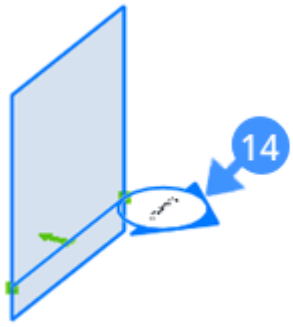
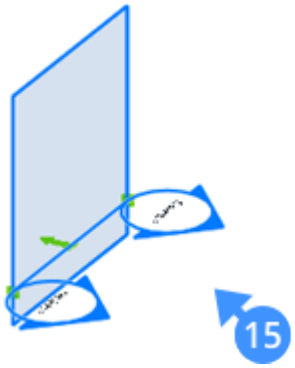
Name	Type a name.
State	Click the field and select a state. The selected state determines how the section result will be clipped. The options are: **Plane** (default), **Boundary** and **Volume**.
Live section	Sets the live section property of the section plane. Only one section entity can have the **Live section** property **ON**. It is recommended to use the **Clip display** property instead.

Geometry:

Elevation	Defines the Z-value of the start point of the section plane.
Top plane	Defines the top plane of the section volume.
Bottom plane	Defines the bottom plane of the section volume.
Vertex	Defines to coordinates of the control points of the section plane. The number of vertices depends on the current section plane state. Plane: 2 vertices Boundary and Volume: 4 vertices. Select the field, then click the arrow buttons to select a vertex. An X indicates the selected vertex in the model.

BIM:

Name	Type a BIM name.
Description	Type a description.
Section type	Sets the BIM section type.

Callout position	Defines the position of the callout(s). The callout block and default position depend on the section type. The options are: **Start** (12), **Mid** (13), **End** (14) and **StartEnd** (15).     Click the green arrow ( ) to flip the view direction of the section.

Initial scale	Sets the initial scale of the papers pace viewport in the generated drawing.
Layer state	Sets the layer state to be used to calculate the section.
Apply layer state	Determines whether the layer state will be used to calculate the section or not. Applying a layer state allows to control which layers are ON/OFF when calculating the section result.
Project section	Controls the display of the section callouts and section line in section drawings which are perpendicular to this section.
Generate tags	Controls whether BIM tags are generated automatically.
Entity property sets	Controls whether 'per instance' entity properties can be used to generate tags. Click the **Browse** (...) button, then select the property sets in the **Enable per-instance property set** dialog box.

20.7.2.4 How to toggle the clipped display property of a BIM section entity

The **Clip display** property of the **BIM** section temporarily clips 3D models. The **Clip display** can be used in all modeling and editing operations on all drawing content.

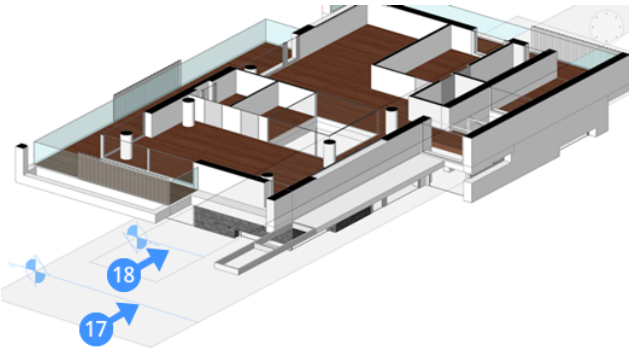
The following illustration shows the 3D model of a villa with a plan section (16) for the ground floor plan.

To clip the 3D model, do one of the following:

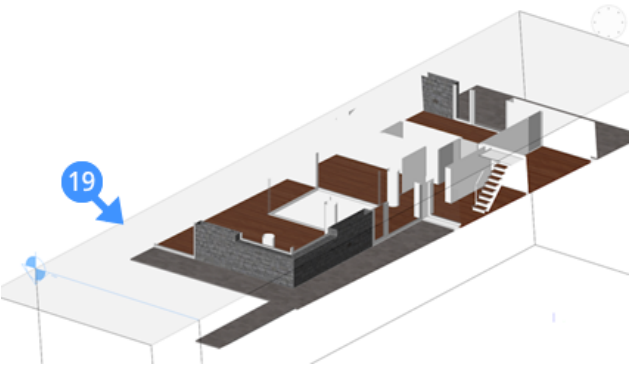
- Double click the section entity.
- Select **Clip display** from the Quad.
- Select the section plane and set the **Clip display** property to **Yes** in the properties panel.
- When **RT** (Rollover Tips) is selected in the **Status** Bar, place the cursor over a section entity to set the **Clip display** property.

Nota: **Clip display** can be on for multiple section entities simultaneously.

In the following illustration, **Clip display** is turned ON for two BIM section entities (17,18). These section entities are in opposite view directions: section plane 18 is a plan section, section plane 17 is a reflected ceiling section.



Option: Set the **State** property for the current section to **Volume** in the properties panel.



A clipped section shown in the **Volume** state (19).

20.7.2.5 Converting a section entity to a BIM section

The BIMIFY command converts a section entity to a BIM section.

1 Launch the BIMIFY command.

1 You are prompted: Run Bimify on entire model or [Advanced] <Model>:

2 Choose the **Advanced** option.

The **Bimify** dialog box appears.

3 Choose **Select entities**, then click the **Select** button.

The **Bimify** dialog box closes.

4 Select the section entity, then right click.

5 The **Bimify** dialog box reopens.

6 Click the **OK** button.

20.7.3 BIM tags

20.7.3.1 Commands

BIMTAG

20.7.3.2 Source files

The following source files are used to create BimTags in a generated BIM section drawing:

- **_SectionTag.dwg**: contains the tag blocks and multileader styles.
- **_TagTypeToStyle.xml**: links a BIM entity type to a multileader style.

Both files sit in the folder:

C:\Users\<UserName>\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\Support\Bim\Sections

Nota: when assigning tags, BricsCAD needs to access the 3D model. For this reason it may take several seconds to place the first tag, if the 3D model is not currently opened, especially in large models. If you want to avoid this waiting time, consider having the 3D model open while placing tags.

20.7.3.3 Assign BIM tags manually

The BIMTAG Manual command allows you to select the BIM element that you wish to create a BIM tag for.

For more information about this command, visit the Command Reference article BIMTAG.

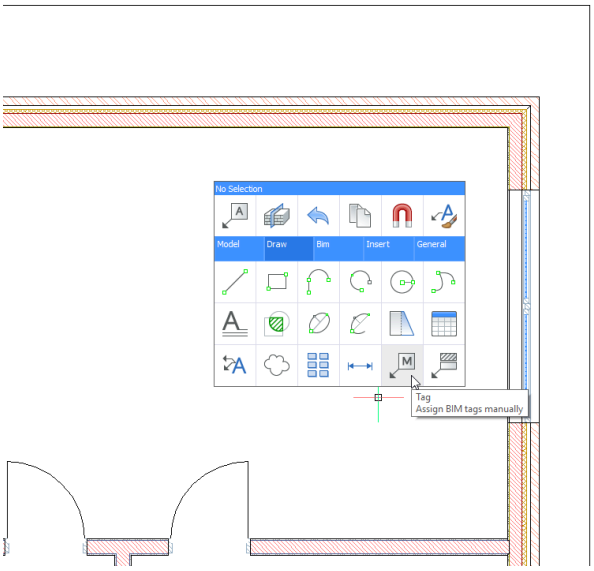
Manually create a BIM tag

1 Open the section drawing for which you wish to create BIM tags.

Nota: For more information about generating section drawings, please read the BIM article **Generating Drawings** first.

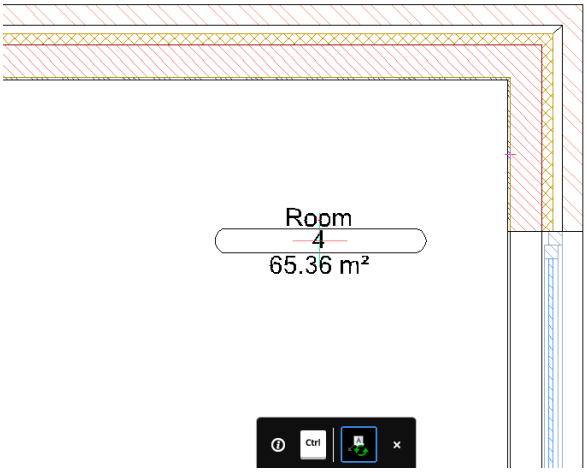
2 With no selection, select **Tag Assign BIM tags** manually from the Quad or enter BIMTAG followed by M

into the Command line to choose the manual option.



You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

3 Select a point on a line of a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible tags.



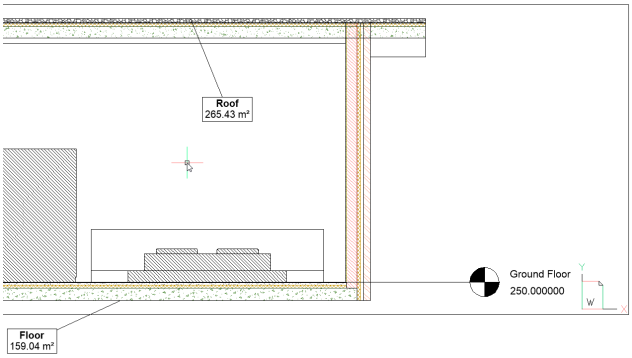
You are prompted: Place the tag [Next element]:

4 Move your cursor in order to adjust the position of the tag and left-click to place it.

5 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

6 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require more tags, press the ESC or ENTER key to complete the command. If you want all elements to be tagged, you can use the BimTag Select all entities command which we will explore next.

7 The same steps above can be used to tag a floor slab or a roof.



20.7.3.4 Assign BIM tags automatically

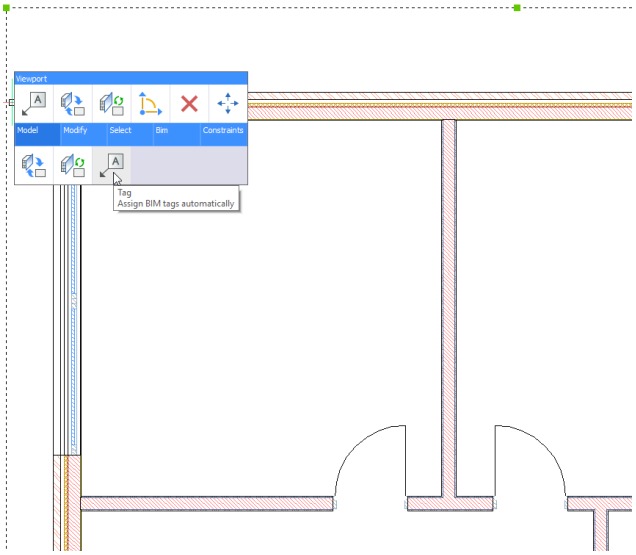
If you want all BIM elements in a section drawing to be tagged it is easiest to use the BIMTAG Select all entities command. This command automatically assigns BIM tags to all BIM entities inside a viewport. If the model is altered or new elements are added, the BIM tags can be updated and new tags added.

For more information about this command, visit the Command Reference article BIMTAG.

Automatically create BIM tags

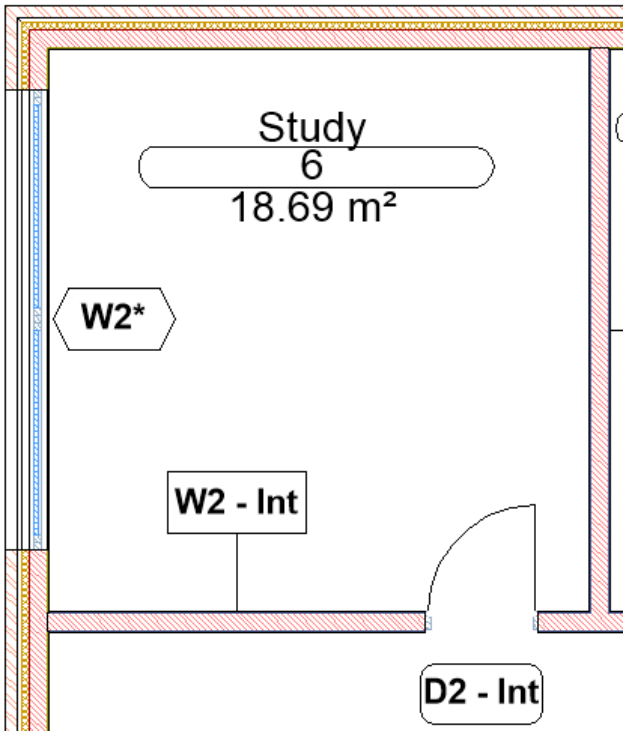
1 Open the section drawing on which you wish to create BIM tags.

2 Select a viewport containing the section which you will add BIM tags to.



3 Select **Tag Assign BIM tags automatically** from the Quad or enter BIMTAG into the Command line.

The BIM tags have now automatically been generated for each of the BIM entities in the viewport.



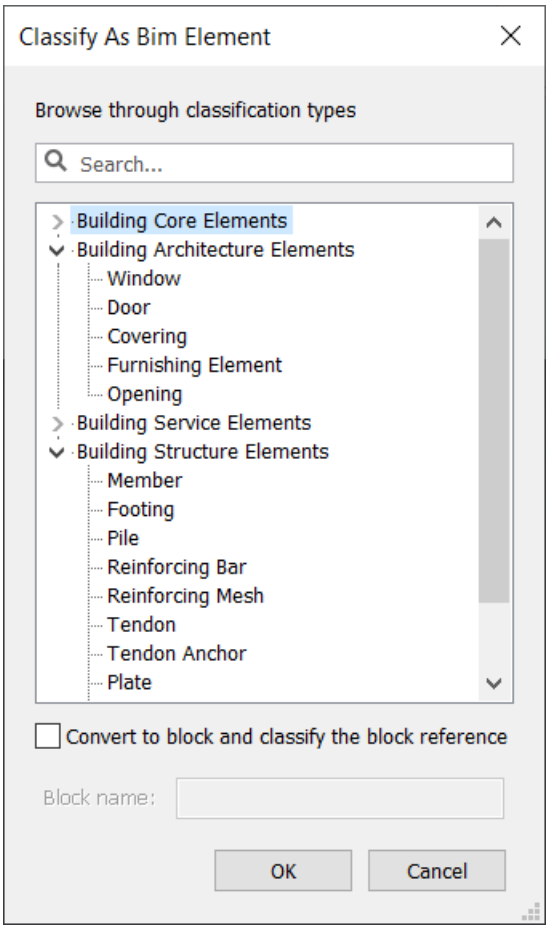
You are prompted: Select BIM section viewport [Manual]:

4 If you wish to automatically create BIM tags in another viewport on the sheet, select the viewport and these will be generated. If you do not require any further BIM tags, press Esc to complete the command.

20.7.3.5 Create the source data for a BIM entity type

By default the source data for the following BIM entity types already exist; **Annotation**, **Beam**, **Column**, **Curtain Wall**, **Door**, **Plies**, **Railing**, **Ramp**, **Roof**, **Room**, **Slab**, **Stair**, **Wall**, and **Window**.

Available BIM entity types are listed in the **Classify As Bim Element** dialog box. To open this dialog box, choose the **Other command** option of the BIMCLASSIFY command.



Create the tag block and multileader style

To create the source data for a Bim Member entity type:

1 Type SUPPORTFOLDER in the Command line and press Enter.

2 Once the folder has opened, navigate to the **BIM** > **Sections** folder and open the _SectionTag.dwg file.

3 Launch the EXPBLOCKS command.

4 The **Drawing Explorer / Blocks** dialog box will display.

5 Select the **_AnnotationTag** block (or any other existing block that you want to use as a template).

6 Right-click and choose **Copy** from the context menu.

7 Right-click and choose **Paste** from the context menu.

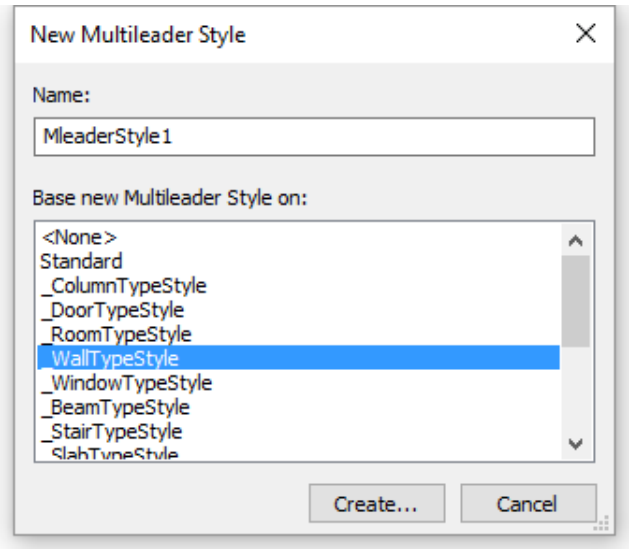
8 Click **Copy**, but keep both on the **Copy/Paste** dialog box.

9 Rename the copied block to **_MemberTag**.

10 Select **Multileader Styles** in the **Drawing Explorer** dialog box.

11 Click the New ( ) tool button.

The **New Multileader Style** dialog box will display:



12 In the Name field type _MemberTypeStyle.

13 Select a source style in the **Base new Multileader Style on** list.

14 Click the **Create...** button.

15 **(Optional)** Edit the **Leader** properties.

16 Click the **Content** tab.

17 Under **Block options**, click the **Source Block list** button and select **_MemberTag** from the list.

18 Close the **Drawing Explorer** dialog box.

19 Save _SectionTag.dwg.

Link the entity type to its multileader style

1 Type SUPPORTFOLDER in the Command line, then go to the **BIM** > **Section** folder and open _TagTypeToStyle.xml.

Use an ASCII text editor, e.g. Notepad.

2 Add the following right before </TagTypeToStyle>:

```
<tag type="BIM_MEMBER">
<style
name="_MemberTypeStyle"
autoPlacement="true"
autoRotation="true"
offset="1"
/>
</tag>
```

3 Save the file.

4 Note that this style has three options:

- **autoPlacement = "true"** will make sure tags are automatically generated when using the automatic BIMTAG option. If this option is set to false, then this tag type will not be generated automatically (you can still generate these tags manually).
- **autoRotation = "true"** will rotate the tag along the main axis of the tagged entity. If set to false,

the tag will be placed horizontally.

- **offset** will offset the tags perpendicular to the main axis of the tagged entity.

20.7.3.6 Customize tag blocks

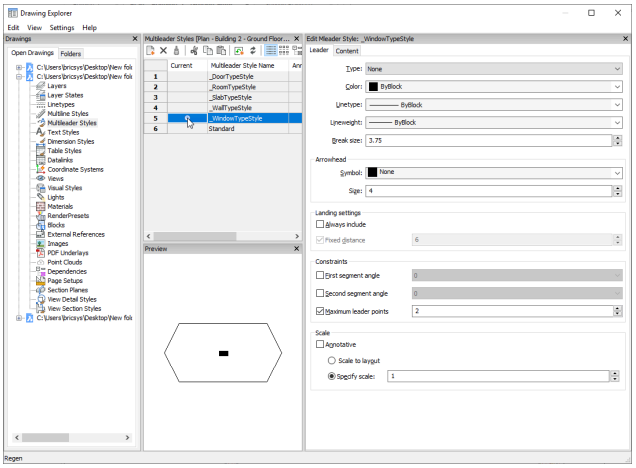
The content of a Bim Tag block consists of:

- Layout entities Linework, fixed texts, etc.
- Attributes Bim entity properties.

Nota: When you execute the BIMTAG command, BricsCAD checks whether the source data (tag blocks and their multileader styles) already exist in the Bim section drawing. If not, the source data is copied from the _SectionTag.dwg file. Otherwise, the tag block definitions and multileader styles in the section drawing are used. Therefore, editing tag blocks in the _SectionTag.dwg file only affects the creation of new Bim tags the first time BIMTAG is executed in a section drawing. If you edit tag blocks in a section drawing it affects Bim tags in that drawing only.

20.7.3.7 Customize tag blocks for the current drawing only

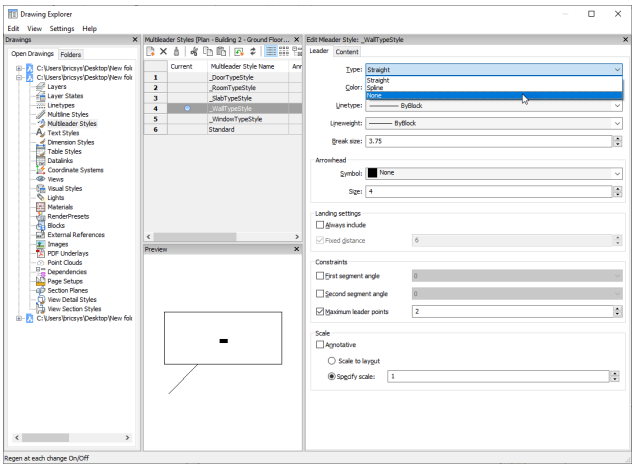
1 In the section drawing for which you wish to customize the BIM Tags, type MLEADERSTYLE in the Command line. The drawing explorer will open in the **Multileader Styles** tab.



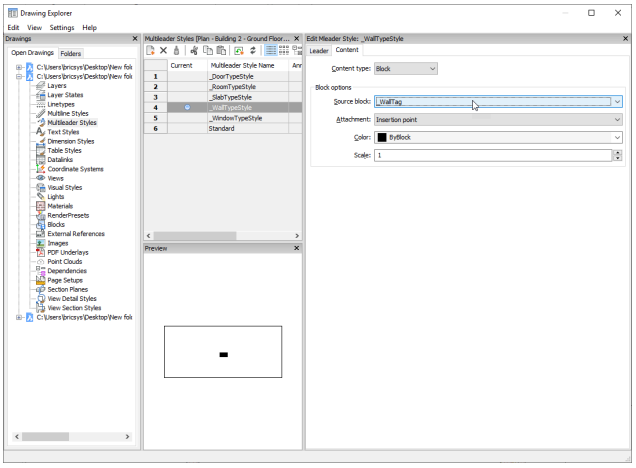
Each BIM tag is a multileader. The multileader style depends on the type of BIM element being tagged. All the multileader styles in the drawing are listed in the drawing explorer.

2 Select **_WallTypeStyle** from the list to see a preview of what the tag will look like. You can change the tag style from here.

3 Change the **Leader Type** from **Straight** to **None**. Now there will be no line connecting the tag to the wall element.



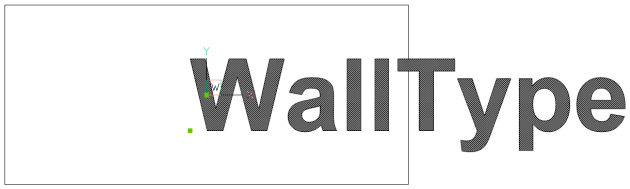
4 In the **Content** tab, the **Source block** is called **_WallTag**. Remember this and now close the drawing explorer.

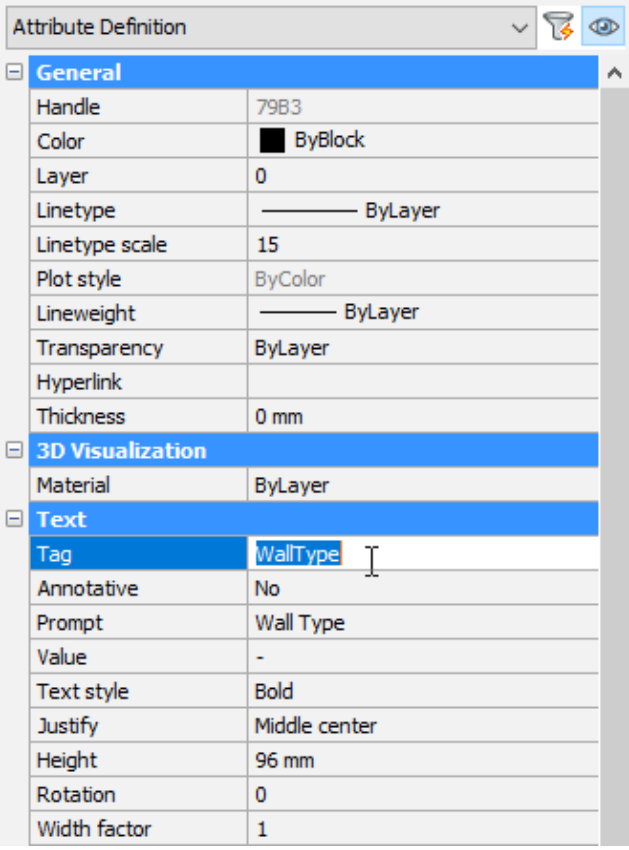


5 In the Command line type BEDIT and select **_WallTag** from the list. Click **OK**.

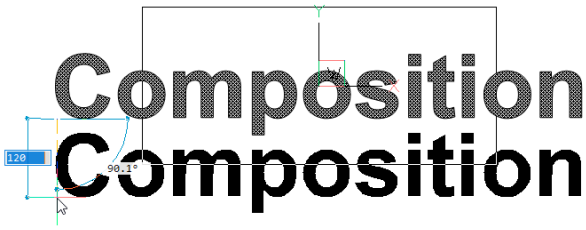
6 Select the **WallType** text and **Tag** value of the properties panel.

7 Replace **WallType** with **Composition** and press ENTER (see the next procedure for the syntax of other property types). Now the attribute text will display **Composition**.





8 Hover over this attribute and select **Copy** from the Quad. Select a base point to paste the copied attribute below the current one.



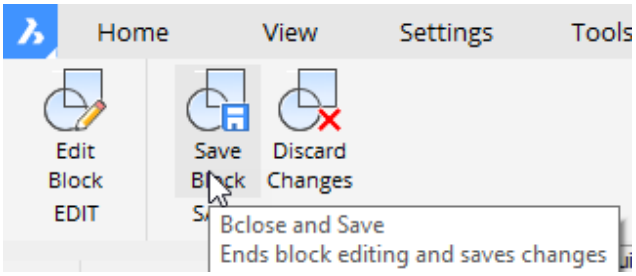
9 Repeat steps 5 - 7 to change the new attribute **Tag** value (**Quantity/Thickness**).

10 Delete the rectangle.

Composition Quantity/Thickness

11 Select the **Save Block** icon from the **Block Editor** tab of the ribbon.

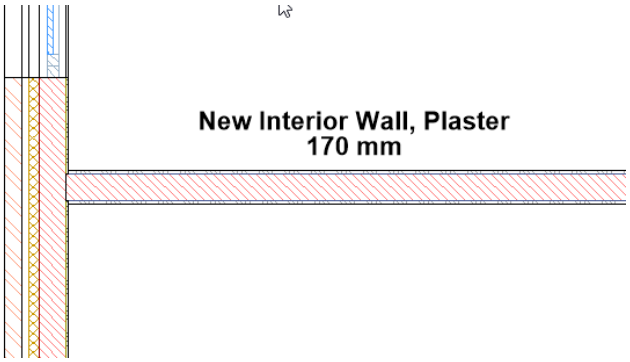
Your section viewport will now display.



12 Highlight the viewport and select **Tag Assign BIM tags automatically** from the Quad.

You are prompted: Tags already have been assigned to the section. Choose an update option [Update existing/update existing and add New/Regenerate all] <Regenerate all>:

13 Click hit the ENTER key to accept the default option of **Regenerate all**. The wall tags will be regenerated with the new style.



20.7.3.8 Edit the source data to customize BIM tags for all section drawings

1 With any drawing open, type SUPPORTFOLDER in the Command line followed. Hit Enter.

2 Once open navigate to the **BIM** > **Sections** and open the _SectionTag.dwg file.

3 Launch the BEDIT command.

4 Select the tag block you want to edit, then click the **OK** button.

5 Add attributes using the following syntax, in the **Tag** field, in the properties panel:

Property type	Syntax	Examples
BIM	<property name>	name room bounding
Parameters	parameters/ <parametername>	parameters/w parameters/framethickness
General	general/<property name>	general/layer general/linetype scale
Mass	mass/<property name> mass/<property name>:<child>	mass/surface area mass/ucs elevation:minimum
Geometry	geometry/<property name> geometry/<property name>:<child>	geometry/position geometry/ucs elevation:maximum
Quantity	quantity/<property name>	quantity/length quantity/net footprint area
IFC	<ifc property set>/<ifc property>	wall common/load bearing door window glazing type/ glass layers
User Defined	<property set label>/<property label>	exterior walls/price
Classification system	<classification system label>/ <classification code label>	omniclass/omniclass classification code

Nota: To know which properties are available for a specific BIM entity type, select such entity in the model, then open the Properties panel.

In the image below the properties of a BIM **Wall** entity are listed.

Click the + button to expand a property type node.

3D Solid	
General	
3D Visualization	
Material	ByLayer
Mass	
BIM	
Type	Wall
Name	E1
Description	Ext. Wall type 1
Building	Building
Story	Floor 1
Composition	Cavity Wall, Brick
GUID	1MvYF3FSfA5gFoL$dkVfY5
Wall type	Cavity
Room bounding	On
Centerline	Off
Number	W12
Quantity	
Length	810 cm
Thickness	320 mm
Height	375 cm
Gross volume	9.34 m³
Net volume	9.34 m³
Gross side area	29.18 m²
Net side area	29.18 m²
Gross footprint area	2.49 m²
Net footprint area	2.49 m²
Plies	4
Ply	1
Wall Common	
Reference	
Acoustic rating	45dB
Fire rating	2h
Combustible	Off
Surface spread of flame	
Thermal transmittance	0 W/m²·K
Is external	On
Extend to structure	Off
Load bearing	On
Compartmentation	On
OmniClass	
OmniClass classification code	22-04 27 23 Cavity Wall Unit Masonry

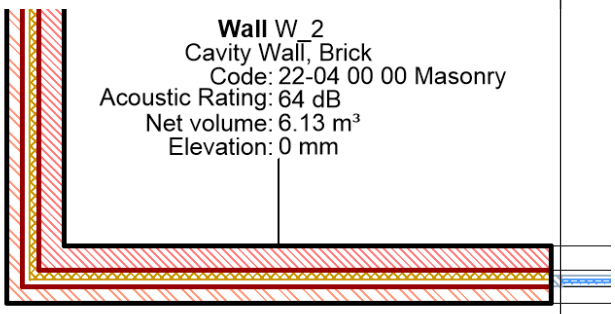
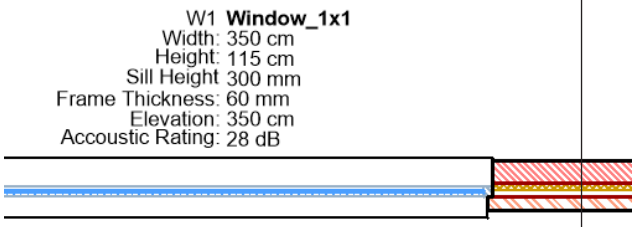
6 For each new attribute do the following:

- Copy an existing attribute or create a new attribute (ATTDEF command).
- Select the new attribute.

  Nota: Do not double click the attribute. The **Edit Attribute Definition** dialog box does not accept spaces in the **Tag** field.
- In the Properties panel, type the property reference in the **Tag** field (see the syntax in the above table).
- **(Optional)** Define the Text properties (**Text Style**, **Justify**, **Height**, ...) in the Properties panel.

- **(Optional)** Add linework and fixed text.

7 On the **Block Edit** toolbar: click the **Save** icon to conclude the BEDIT command.

Definitions in the tag block	Resulting tag
type number composition Code: Omniclass/omniclass classification c Acoustic Rating: wall common/acoustic rating Net volume: Quantity/net volume Elevation: mass/ucs elevation:minimum	
_WallTag block	Wall tag in a plan section
number **name** Width: parameters/W Height: parameters/H Sill Height sillheight Frame Thickness: parameters/framethickness Elevation: geometry/ucs elevation:minim Accoustic Rating: window common/acoustic rati	
WindowTag block	Window tag in a plan section

20.7.3.9 Assign composition tags

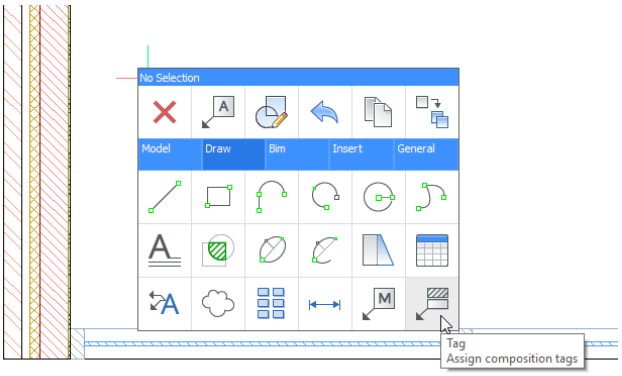
Composition tags allow you to tag a wall or a slab element in order to specify its composition. These BIM tags are a list of the ply materials which make up the element.

For more information about this command, visit the Command Reference article BIMTAG.

Create composition tags

1 Open the section drawing for which you wish to create composition tags.

2 Select **Tag Assign composition tags** from the Quad.

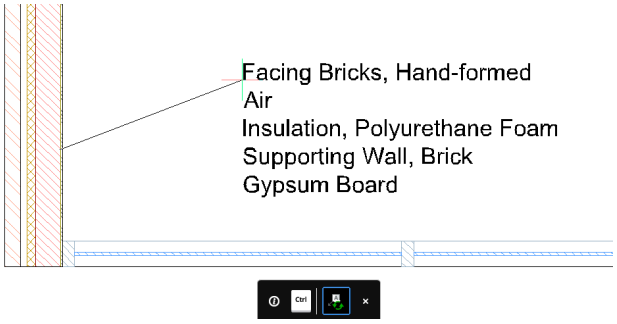


Or type BIMTAG in the Command line, then do one of the following:

- Type M+Enter for the manual option; then T + Enter for **Change Tag** type, followed by O+Enter for the composition option.
- Click the options in the Command line.
- Click the options in the **Prompt** Menu.

3 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

4 Select a point on a BIM element to generate a tag for it. The tag will appear. If the point selected is shared between multiple BIM entities, you can use the CTRL key to cycle through the different possible composition tags.



You are prompted: Place the tag [Next element]:

5 To adjust the position of the tag, move your cursor and left click to place it.

6 You are prompted: Pick point on a sectioned BIM element [Auto/Current mleader style/change Tag type]:

7 Repeat steps 3 and 4 for all of the entities you wish to be tagged. If you do not require any more tags, press the Esc or Enter to complete the command.

20.7.4 Create drawing types with drawing customization

20.7.4.1 About drawing customizing

BricsCAD BIM uses drawing templates to customize your generated drawings. For a general understanding of how these templates are created and work, please visit the Templates for generated drawings article.

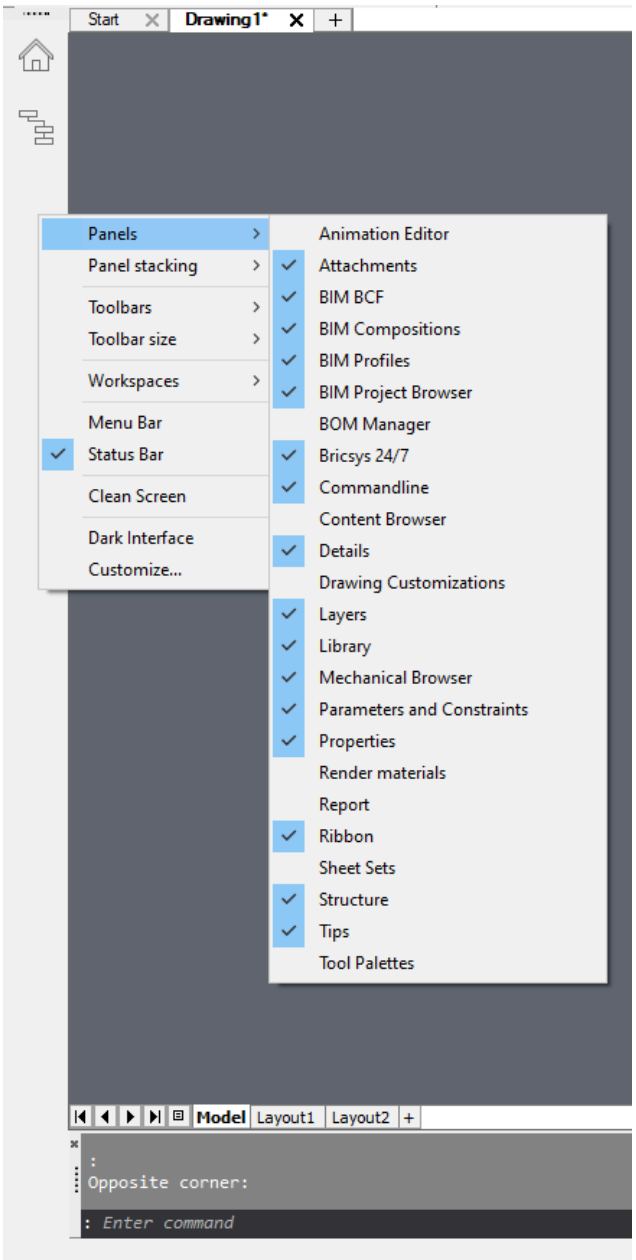
In V21, **Drawing Customization** allows you to create bespoke templates for drawing types based on a set filter parameter rule against an entity type's value or property within the 3D BIM model. This is especially helpful for customizing graphics of model entities independent of their assigned layers or layer states, therefore transferable and usable on other projects with similar entity properties.

The **Drawing Customizations** tool is operated by 3 principles – **Entity Customizations, Filter Rules & Styles** – and is only accessible from the panel.

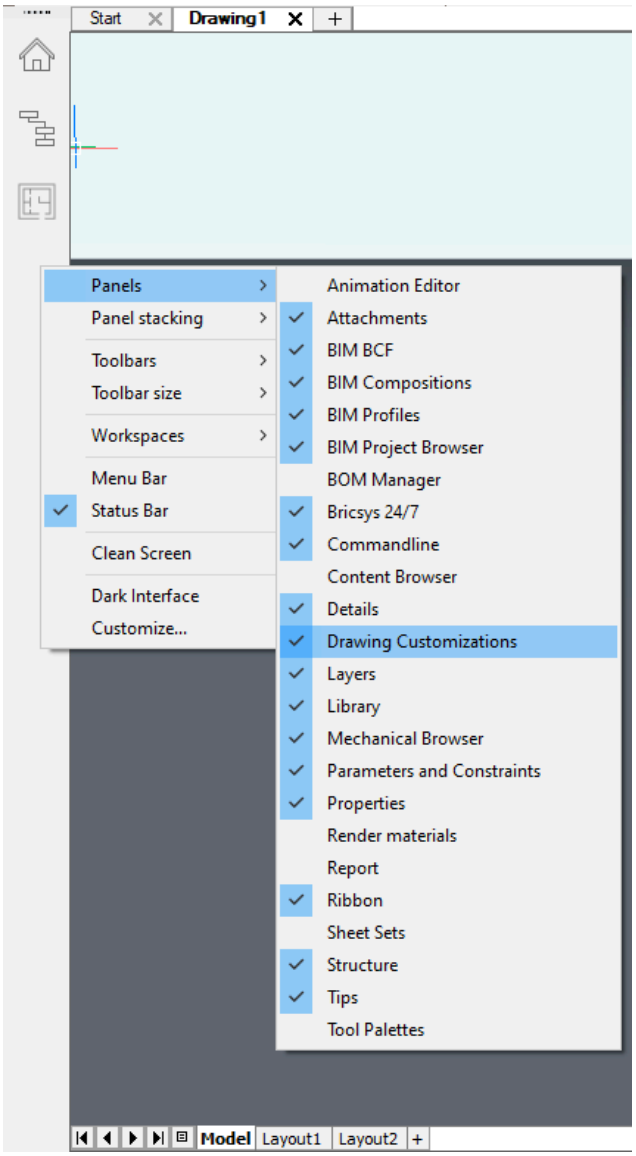
Nota: Version Control for **Drawing Customization** will be implemented from V21.1.06. As features and parameters are constantly improved and added in future V21 updates, using older versions of BricsCAD to open **Drawing Customization** templates may affect their behavior and performance. Therefore, a versioning system would help to prevent customizations from being accidentally opened or modified by BricsCAD versions that are too old to handle them properly.

20.7.4.2 Drawing customizations panel

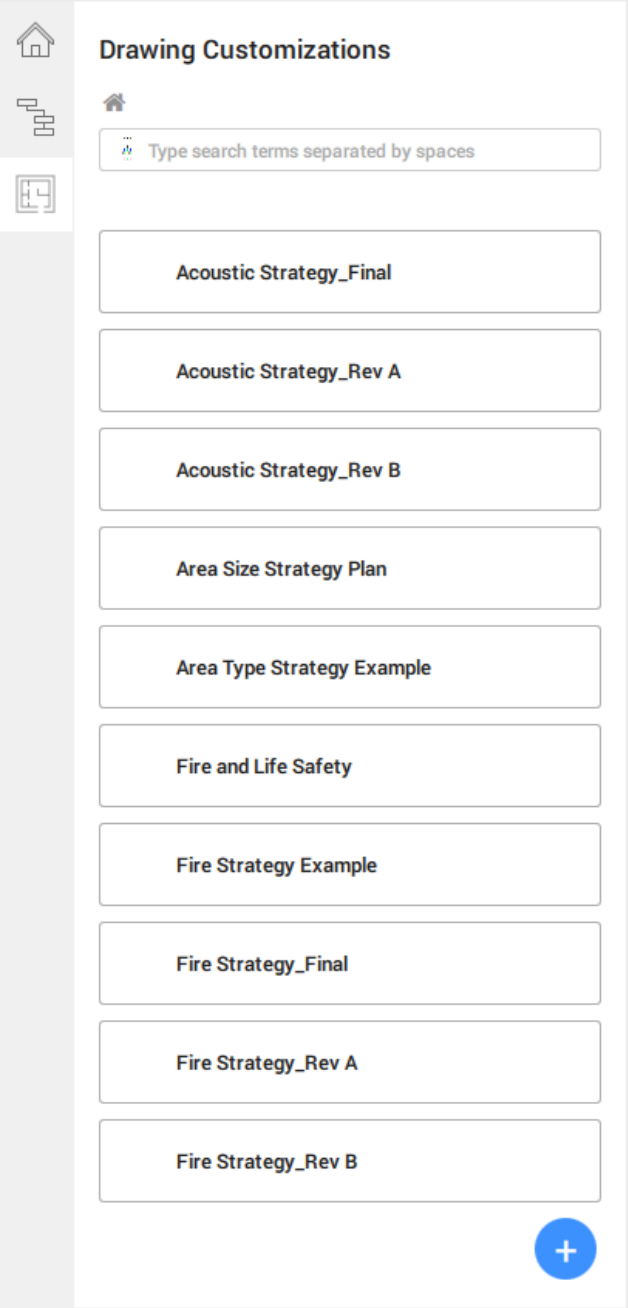
1 Right-click the panel and hover to **Panels**.



2 Select **Drawing Customization** to dock the tool icon in the panel.



3 Open the panel, it allows you to create view templates to control and customize the visibility settings of your drawings on layouts.

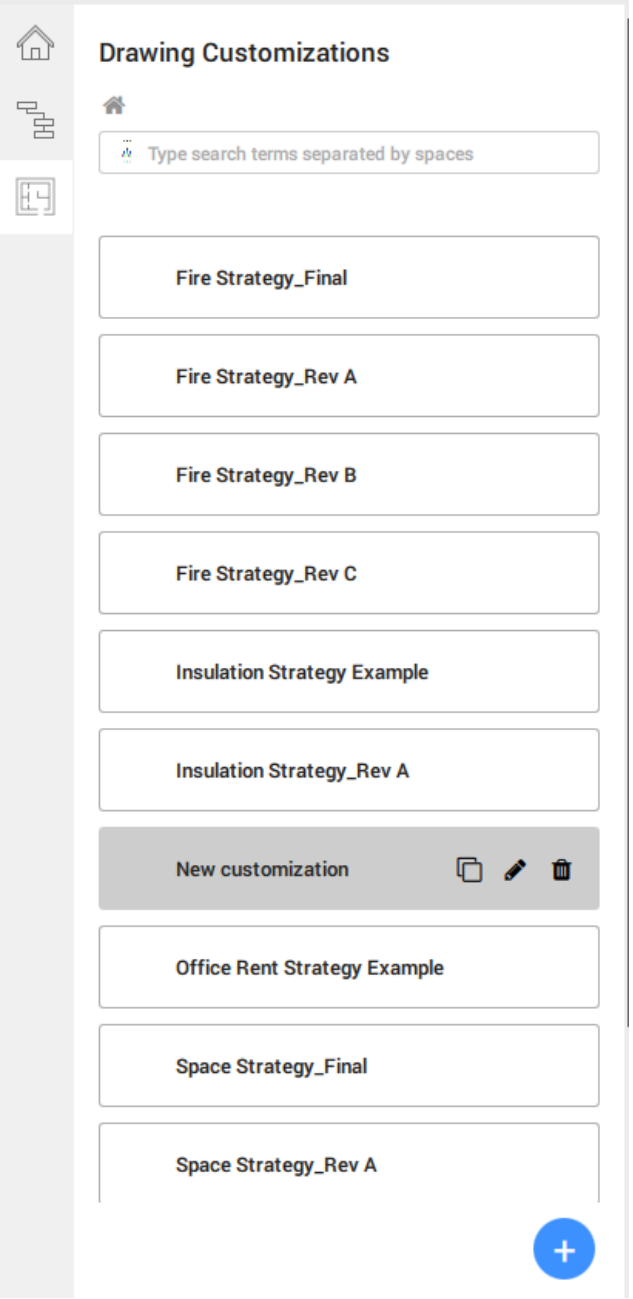


4 Click to create a new template. By default, the template is named **New Customization** with the text highlighted.

Nota: All the controls of **Drawing Customizations** are represented as graphic icons within the **User Interface** panel, such as to create a new control item. They are repeated throughout the tool in

different settings, for example **Entity Customizations**, **Ply Customizations**..., but their functions and purposes remain consistent.

5 Rename the template whilst the text is being highlighted. Once done, press Enter to save and apply the name.

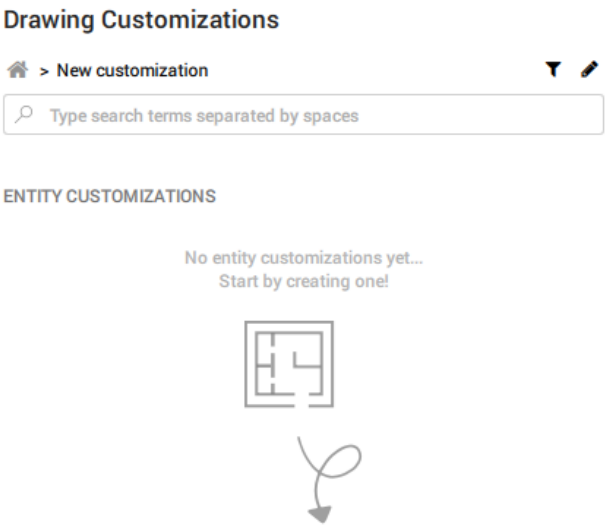


Nota: You can duplicate, rename or delete your customization templates.

6 Click the newly created **Drawing Customization template** tab to begin defining its customization settings.

The first step into the template brings you to **Entity Customizations**, where you will create the

necessary entity customizations template and will be first described later in this article.

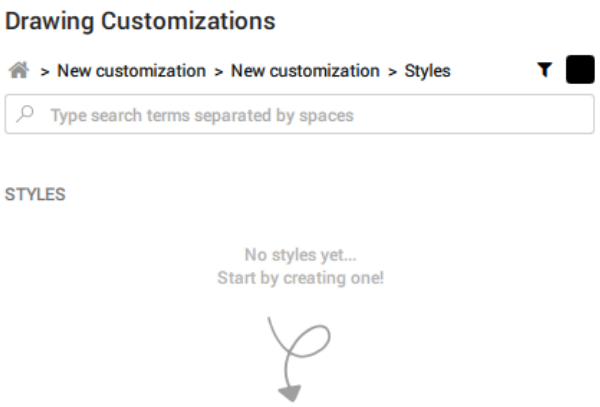


You will also notice a funnel icon and a pencil icon on the top right part of the panel. They are the **Filter Rules** and **Styles** tabs respectively.
The 3 principles – **Entity Customizations, Filter Rules & Styles** – which make up the **Drawing Customization** tool as described earlier in the article, are now accessible on this main page.

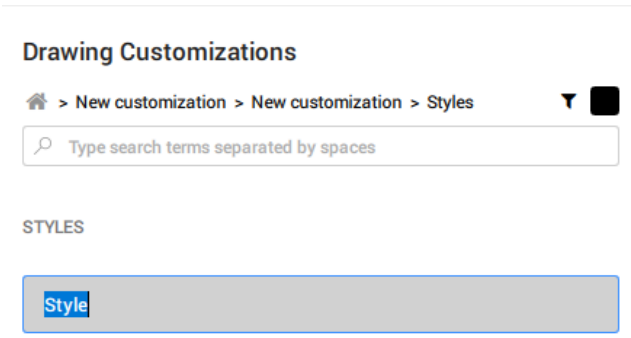
20.7.4.3 Drawing customizations styles

1 Click the pencil icon to access the **Styles** tab.

This is where your desired style options, such as line weight, line colors..., are stored and will appear as drop-down options in the **Entity Customizations** and **Center Customizations** settings as shown in the previous section above.



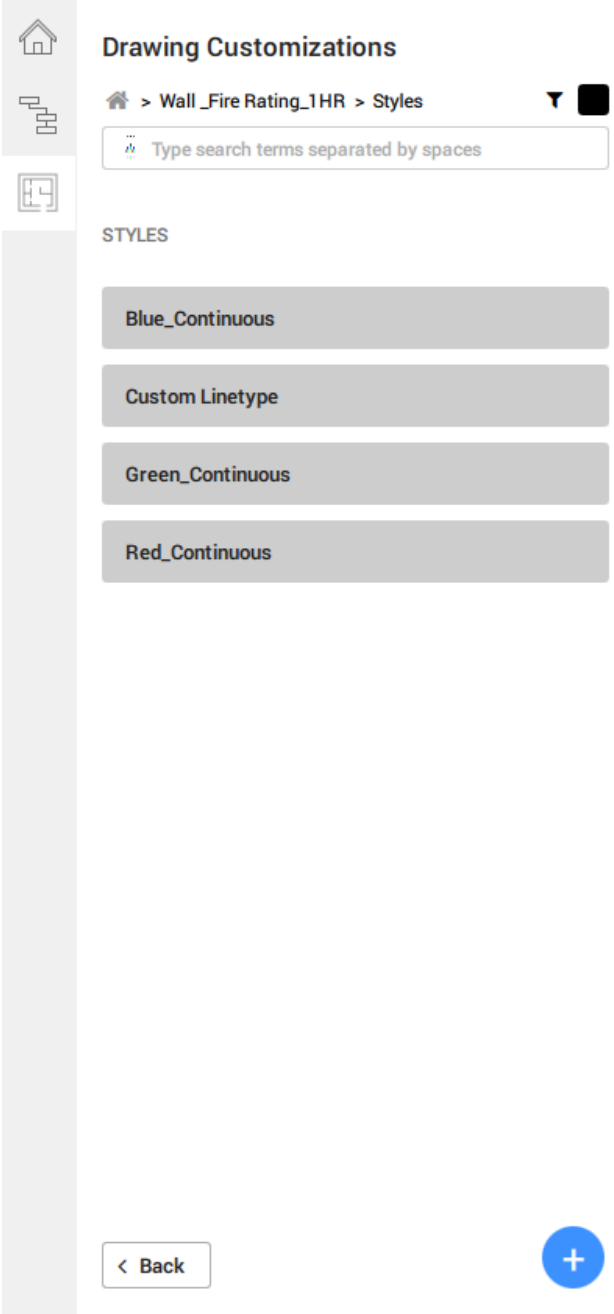
2 To create a new style, click at the bottom right of the panel and a new **Style** template tab will appear.



3 By default, the tab is named **Style**. Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.

4 Create a list of desired styles that you wish to use in this **Drawing Customization** template. Each tab contains its individual styles settings, such as the style's color, line type, line weight and line type scale.

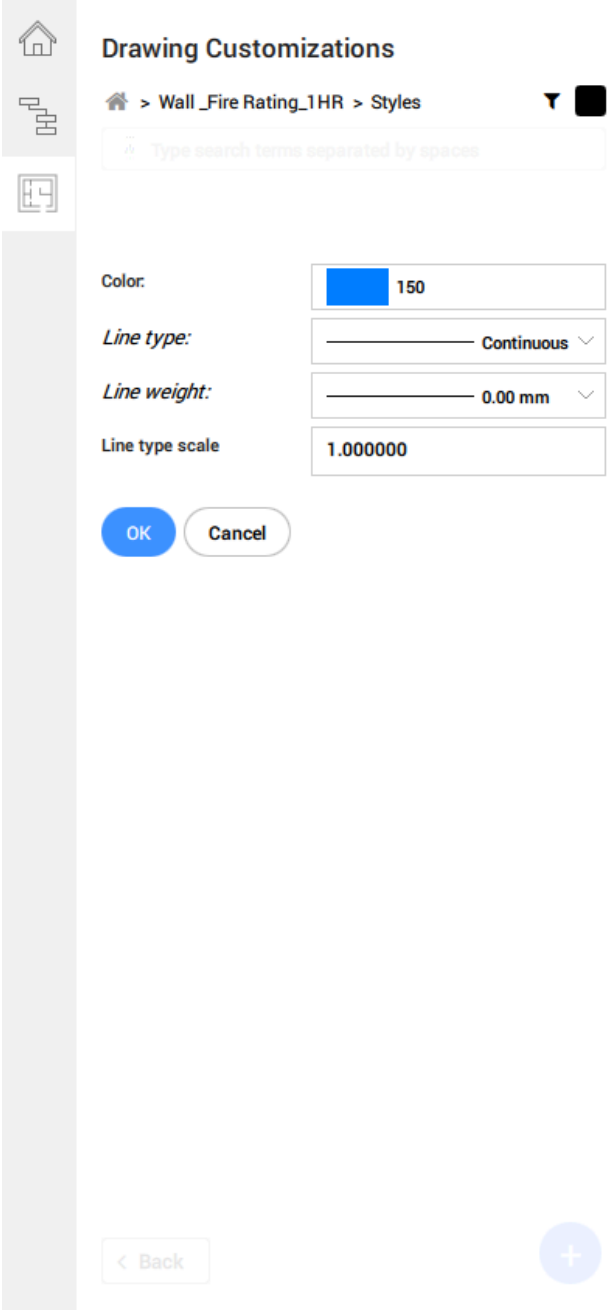
An example of the style pages a list of templates:



5 To edit any of the styles template available, click on its relevant tab to access its settings.

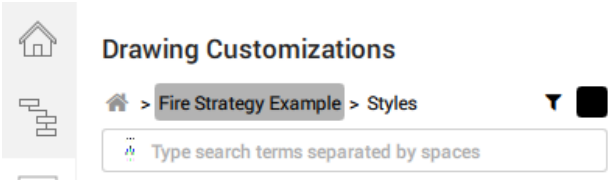
Nota: Similar to the **Drawing Customization template** tabs, hover over the styles template tabs to reveal the function icons to duplicate, rename or delete.

Style settings include the options to define **Color**, **Line Type**, **Line Weight** and **Line type scale** in their respective drop-down dialogs. Once done, click **OK** to save and apply the changes.



6 To leave the **Styles** page, click **Back** at the bottom left side of the panel to return to your previous page. From there, you can navigate to modify the **Entity Customizations** or **Rules** templates.

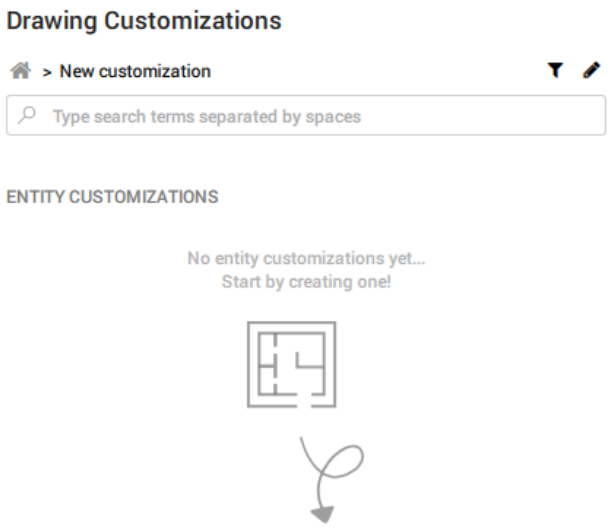
Or click on your desired branch in the tree structure map above the search bar to navigate to your desired page.



Nota: The tree structure map is helpful to keep yourself orientated within the **Drawing Customization** tool. Similarly, the search bar allows you to narrow to a specific template if you have a long list of values to navigate through.

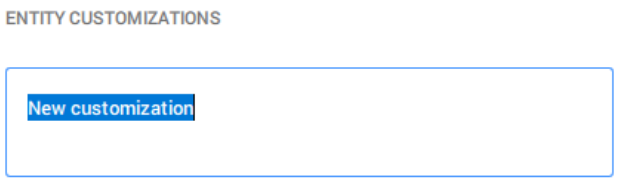
20.7.4.4 Entity customizations

Entity Customizations refer to the visibility settings of how you would like to have your elements in your model drawing to look like.

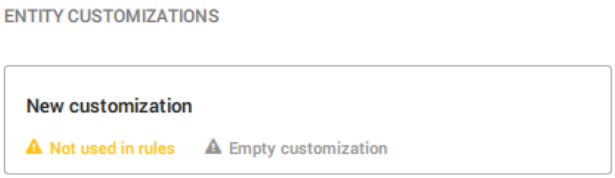


1 Click .

2 To create a new **Entity customization** template. By default, the template is named **New Customization** with the text highlighted.



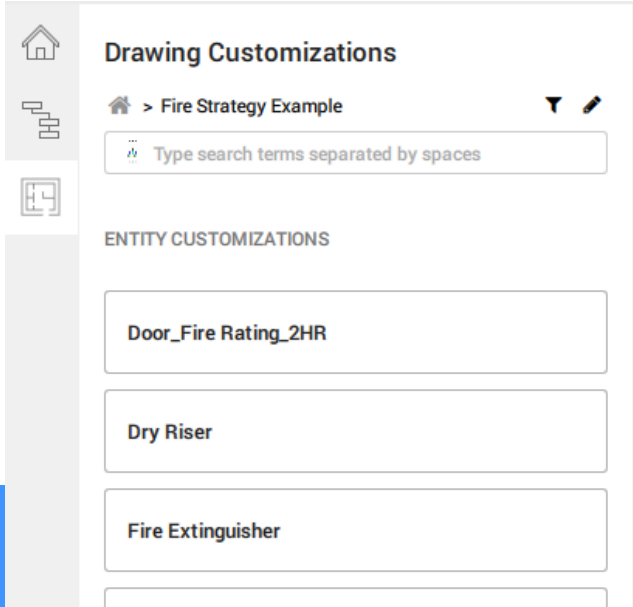
3 Rename it whilst the text is being highlighted as shown. After you are done, hit Enter to save and apply.



As the **Entity customization** was newly created and not fully set up, you will find the relevant notices at the bottom of the **Entity customization** tab. This is a helpful mechanism to remind you if any customization settings have not been defined yet, saving you the need to check the settings manually.

4 Create a list of templates for the entities you wish to customize.

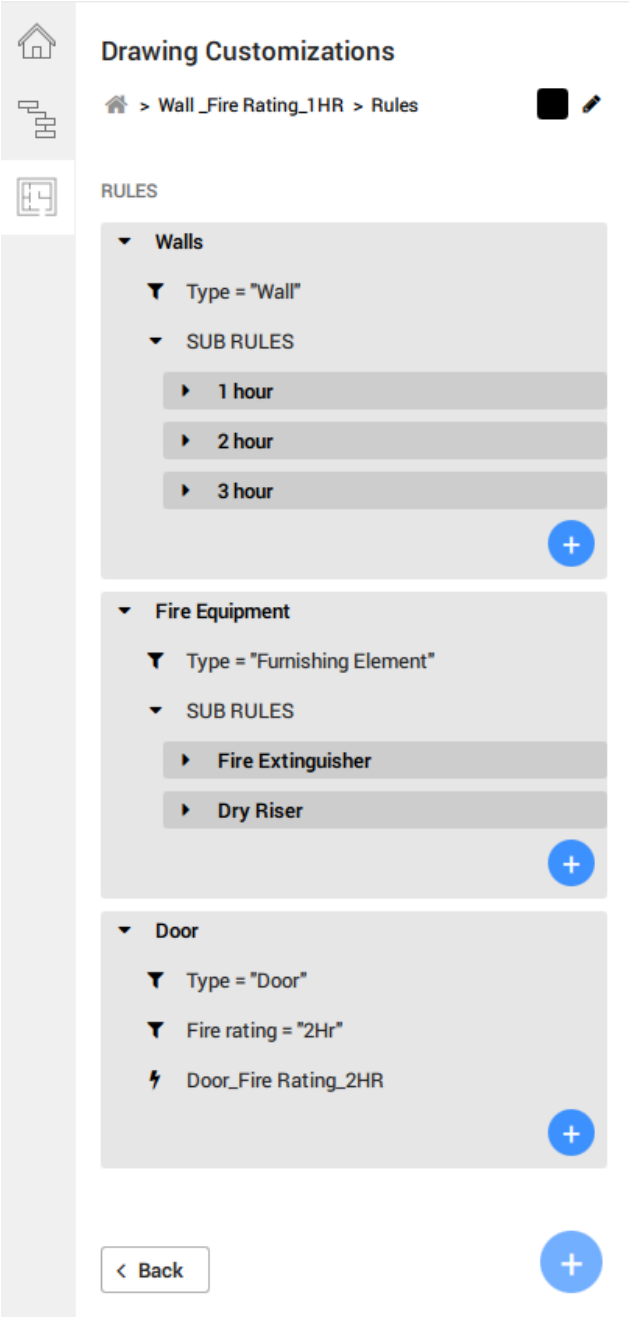
5 A list of **Entity Customizations** templates could look like this:



20.7.4.5 Filter rules

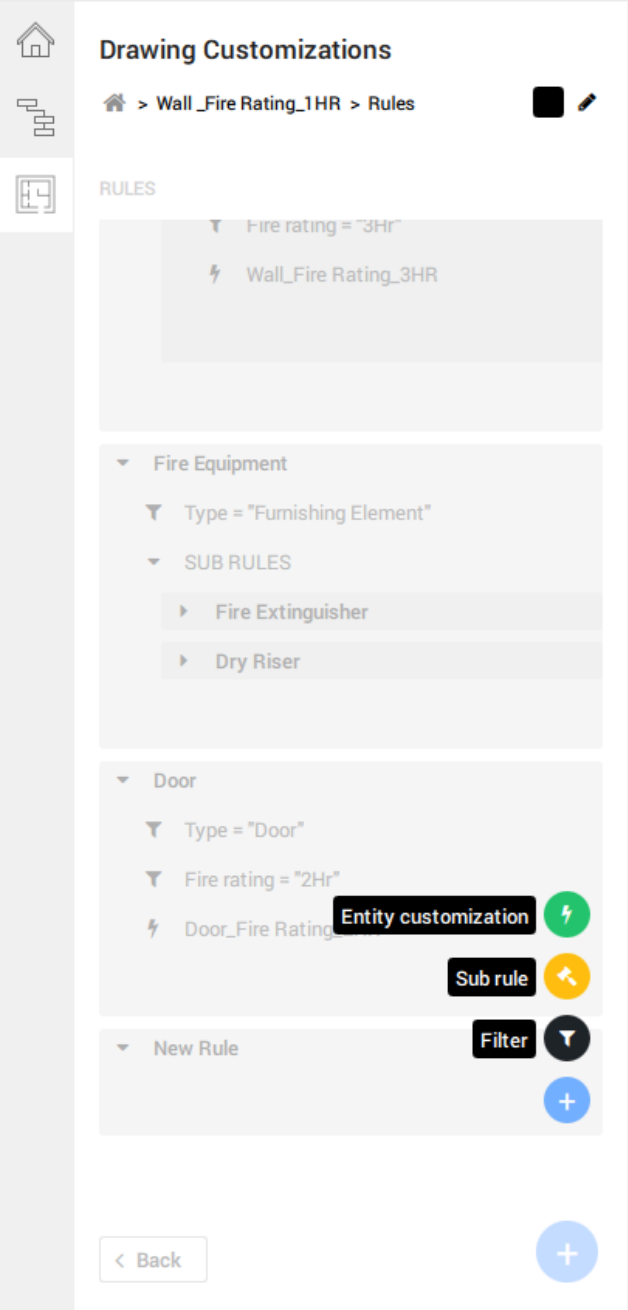
To be able to apply the right customization styles to specific entities, you have to set the appropriate filter rules to target those entities correctly in the **Filter Rules** tab located as a funnel icon on the top right-hand part of the panel next to the **Styles** icon.

Nota: The **Nested Property Search** function has been introduced from V21.1.06 onwards, allowing you to specify how nested objects within an assembly are filtered by and displayed.

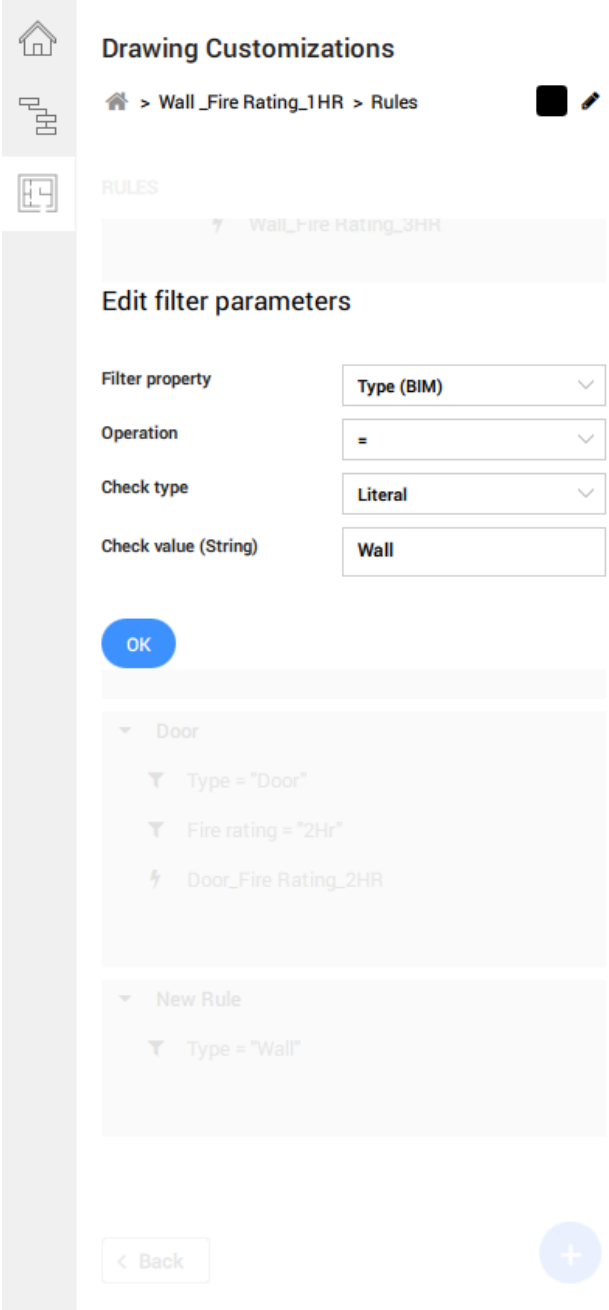


1 **Filter rules** can be further classed into **sub rules**, which falls as a sub-set of the parent rule, e.g. **Fire Equipment** as a **Filter Rule**, whereas **Fire Extinguisher** is classed as a sub rule.

also appears respectively and accordingly at each rule level. At the end of each rule, regardless of it being a subset or otherwise, it has to end with an **Entity Customization**.

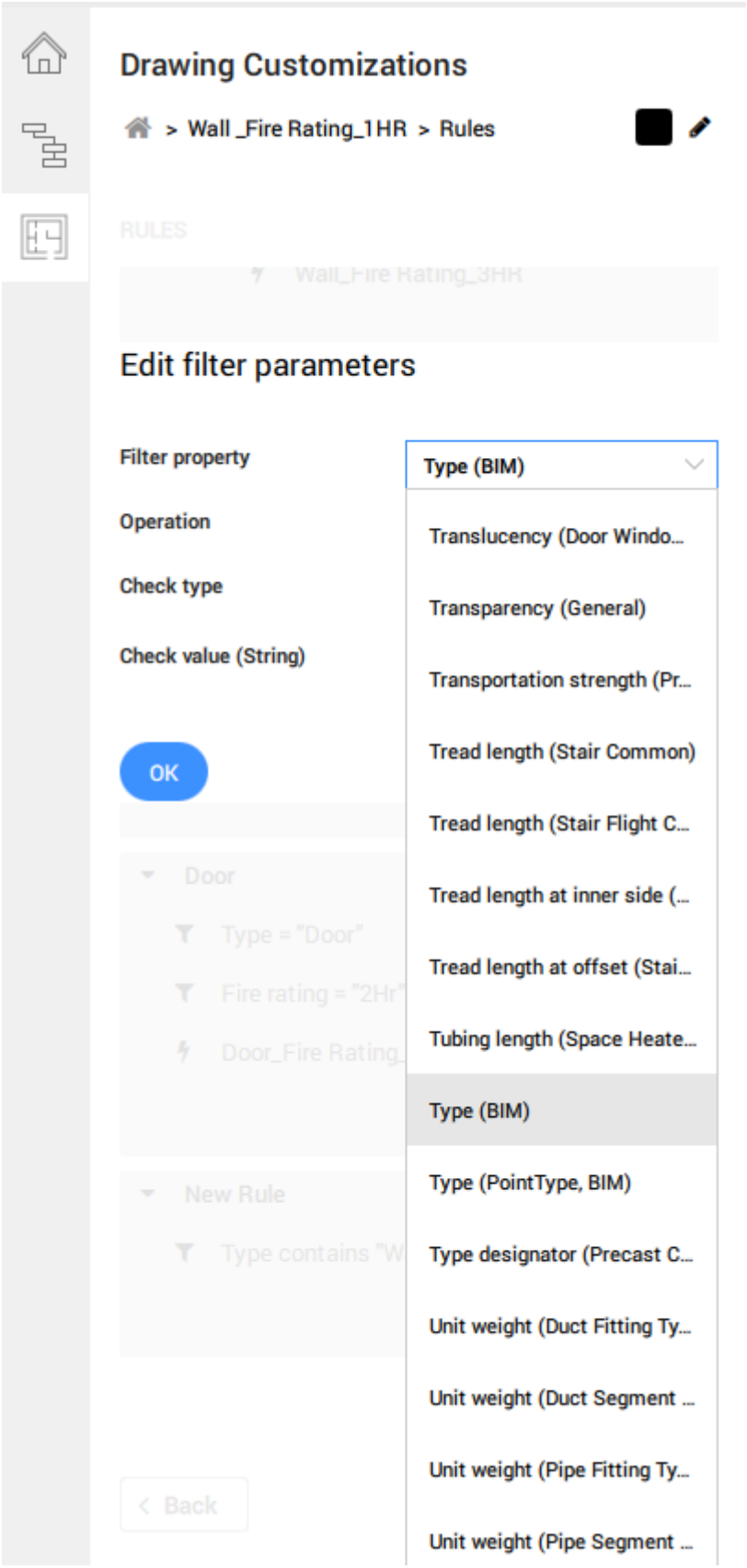


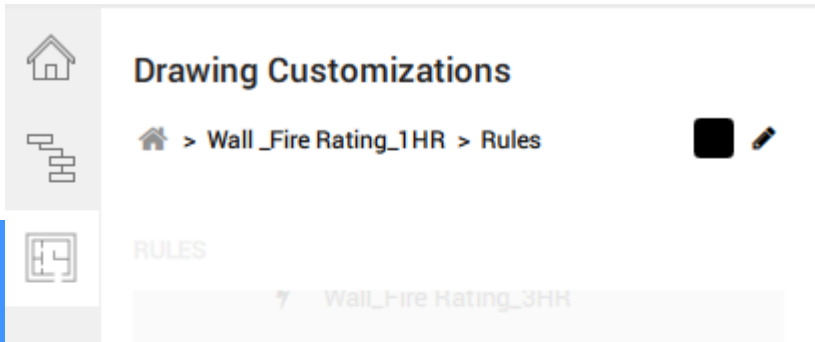
2 Adding and editing a filter parameter will lead you to a prompt, where the options of filter property are determined by the BIMPROPERTIES imported in the **namespace** settings.



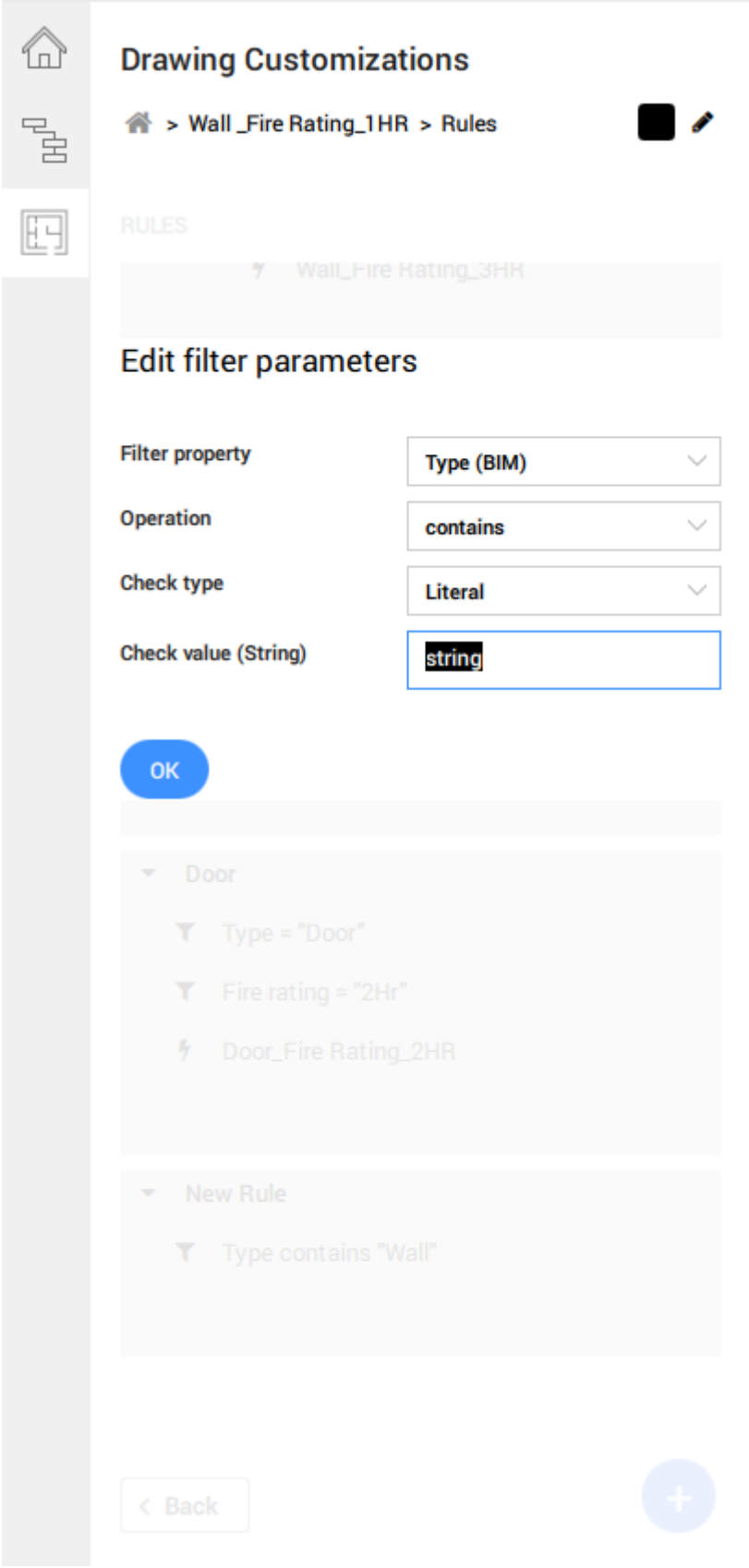
3 The filter **Operation** values are also available as a drop-down list.

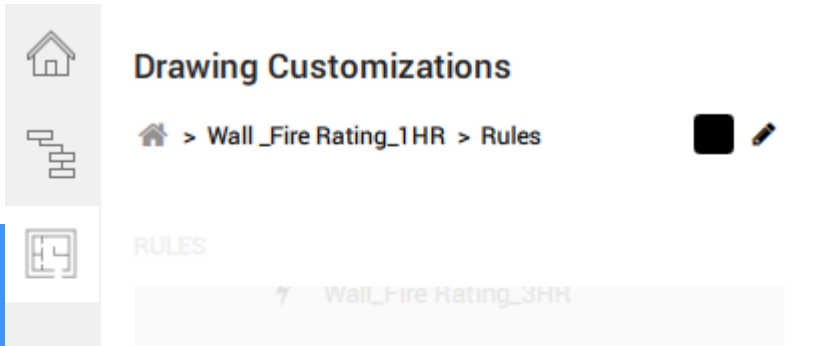
Nota: You can also start typing a certain keyword to search a specific property that you want to filter on.


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR
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"
Door_Fire Rating_
New Rule
Type contains "W
Back

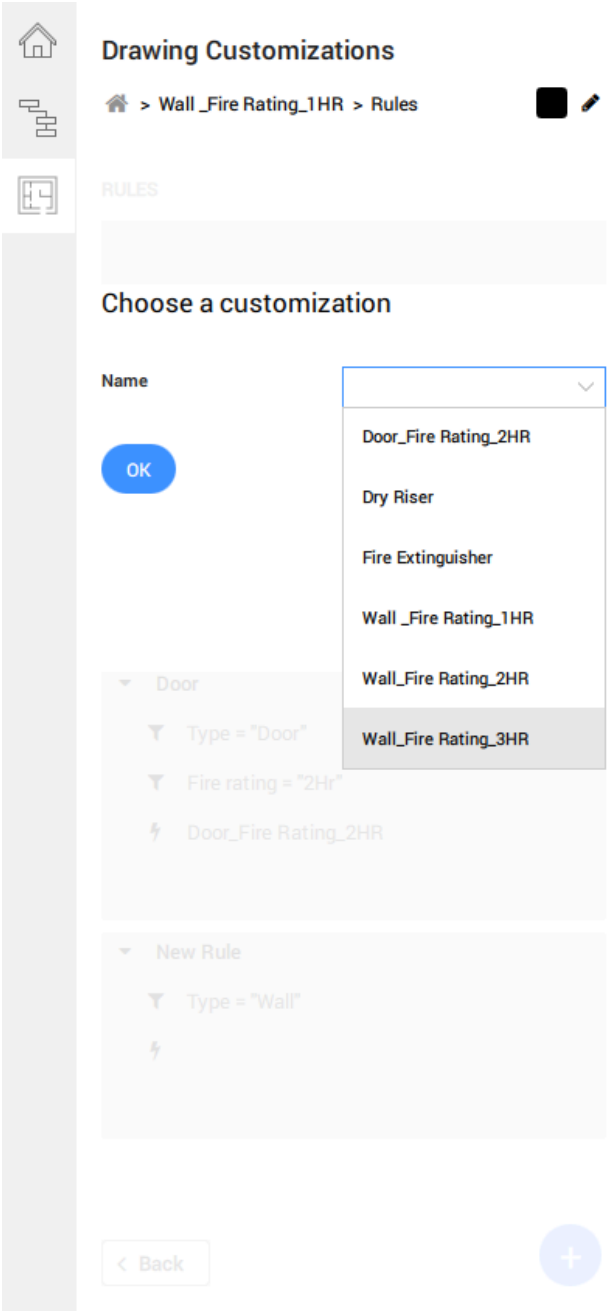

Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR

4 You can specify a string manually in your check value if your **Check** Type is a **Literal**. If you have selected a **Property** value for **Check Type**, your **Check Property** value will refer to the same values found in the **Filter Property** drop-down.


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR
Edit filter parameters
Filter property
Type (BIM)
Operation
contains
Check type
Literal
Check value (String)
string
OK
Door
Type = "Door"
Fire rating = "2Hr"
Door_Fire Rating_2HR
New Rule
Type contains "Wall"
Back


Drawing Customizations
> Wall _Fire Rating_1HR > Rules
RULES
Wall_Fire Rating_3HR

5 To end and legitimize a **filter rule**, the **Entity Customization** value must be added and filled in. The options available in the drop-down selection reflects the existing customizations tab available in the **Entity Customization**page.

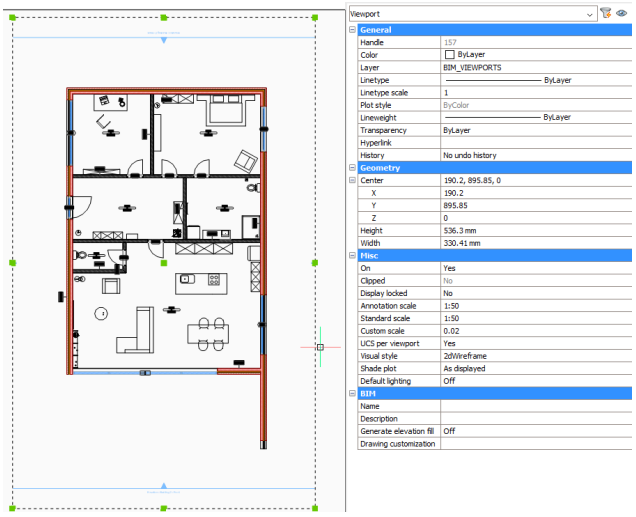


20.7.4.6 Applying templates to viewports on sheets

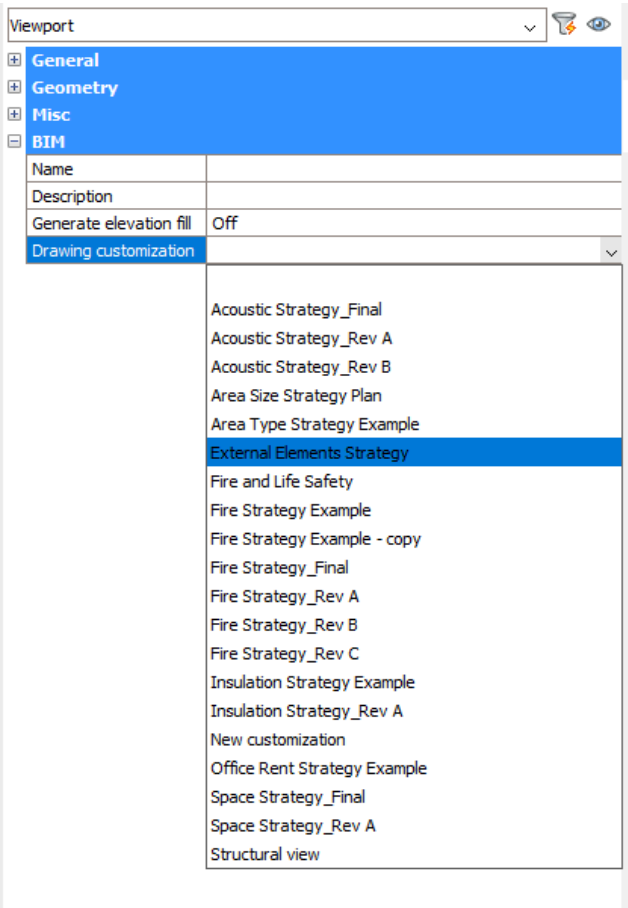
With your **Entity Customizations**, **Filter Rules** and **Styles** all set, your **Drawing Customization** template is now ready.

1 To apply the template, navigate to your sheet drawing with your desired viewport.

2 Open the Properties panel and select the viewport to view its properties.

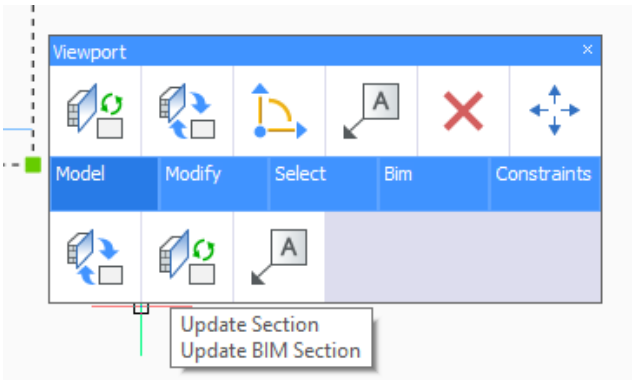


3 Navigate to the **Drawing Customization** property and click on the drop-down selection to the right.



4 Select your desired **Drawing Customization** template.

5 Once done, hover over your selected viewport to access the Quad. Click **Update Section** in the Quad under **Model** tab.



6 You should see a refreshed **Viewport** with the graphical changes as per your **Drawing Customization** settings.

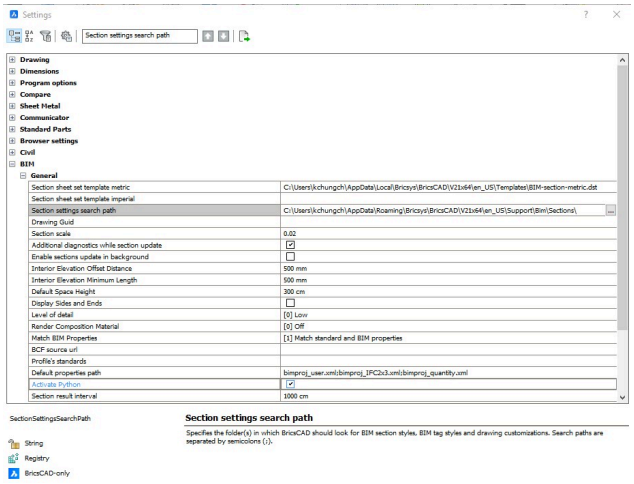
20.7.4.7 Accessing customization templates

By default settings, these templates are stored in the **Customizations** folder within your **Support** folder in the following path, under the **Bim** folder and then **Sections** subfolder:

C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V21x64\en_US\Support\Bim\Sections\Customizations

The **AppData** folder is hidden by default, so you may have to unhide it first. This path can also be accessed by entering SUPPORTFOLDER in the Command line.

You can specify a different path if your Customization files are stored elsewhere in an accessible folder. Simply redefine the path in the **Section settings search path** in the **Settings** dialog box or typing SRCHPATH in the command-line:



New customizations created with the panel will be stored in a **Customizations** subfolder in the specified path. Each customization template is saved as an individual folder which contains in its basic form Settings.dwg, a Filter.json and a New customization.json files. The folder will also contain any external symbol sources which you have created new using the **Create New** buttons.

Settings.dwg contains the values created in the **Style** tab in the form of layers and if you have created any custom properties that you wish to use as a **Filter** parameter, you have to ensure they are also reflected in the Settings.dwg.

Circle 250mm dia XZ.dwg	18/09/2020 11:24	BricsCAD Drawing...	32 KB
DryRiserSymbol.bak	20/10/2020 16:53	BAK File	30 KB
DryRiserSymbol.dwg	20/10/2020 16:53	BricsCAD Drawing...	29 KB
Filter.json	06/11/2020 10:04	JSON File	10 KB
Fire Strategy Example.json	06/11/2020 10:04	JSON File	5 KB
FireExtinguisherSymbol.dwg	20/10/2020 16:53	BricsCAD Drawing...	26 KB
Settings.bak	06/11/2020 12:31	BAK File	212 KB
Settings.dwg	06/11/2020 12:32	BricsCAD Drawing...	212 KB

Nota: As these **Customization** folders are stored locally in the machine you have created it with, you need to ensure the folders and their contents within are copied and pasted into the same location on other machines, should you wish to share it with other users on other machines. This is the current, temporary procedure to do so, as a dedicated function to streamline the sharing process within the **Drawing Customization** tool will be released shortly in an update.

20.7.5 Define detailed section

20.7.5.1 Command

BIMSECTION

20.7.5.2 Define a detailed section from scratch

1 Select **Detail Section** from the ribbon or use the Command line by entering BIMSECTION followed by **D** to choose the detail option.

You are prompted: Select the first corner of the section box or [Based on an existing section/Scale]:

2 Select the first corner of the area you wish to be contained within the detailed section.

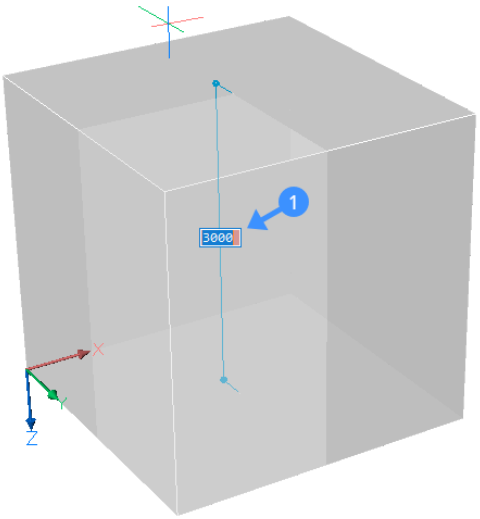
3 **You are prompted**: Select the opposite corner of section box:

4 Select the opposite corner snap to a point or enter the distances. Use the TAB key to switch between dynamic dimension fields.

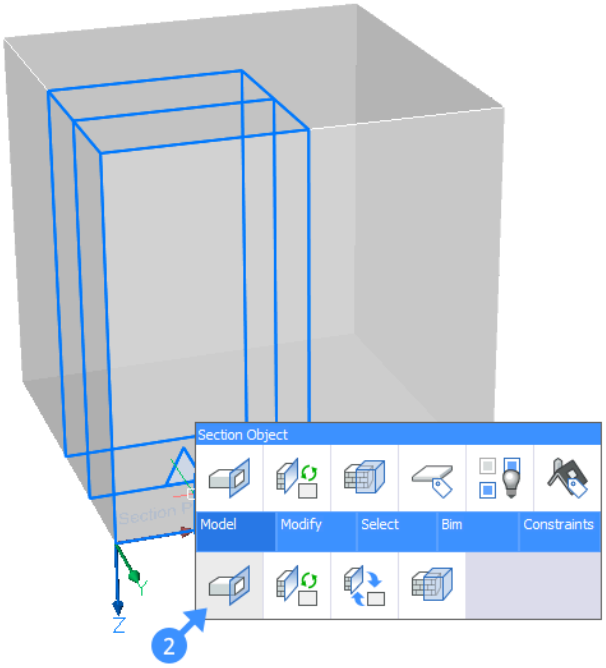
5 **You are prompted**: Select the height of volume box <4000>:

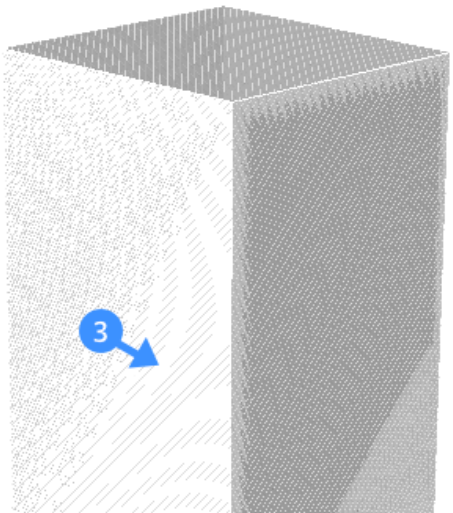
In this case, the default height is 4000mm.

6 To complete the volume, do one of the following:
- Press Enter to accept the default height.
- Specify a point.
- Type a value in the dynamic dimension field (1).



To view the detailed section which has just been defined, hover over the section and select **Clip display** (2) from the Quad **or** double-click the selected section. Now only the detailed section will be visible (3).

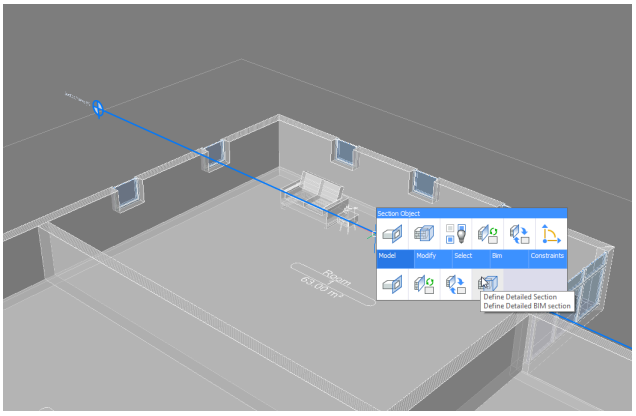




20.7.5.3 Define a detailed section based on an existing section

Here you will be shown how to position the base of your detailed section on an already existing section.

1 Highlight the existing section that you want the detailed section base to be positioned on and select **Detail Section** from the Quad.

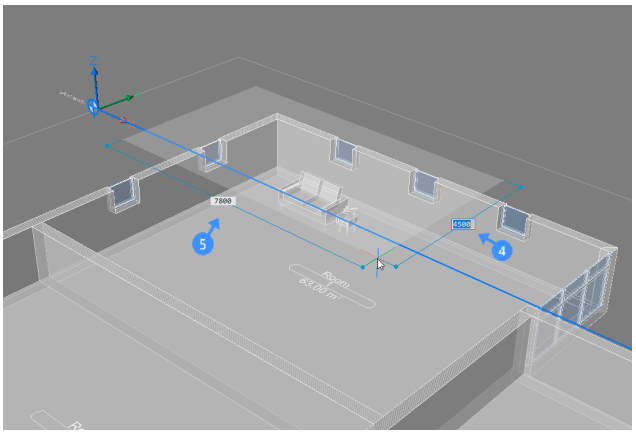


You are prompted: Select first corner of volume box base rectangle:

2 Draw a rectangle around the desired area by typing the values in the dynamic dimension fields.

3 Hit the TAB key to go back and forth between the dynamic dimension fields (4, 5). Your selection will be aligned with the plane of the existing section.

4 Press Enter to confirm the values. The **volume section** box displays dynamically.



You are prompted: Select the height of the volume section box.

5 Press Enter to accept the default height.

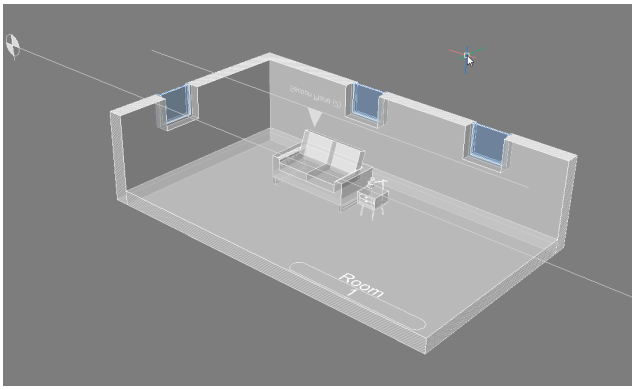
6 Specify a point.

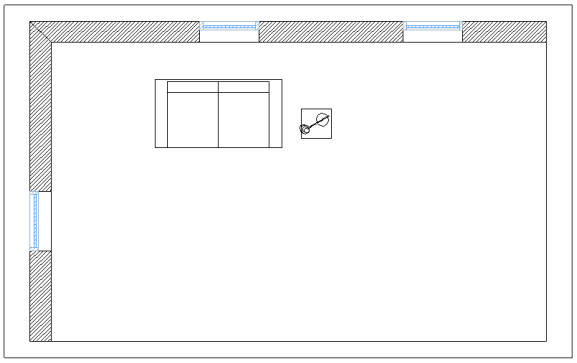
7 Type a value in the dynamic dimension field.

8 The detailed section has now been defined and can be found in the structure browser. Hover over the section and select **Clip display** from the Quad **or** double-click the selected section to view it.

9 To generate the detailed section on a sheet, select the detailed section and select **Generate Section** from the Quad.

10 You can open the section results by clicking **Open Model** when the section plane(s) are selected. The sheet is saved in the same folder containing the model drawing.





If you need more help with how to generate drawings, visit the **Generating Drawings** page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

20.7.6 Define reflected ceiling plan

20.7.6.1 Command

BIMSECTION

20.7.6.2 About

Reflected ceiling plans show a mirror image of the ceiling. These are useful to show the layout of lights and other installations in the ceiling. To produce the reflected ceiling plan a section is created. This section displays the part of the building above the section boundary.

For more information about this command, visit the Command Reference article BIMSECTION.

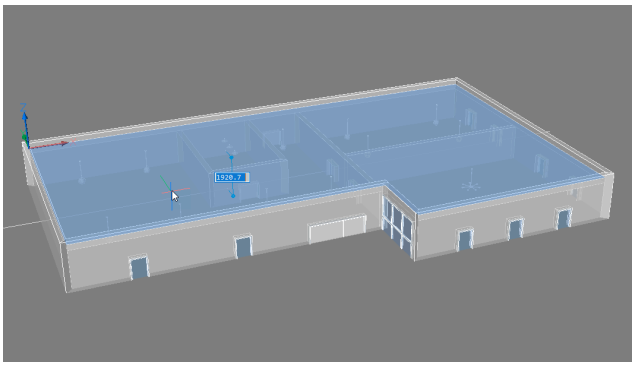
20.7.6.3 Create a reflected ceiling plan

1 Select **Reflected Ceiling Plan** from the ribbon **or** enter BIMSECTION into the Command line followed by

R to choose the **Reflected ceiling** option.

You are prompted: Select a point to place section:

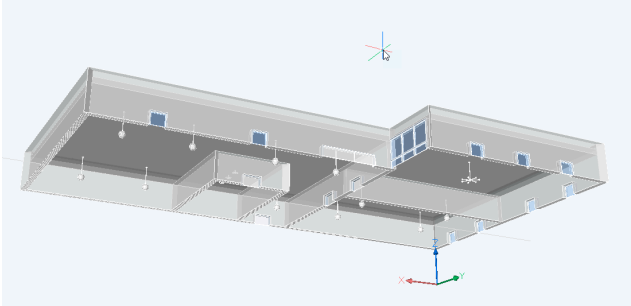
2 Select the ceiling or a face with a plane parallel to that of the ceiling.



You are prompted: Specify distance:

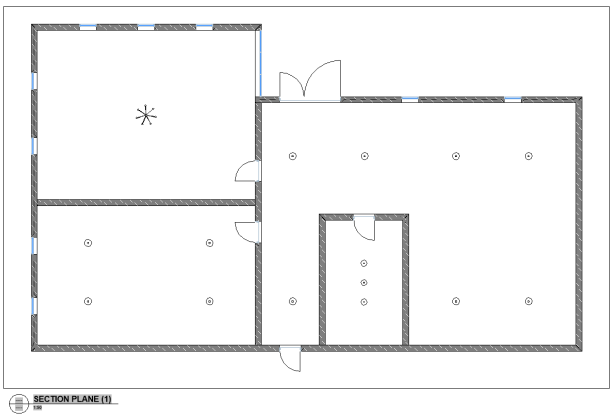
3 Do one of the following:
- Type a value in the dynamic dimension field to offset the section from the initial position.
- Left-click when the section plane is at the location you want.

4 The reflected ceiling section has now been defined. To view hover over the section and select **Clip display** from the Quad.



5 To generate the reflected ceiling plan on a sheet, hover over the section in the model and select **Generate Section** from the Quad.

6 To open the section result drawing click **Open Model**. The sheet is saved to the same folder that contains the model drawing.



If you need more help with how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see how to add tags and linear dimensions.

20.7.7 Detailing with BimPatch

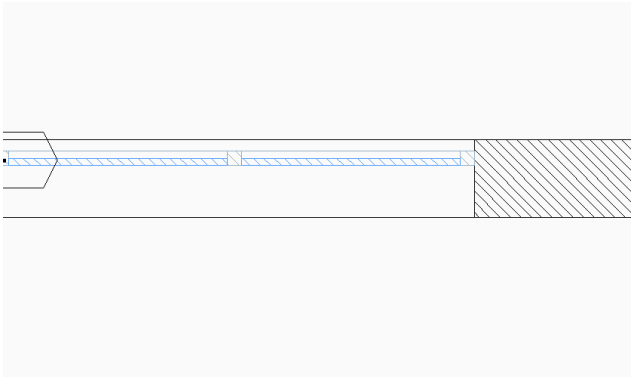
Nota: BIMPATCH cuts out part of geometry and replaces it by a block reference. In this cutting process, entity data linked to the 3D model are lost. Therefore, starting from V21.2, this command is no longer available.

20.7.7.1 Command

BIMPATCH

20.7.7.2 About

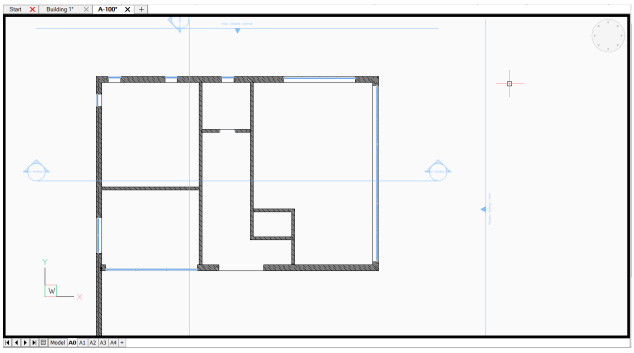
In this procedure, we will add fixtures and a vapor barrier to our window-wall connection. We will detail this window-wall connection.



20.7.7.3 Procedure: detailing a window connection

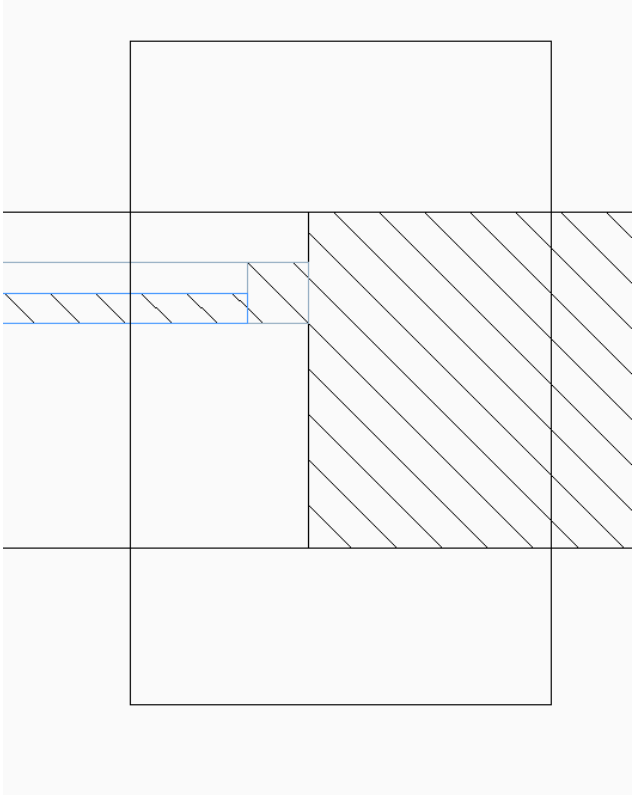
1 Open the section.

2 Go to the model space by double-clicking in the viewport **or**toggle the viewport mode in the **Status** Bar.

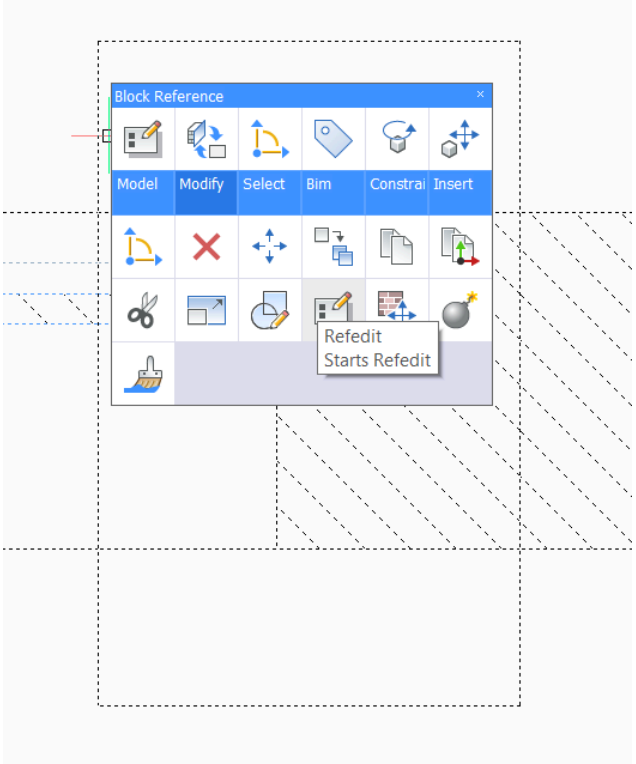


3 Launch the BIMPATCH command in the model space.

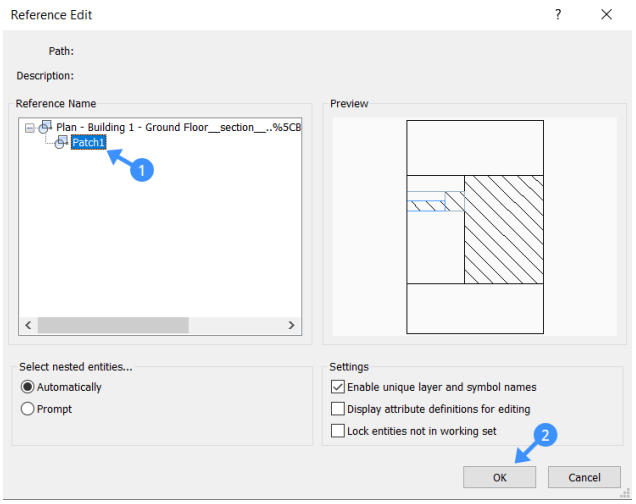
4 Draw the rectangle over the detail and hit enter.



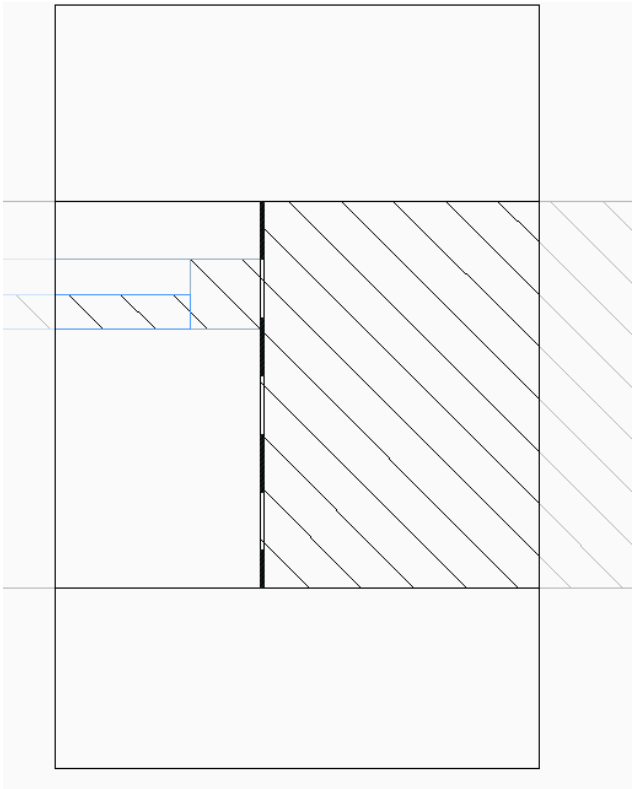
5 Hover over the section object and launch **Refedit** from the quad.



6 A menu appears, select the patch (1) and click **OK** (2).

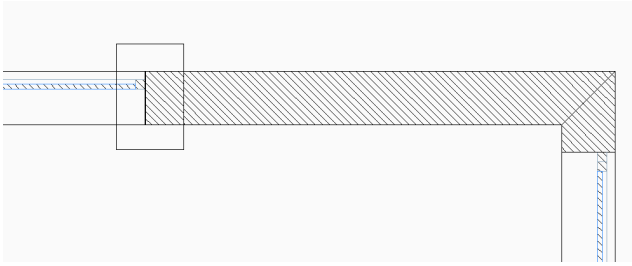


7 Draw the required detail for the window-wall joint. We will add a vapor barrier.

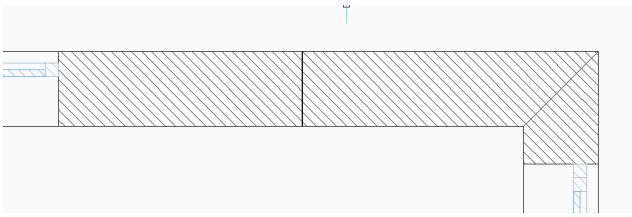


8 Save the changes made in the **RefEdit** session with the **Save** option of the REFCLOSE command.

9 Go back to the paper space. The detailing of the window connection is done. If we now move the window to the left in the 3D model, you will see that the detail you made a patch of moves along.



Nota: If you would have added the vapor barrier in the model space of the section without making a patch of it, the vapor barrier wouldn't be seen as a part of the detail. If you would move your window, the vapor barrier would stay in the same position as you drew it before.



20.7.8 Editing data extraction definition (.dxd) files

DXD (Data Extraction Definition) files are plain text files. You can edit DXD files created by the DATAEXTRACTION command using a text editor such as MS Notepad and use them in the Creating Schedules procedure in the **Project Browser**.

Lines starting with semicolon (;) are comments and ignored on parsing. Empty lines are ignored as well.

The file can contain 4 sections, each section name is written between square brackets, section names are English case-insensitive (ECI) strings. The sections are:

- **Settings**
- **Entity Types**
- **Properties**
- **Filter**

20.7.8.1 The [Settings] section

The section contains values of the following parameters which control the behavior:

- **ExtractFromBlocks** - whether to take entities from blocks into account, considering all nesting levels.
- **ExtractFromXrefs** - whether to take entities from external reference blocks into account.
- **CountXrefs** - whether xrefs are taken into account at all.
- **CombineEqualRows** - whether to group equal rows in the output table; if yes, then the **Count** column is added, which indicates how many equal data rows are included.
- **IncludeNameColumn** - whether the output includes the **Name** column. The **Name** column indicates the source entity type: 3D Solid, 3D Solid Ply, ...

Parameter names are ECI strings. The format is:

ParameterName=value,

The value is either 1 or 0. No spaces should be written around the equal (=) character.

20.7.8.2 The [Entity Types] section

This section contains list of entity types which will be processed. The format of each line can be one of the following types:

EntityType | **DisplayEntityType**

- **EntityType** is the ECI entity type name, such as **Circle** or **AttributeDefinition**. This is the mandatory part of the line.
- **DisplayEntityType** is optional and can be any character string including spaces in any language. If present, this string appears in the **DATAEXTRACTION** dialog and in the output table.

Block | **BlockName** | **DisplayBlockName**

- **Block** is the predefined ECI word, and **BlockName** is case-insensitive name of the block to be processed, including **xRefs** and **MInsert**(s).
- **DisplayBlockName** is optional and has the same meaning and format as **DisplayEntityType**.

Block | *

This is a variant of the previous format meaning that all blocks will be taken into account; if present, then all other lines regarding blocks in this section are ignored. You can use the Filter section to filter the content to be processed.

*

All entity types and blocks will be taken into account, all other lines in this section are ignored then. You can use the **Filter** section to filter the content to be processed.

20.7.8.3 The [Properties] section

This section contains a list of all properties to be included in the data extraction output. The format of each line can be one of the following types: **Entity Property**, **Attribute**, **Drawing**, **Formula** or **Block View**.

EntityPropertyName | DisplayEntityPropertyName | ValueOutputFormat

- **EntityPropertyName** is the ECI name of entity property, e.g. **Lineweight**, **Color**, **Center Y** and so on. Note, all names do not contain spaces except those which represent coordinates of 2D or 3D points or vectors. In the last case, coordinate letters X, Y and Z should be separated by single space from the general property name.

  EntityPropertyName is the mandatory part of the line, the remaining part is optional.
- **DisplayEntityPropertyName** is user-defined and can be in any string including spaces in any language. By default, it is equal to **EntityPropertyName**.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Attribute | AttributeTagName | DisplayAttributeTagName | ValueOutputFormat

- **Attribute** is the predefined ECI word.
- **AttributeTagName** is the ECI attribute tag name as it is stored in the drawing.
- **DisplayAttributeTagName** and has the same meaning as for **EntityPropertyName** (see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Drawing | DrawingPropertyName | DisplayPropertyName | ValueOutputFormat

- **Drawing** is a predefined ECI word.
- **DrawingPropertyName** is an ECI name, which can be one of the following:
  - **Comments**
  - **DrawingRevisionNumber**
  - **FileAccessed**
  - **FileCreated**
  - **FileLastSavedBy**
  - **FileLocation**
  - **FileModified**
  - **FileName**
  - **FileSize**

- **HyperlinkBase**
- **Keywords**
- **Subject**
- **Title**
- **TotalEditingTime**

- **DisplayPropertyName** has the same meaning for **EntityPropertyName** (see above).
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

Formula | EntityPropertyName | DisplayEntityPropertyName=FormulaExpression | ValueOutputFormat

- **Formula** is a predefined ECI word.
- **EntityPropertyName** is a unique name for this formula item.
- **DisplayEntityPropertyName** is an optional output name of the property, if it is omitted, then **EntityPropertyName** is used as the table column title.
- = is a mandatory sign, which must exist even if **DisplayEntityPropertyName** is omitted.
- **FormulaExpression** is the expression in format as in \AcExpr formula fields, which can contain constants such as PI or functions such as SIN (see Using Expressions). An operand in an expression can be a **DisplayPropertyName** enclosed in arrow brackets (<>).

  For instance: =2 * PI * <Radius> calculates the circumference of a circle.
- **ValueOutputFormat** is a string which is used to format the output. The syntax of the string is the same as in fields formatting.

BlockView | BlockViewPropertyName | DisplayBlockViewPropertyName | BlockViewOptions

- **BlockView** is a predefined ECI word
- **BlockViewPropertyName** is a unique name for this block view item
- **DisplayEntityPropertyName** is the output name of the property; cannot be omitted
- **BlockViewOptions** is a string in the following format:

  ViewType;DwgPath;SectionName

  - **ViewType** is one of the following: **Front**, **Top**, **Left** or **Right**; cannot be ommited.
  - **DwgPath** is path to dwg file, may be relative with respect to the dxd file; can be ommited.
  - **SectionName** is the name of the section to extract the section settings from for the block view construction,;can be ommited.

If **DwgPath** or **SectionName** are ommited or not found, default settings will be applied.

An example:

BlockView|BlockViewProperty1|Elevation symbol|Front;../Sections/_SectionSettings.dwg;_BlockViewSettingsSection

*

If present all entity properties are taken into account; all other lines in this section, except Drawing and Formula lines, are ignored.

20.7.8.4 The [Filter] section

Syntax description:

Relational operators:

Operator	Meaning
==	Equal to
!=	Not equal to
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to

- Relational expressions are enclosed in '()' brackets.
- Logical operators AND and OR are case-insensitive and enclosed in '<>' brackets.
- Logical expressions use a prefix notation.
- No spaces are allowed between '<' and a logical operator ('<OR', '<AND').
- Tokens can be separated by zero or any number of spaces, except logical expression begin,
- example:

  <AND (Layer == "MyLayer")(Thickness<=3) >
- Expressions can be split into any number of lines; tokens are not broken.
- Use the **EntityPropertyName** to refer to properties (not the **DisplayEntityPropertyName**).
- Lines can be commented with ';'.
- String values are enclosed in double quotation marks "" .

<OR (Type == Window)(Type == Door)>

The result of the filter in the above example is that only BIM entities classified as windows or doors are included in the data extraction. See the BIMCLASSIFY command to learn more about BIM entity types.

The filter expression is either a single relational one or several ones grouped by logical operations.

Example:

<OR <AND (0) <OR (1) (2)>> <AND (3) (4) (5)> (6)>

where (0),...,(6) are relational expressions such as (Radius >= 3).

```
                OR
                /|\
        -------  | ------
       /         |       \
   AND          AND      (6)
   / \        /  |  \
 (0)  OR    (3) (4) (5)
     /  \
    (1) (2)
```

20.7.9 Editing generated drawings

When a BIM Section is generated, a block containing the drawing generation result is added to the model space of the target drawing, as well as a paper space viewport looking onto this block. The block and the viewport initially are placed so they do not overlap with existing geometry contained in the drawing. You can move this block or viewport to a new location, which will be preserved on subsequent updates.

In the default template drawing which is used to create the generated drawings **Viewlabel** and **NumberBubble** blocks exist. You can place them associated to the selected sheet view. The **Title** block in this drawing contains attributes that will be filled in automatically when the associated **Sheet Set** properties are defined.

A sheet list table can be placed on any sheet of the sheet set. In the current implementation, the table content is not associative yet.

20.7.9.1 Inserting the title block

1 Open the **Drawing Explorer - Blocks** dialog.

2 Double click the **Title** block.

The **Drawing Explorer**dialog closes and the block is attached to the cursor.

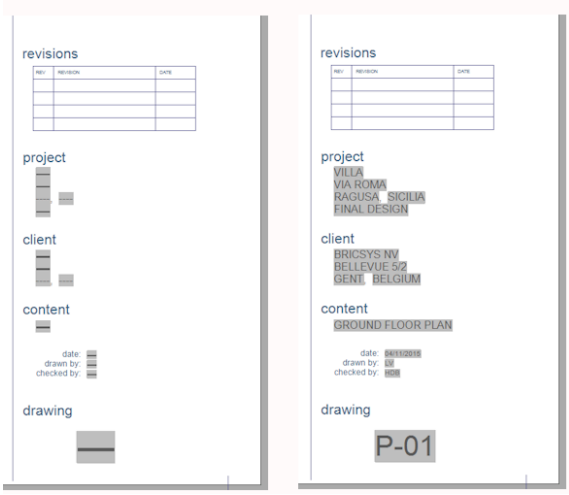
In a metric units project the size of the **Title** block fits the A1 paper size (594 x 841 mm)

In an imperial units project the size of the **Title** block fits the Arch D paper size (24 x 36 inches)

3 Click at the lower left corner of the paper sheet in the layout.

4 Open the **Sheet Sets** panel (SHEETSET command) and define the **Project** and **Custom Properties** available in the context menu.

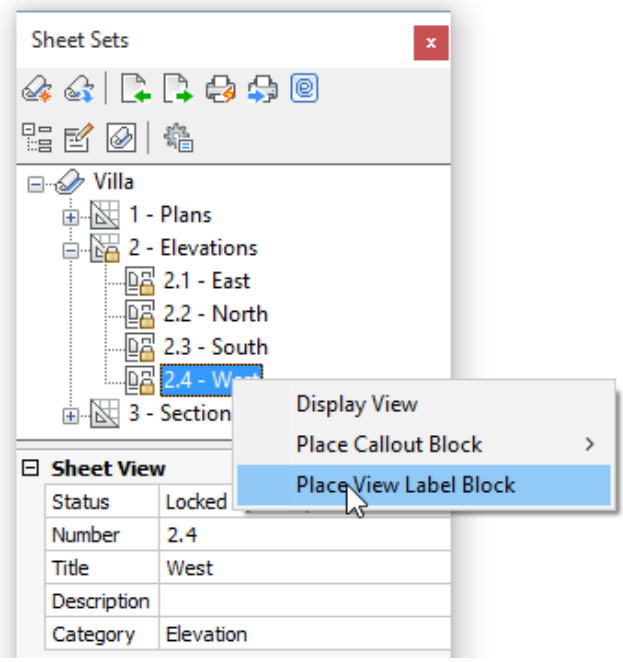
5 Select the **Title** block, then click the **Update Fields...** tool button ( ) in the **Redraw/Regen** toolbar.



20.7.9.2 Inserting a view label for a sheet view

1 Select the sheet view in the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the view label block.



3 Choose **Place View Label Block** in the context menu.

You are prompted: _VIEWLABEL

Units: Millimeters Conversion: 1

Insertion point for block:

4 Specify the insertion point.

5 **You are prompted**: Scale factor for block <1.0>:

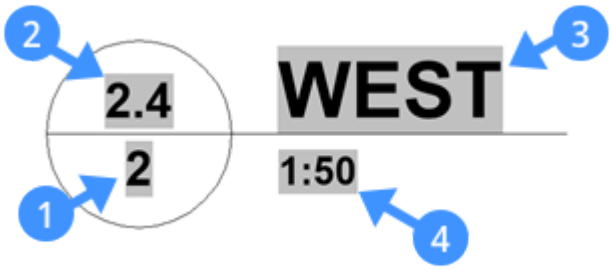
6 Specify the scale factor or right-click to accept the current scale.

7 **You are prompted**: Rotation angle for block <0>:

8 Specify the rotation angle or right-click to accept the current angle.

9 The View Label block is inserted.

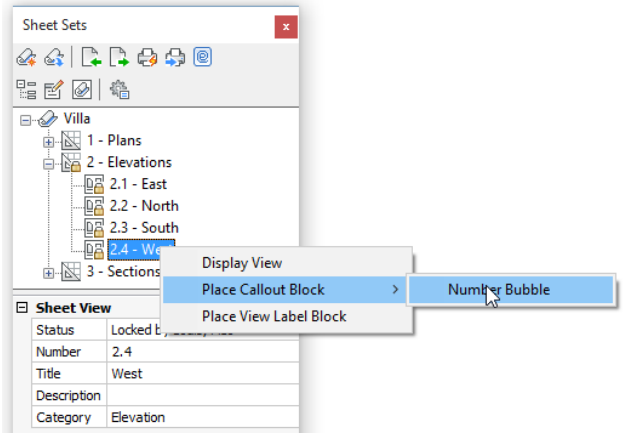
The Sheet Number (1), Sheet View Number (2), Sheet View Title (3) and Viewport Scale (4) of the selected sheet view are filled in automatically.



20.7.9.3 Inserting a number bubble block for a sheet view

1 Select the sheet view In the **Sheet Sets** panel.

2 Right-click the sheet view for which you want to insert the **Number Bubble** block.



3 Choose **Place Callout Block** > **Number Bubble** in the context menu.

You are prompted: _CALLOUT

Units: Millimeters Conversion: 1

Insertion point for block:

4 Scale factor for block <1.0>:

5 Rotation angle for block <0>:

6 Specify the insertion point.

7 **You are prompted**: Scale factor for block <1.0>:

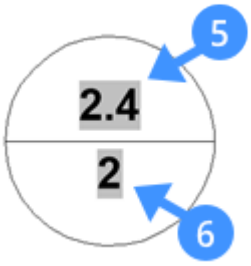
8 Specify the scale factor or right-click to accept the current scale.

9 **You are prompted**: Rotation angle for block <0>:

10 Specify the rotation angle or right-click to accept the current angle.

11 The **Number Bubble** block is inserted.

The Sheet Number (5) and Sheet View Number (6) of the selected sheet view are filled in automatically.



20.7.9.4 Inserting a sheet list table

A **Sheet List** table can be inserted on any sheet of the sheetset. In the current implementation, the table content is not yet associative and therefore will not be updated automatically when sheets are added, removed or renamed.

1 Right-click the sheetset name in the **Sheet Sets** panel and choose **Insert Sheet List Table** in the context menu.

You are prompted: Select insertion point.

2 Specify a point in the drawing.

20.7.10 Elevation hatch

20.7.10.1 About elevation hatch

Generate elevation fill is a property of a section that can display the texture or solid hatch of a composition when a 2D elevation is generated. Hatches can also be generated for other types of sections when activated in the properties panel.

The hatch is generated for every visible boundary on the 2D elevation including boundaries that are not perpendicular to the section. Hatches are generated without perspective corrections. The union of elevation lines can be changed independently from the 'union section' settings in the **Physical Materials** dialog.

20.7.10.2 Make elevations

Nota: **Elevation** here refers to a *Section Plane* in a 3D model that has its **Type** property set to **Elevation**. This can then be used to generate 2D drawings that represent the elevations of your model.

1 Create BIM elevations by using **Bimify**.

2 Select the BIM elevation and choose **Generate BIM Section** in the **Model** command tab in the Quad.

3 Select the BIM elevation and choose **Open BIM Section Model** in the **Model** command tab in the Quad.

The section result displays in the layout viewport of the destination drawing.

4 Select the paper space viewport.

5 The viewport properties display in the Properties panel.

6 Under BIM, set the **Generate Elevation Fill** property to **On**.

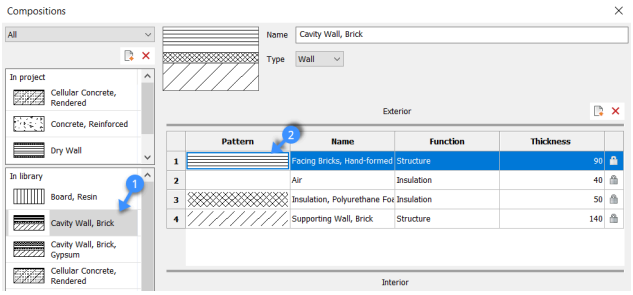
7 Select the viewport and choose **Update BIM Section** in the **Model** command tab in the Quad.

8 **(Optional)** Copy the viewport, then set the **Generate Elevation Fill** property to **Off** to create two versions of the section result, one with and one without elevation hatch.

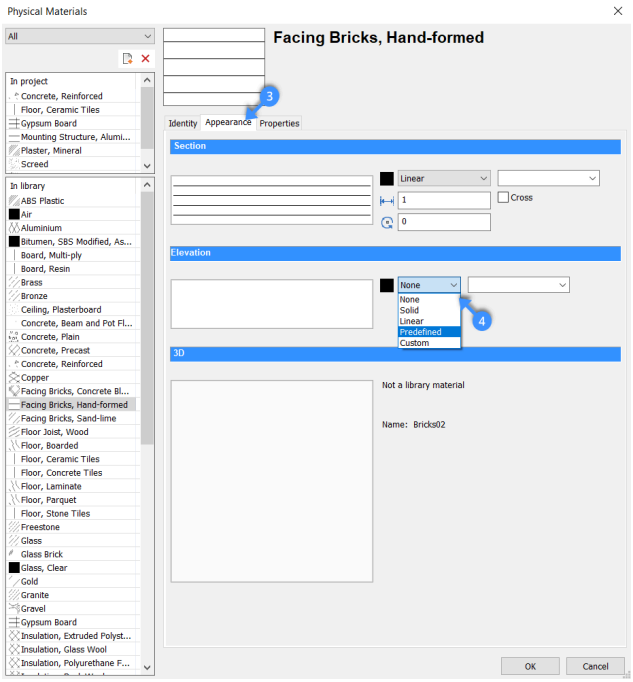
20.7.10.3 Configure the hatch

1 Go to the **Physical Materials** dialog by typing BLMATERIALS or BLCOMPOSITIONS (1) into the Command line.

2 Click on the composition and double click on the pattern of the composition to open the **Physical Materials** dialog (2).

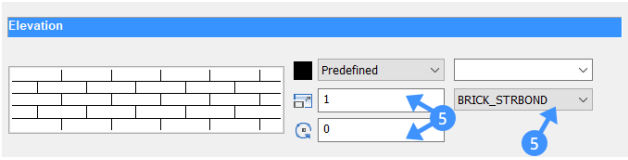


3 To configure the texture or solid fill, go to the **Appearance** tab page (3). The elevation hatch is set by default to **None**. Click the elevation hatch type button and select the type of hatch: **None**, **Solid**, **Linear**, **Predefined** or **Custom** (4).



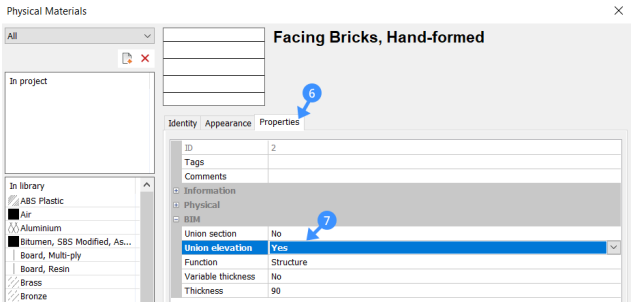
4 Configure the hatch. Define the scale, angle, color and/or layer of the pattern. (5)

Nota: Type a name in the layer field or click the arrow button and select a layer in the list.

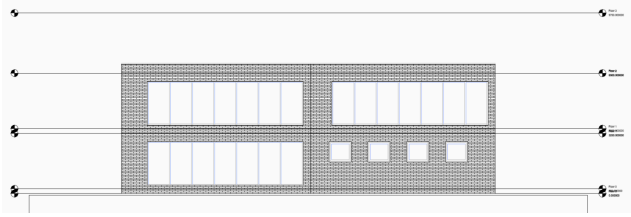


5 To remove connection lines in the elevation between different solids of the material go to the **Properties** tab (6) and set **Union elevation** to **Yes** (7).

Nota: **Union elevation** works independently from the **Union section** setting.



6 After regeneration, the hatch will appear on the elevation.



20.7.11 Enscape integration

20.7.11.1 Commands

STARTENSCAPE, CLOSENESCAPE, ENSCAPEEXPORTSETTINGS, ENSCAPEIMPORTSETTINGS, ENSCAPEBATCHSCREENSHOT, ENSCAPELOADVIDEOPATH, ENSCAPEPANORAMA, ENSCAPESAVEVIDEOPATH, ENSCAPESCREENSHOT, ENSCAPESETTINGS, ENSCAPESTARTVIDEO, ENSCAPESTOPVIDEO, ENSCAPETOGGLELIVEUPDATE, ENSCAPECREATESTANDALONE

20.7.11.2 About

Enscape is a real-time rendering plugin for BricsCAD. With a single click, you can start Enscape in seconds and walk through your fully rendered 3D model. There is no need to export to other programs. All changes in BricsCAD are immediately available for display and evaluation in Enscape.

You can quickly explore different materials and design options to present to clients. With the option to create output renders, videos, and panoramas of your project, you can send the output files to your clients or colleagues, which allows for quick demonstrations. You can edit the changes in BricsCAD, and Enscape will immediately show the changes you have made to the project, even in VR.

20.7.11.3 Enscape system requirements

The minimum requirements to run Enscape:

- Windows 7 64 Bit or higher.
- Intel i5 CPU or higher.
- NVIDIA or AMD GPU with 2GB VRAM.
- Supports OpenGL 4.3.
- NVIDIA GeForce GTX 660/Quadro K2000 and newer.
- AMD Radeon R9 260/FirePro W5100 and newer.
- 4 GB RAM (8 GB recommended).

- 2 GB Video-RAM (4 GB recommended).

For more information about the Enscape system requirements, visit https://enscape3d.com/knowledgebase/system-requirements/.

For more information about specific Enscape features, you can visit their knowledge-base on their website.

20.7.11.4 How to install/uninstall Enscape

1 Download the Enscape-BricsCAD Connection from the Application Store.

2 The installer copies the necessary files to the 'Program Files\Bricsys\Enscape'-folder.

3 Restart BricsCAD.

4 You now have a new tab in the ribbon. Use this to launch Enscape.

Nota: The first time you will start the Enscape-plugin, you will be asked to provide a license key. If you don't have a license key yet, you can start by choosing the trial option.

How to uninstall Enscape

Uninstall the connector from **Add or Remove Programs** in Windows, by searching Enscape and clicking **Uninstall**.

20.7.11.5 Navigate your model in Enscap

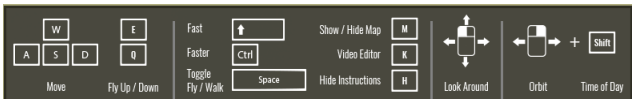
1 Click on **Start Enscape** in the ribbon **or** type in STARTENSCAPE in the Command line.

Changes made in BricsCAD (e.g. deleted furniture, new material, a new section, etc.) can be seen instantly in Enscape.

2 Follow the instructions to navigate through your model.

3 You can switch between **Walk** mode and **Fly** mode by pressing the Space bar.

Type H to hide the instructions.



20.7.11.6 Settings in Enscape

Open the **Enscape Settings** dialog by clicking **Settings** in the ribbon or type in ENSCAPESETTINGS in the Command line. When you change the settings, they will be automatically saved for the user, not in the drawing. There is also an option to create and import presets for the Enscape settings.

You can find more information about saving and importing presets in the following link: https://enscape3d.com/community/blog/knowledgebase/settings-presets/.

The **Enscape Settings** dialog consists of 7 tabs:

- **General**: general **Rendering Style** options, **Depth of Field** and **Exposure** settings.
- **Image**: Contrast and Color settings.
- **Atmosphere**: Horizon/background options, **Fog**, **Clouds** and **Sky Orbs**.
- **Input**: Mouse navigation settings.
- **Advanced**: Spectator options.

- **Capture**: **Image**, **Video** and **Panorama** options.
- **Customization**: Interface options.

20.7.11.7 Render materials in BricsCAD

In BricsCAD BIM, there are two methods to add render materials to your model.

- Directly apply to render materials (without compositions) to the solids/surfaces in your model, using the **Render Materials** panel.
- Apply render materials via compositions, using the **Compositions** panel.

Nota: When render materials are attached to an entity, both applied directly with the **Render Materials** panel and compositions attached, the composition materials will be shown (when the LEVELOFDETAIL system variable = 2).

Thus, if an entity already has a composition and you want to change the render material, don't try to overrule it with the Render Materials panel, but change the composition plies instead.

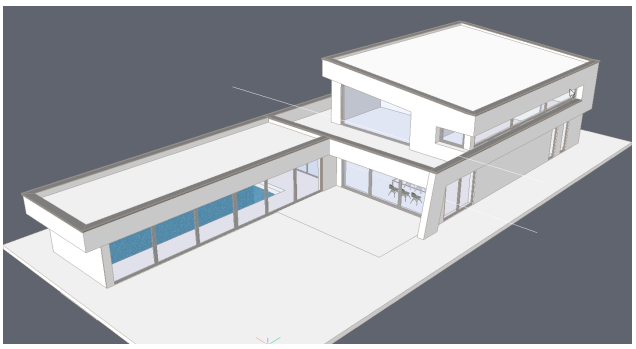
Render materials with the Render Materials panel

- To assign a material to a single entity
  - a Select the material in the **Render Materials** panel.
  - b Click on the entity or press and hold the left mouse button, then drag the material onto the entity.
- To assign a material to multiple entities
  - a Select the entities.
  - b Search the material in the **Render Materials** panel.
  - c Click on the material.

  For more information about render materials, see the article Rendering materials.

Render materials with Compositions

1 Set the LEVELOFDETAIL system variable to 2. The materials of the composition plies will display.



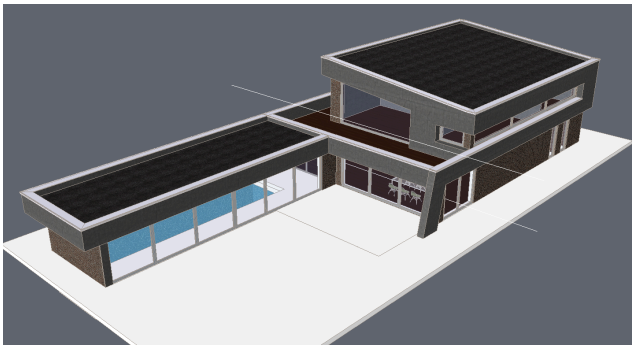


Figure 1: Level of detail = 0,Figure 2: Level of detail = 2

2 Open the **BIM Compositions** panel.

For more information about compositions, see the article BIM compositions.

3 Double-click on the composition **Cavity Wall, Brick**. The **Compositions** dialog box will display.

4 Double-click the material **Facing Bricks, Hand-formed** to edit the ply material. The **Physical Materials** window displays.

5 Click on the material image under **Appearance** > **3D** to change the render material for the ply.

Editing render materials in BricsCAD

1 To edit all the render materials used in your drawing, type in MATERIALS in the Command line.

20.7.11.8 Keywords in Enscape

Grass

Any time the word ‘Grass’ is used in a material name, grass will be rendered in Enscape. You can use diffuse colors to choose options of green, but also texture maps can be used as well.

1 Go to the **Materials** dialog box.

2 Click on **Add New Material**.

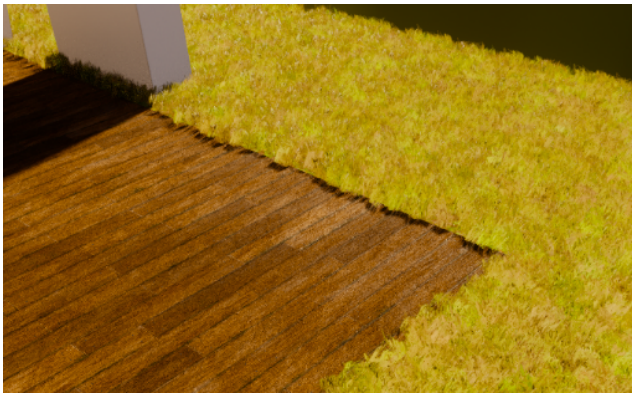
3 Give the new material a name that includes the name 'Grass', for example, ‘Grass_ground’.

4 Close the **Materials** dialog.

5 Select the solid that you want to render in grass.

6 Open the **Properties Panel**.

7 Change the property **3D Visualization** > **Material** from **ByLayer** to **Grass_ground**.



Water

Any time the word 'Water' is used in a material name, water will be rendered in Enscape. You can also create your own water material by including the word 'Water' in the material name and give it a diffuse color in a shade of blue.

1 Select the solid that you want to render in water.

2 Open the **Render Materials** panel and choose the folder named **Water**.

3 Drag and drop material **Water Blue Two** into the drawing.

20.7.11.9 Saved views in Enscape

1 Navigate in the BricsCAD drawing until you get a view that you want to save.

Nota: Make sure your viewing style is in **Perspective** to have a more accurate saved view in Enscape. You can check this in **Properties Panel** > **View** > **Perspective** > **ON**. And close the **Properties Panel** afterward.

2 Click on **Create View** in the Enscape ribbon or type -VIEW in the Command line.

You are prompted: View [? to list/Delete/Orthographic/Restore/Save/Window]:

3 Type **S** and press Enter to save the view.

You are prompted: Save view as:

4 Name the view and press Enter to confirm.

5 You can delete or adjust existing views by clicking **Views** in the Enscape ribbon **or** type in VIEW in the Command line.

6 In the Enscape window: on the right-hand side of the window, you can find all the saved views in the drawing.

7 Click on the view you want to see to switch between views.

20.7.11.1 Render images with Enscape

0

1 Navigate through the Enscape window.

2 **Or** you can click on a saved view on the right side of the Enscape window. The saved view will display.

3 Go back to BricsCAD and select **Render Image** in the Enscape ribbon to render the current view or type in ENSCAPESCREENSHOT in the Command line.

4 Select **Batch Rendering** in the ribbon, or type in ENSCAPEBATCHSCREENSHOT in the Command line, to render all the saved views to a folder.

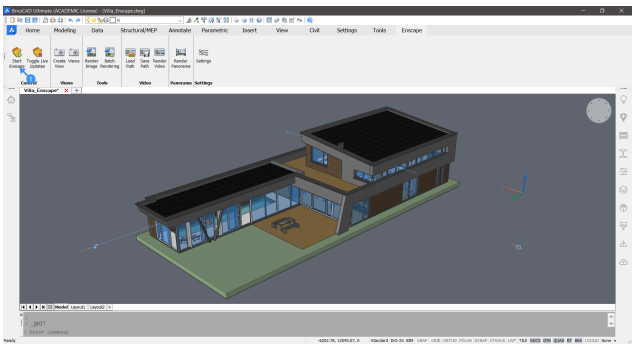
5 Specify a destination file to save the rendered image and click **Save**.

The rendered image is now created.

20.7.11.1 Render a video with Enscape

1

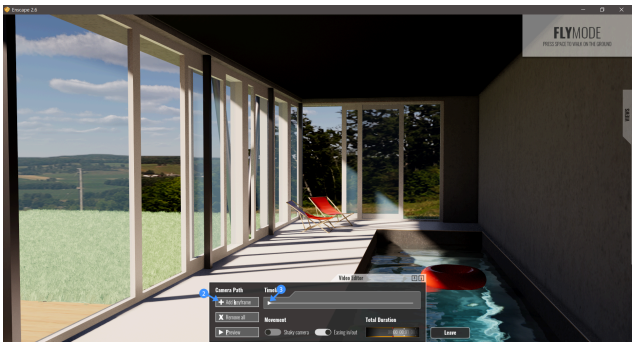
1 Click on **Start Enscape** in the ribbon or type in STARTENSCAPE in the Command line.



2 In the Enscape window: type K to open the **Video Editor**.

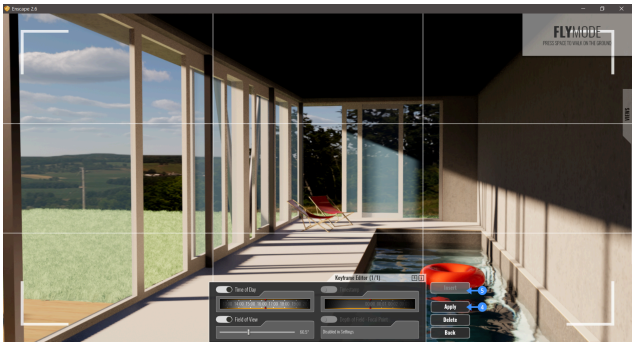
3 Click **Add Keyframe** (2) to add the first keyframe.

4 Click on the triangle to open the keyframe settings (3).



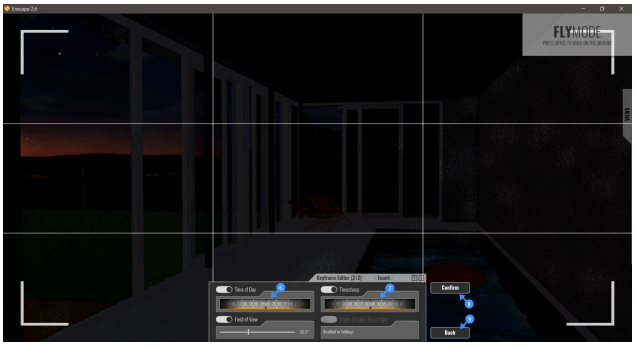
5 Change the parameters as required.

6 Click **Apply** (4).



7 Click **Insert** (5) to add a second keyframe.

8 Change the **Time of Day** to 7 pm (6) and the **Timestamp** to 8 seconds (7).



9 Click **Confirm** (8).

10 Click **Back** (9).

Nota: Before you click **Leave** to exit the **Video Editor**, your video path will not be automatically saved. So, before you exit your **Video Editor**, you should save your video path using the command ENSCAPESAVEVIDEOPATH.

11 Go back to BricsCAD and click on **Render Video** in the Enscape ribbon. Specify a destination file and

click **Save**. The video is now rendered and saved.

20.7.11.1 Export a standalone .exe

2

The Executable export function provides the same experience and quality that Enscape does, but does not require Enscape and BricsCAD to run.

1 Click **Export Standalone** in the Enscape ribbon or type in ENSCAPECREATESTANDALONE in the Command line.

2 A new dialog will pop up asking you where to save your standalone file. Choose your destination file, name your standalone and click Save.

20.7.12 Generating Drawings

20.7.12.1 Commands

BIMSECTION, BIMSECTIONUPDATE, BIMSECTIONOPEN

20.7.12.2 About BIMSECTIONUPDATE

BIMSECTIONUPDATE generates section drawings from BIM Section entities. The command can be used either by selecting a BIM Section entity in the 3D model or by selecting an existing section in a generated drawing. The result is placed in a new drawing, saved to the same folder as the model drawing unless a different folder is specified in the **Destination File** property of the BIM Section entity.

For more information about this command, visit the Command Reference article BIMSECTIONUPDATE.

20.7.12.3 Procedure: generating BIM sections

BIM Sections can be generated in 2 ways:

- In the model space.
- Using the BIM **Project Browser**.

Model space

1 Select one or more BIM section entities.

2 **(Optional)** Edit the section properties (e.g. **Scale**, **Name**, **Destination File**).

3 Choose **Generate Section** in the **Model** command tab of the Quad.

BricsCAD reports the number of generated sections and their name(s) in the Command line e.g.:

8 section(s) were generated:

D:\Bricsys Reports\V16\Villa\Elevations.dwg

D:\Bricsys Reports\V16\Villa\Plans.dwg

D:\Bricsys Reports\V16\Villa\Sections.dwg

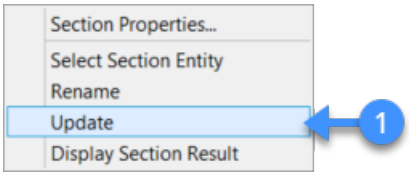
In the example above 2 plans, 4 elevations, and 2 cross-sections were generated and exported to 3 drawings.

Project browser

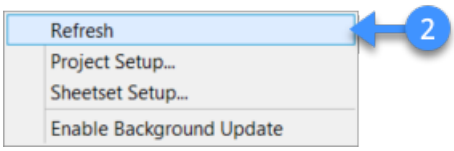
1 In the **Sections** segment of the **Project Browser**, right-click the section.

2 **(Optional)** Choose **Select Section Entity** to change its properties in the properties panel.

3 Choose **Update** (1).



4 Choose **Refresh** in the project browser menu (2).



A new section is generated in the **Sheets** segment of the **Project Browser**.

Notes	• BIM_SECTIONS_PREVIOUS layer: the result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. This is frozen by default. • BIM_VIEWPORTS layer: generated viewports are placed on the dedicated BIM_VIEWPORTS layer if desired. • The **SheetSet manager** automatically and silently adjusts the file paths after the project folder has been relocated if the same folder is used for all files.

20.7.12.4 Open BIM sections

BIMSECTIONOPEN navigates between the model drawing containing a BIM Section entity and the corresponding generated drawing. The command works bi-directionally.

For more information about this command, visit the Command Reference article BIMSECTIONOPEN.

BIM Sections can be opened in different ways:

- From the 3D Model.

- Using the BIM **Project Browser**.

From the 3D model

To open a 2D document from the 3D model:

1 Select one or more BIM sections.

2 Choose **Open Model** in the **Model** command group of the Quad.

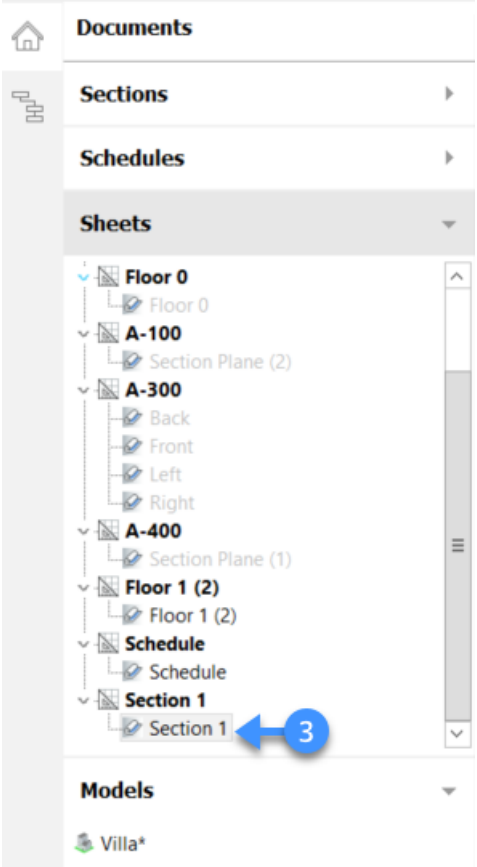
3 The 2D section drawing(s) open(s).

Project browser

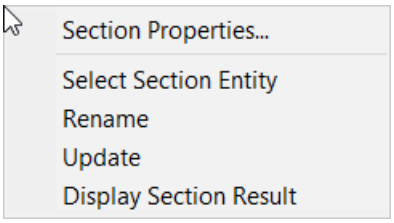
To open the 2D section drawings from the **Project Browser**, do one of the following:

- Double click the sheet view in the **Sheets** segment of the **Project Browser** (3).

  The 2D section drawing opens with the selected sheet view.



- In the **Sections** segment of the **Project Browser**, right-click the section and choose **Display Section Result** in the context menu.

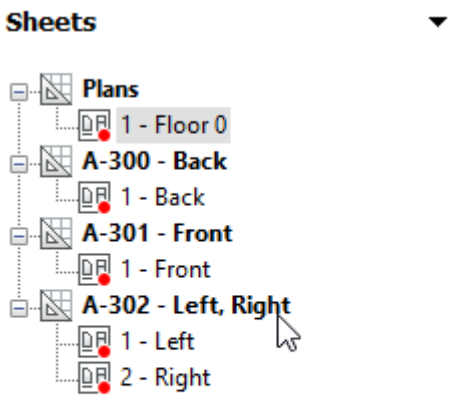


20.7.12.5 Update BIM sections

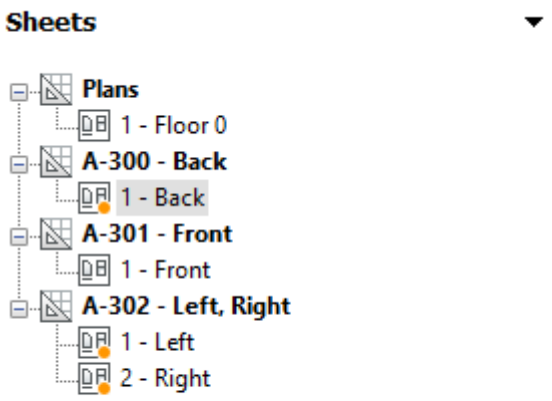
BIM Sections can be updated in different ways:

- In the 3D model.
- In the generated 2D documentation file.
- In the BIM **Project Browser**.

When the 2D documentation is no longer in sync with the 3D model, the corresponding sheet views display a red mark in the **Sheets** segment of the **Project Browser**.



Type BIMIFY command, then the corresponding sheet views display an orange mark in the **Sheets** segment of the **Project Browser**.



20.7.12.6 Update BIM sections in your 3D model

To update BIM Sections via your 3D model using the same process as used to generate views.

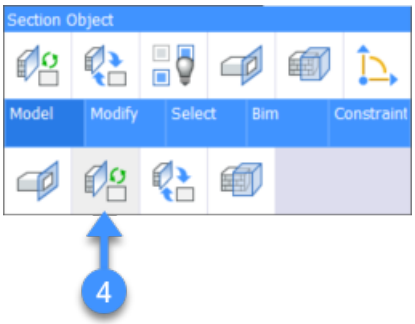
Update BIM sections in the 2D documentation file

A red frame around the section result indicates that the BIM Section generated block is no longer in sync with the 3D model and needs to be updated. This can be done in **Model** space or in the **Layout**.

Update a BIM section in model space

1 Place the cursor over the section block to update.

2 Choose **Update Section** in the **Model** command group in the Quad (4).

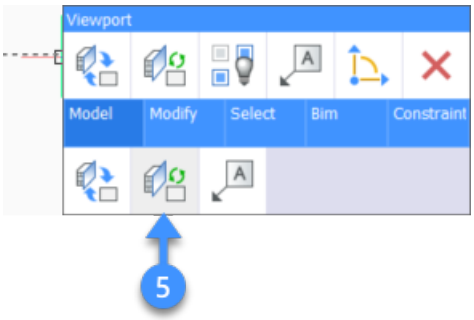


Update a BIM section in a layout

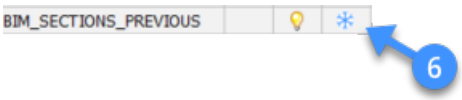
1 Click the **Layout** tab.

2 Place the cursor over the viewport of the BIM Section to update.

3 Choose **Update Section** in the **Model** command group of the Quad (5).

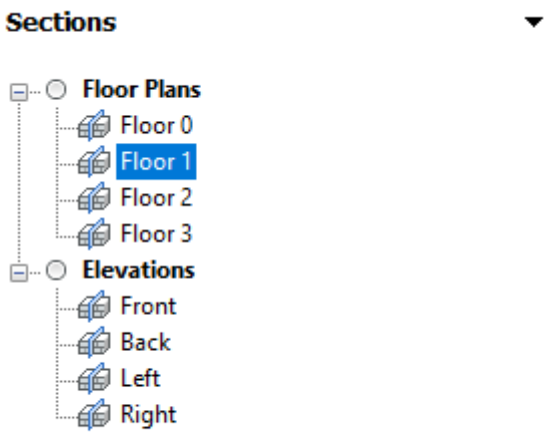


The result of the previous section generation is stored on the dedicated BIM_SECTIONS_PREVIOUS layer. Thaw the layer (6) to check the modifications.



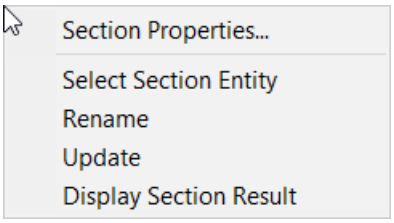
The content of the BIM_SECTIONS_PREVIOUS layer displays in magenta.

Update BIM sections in the project browser



1 Right-click on a section in the **Project Browser**.

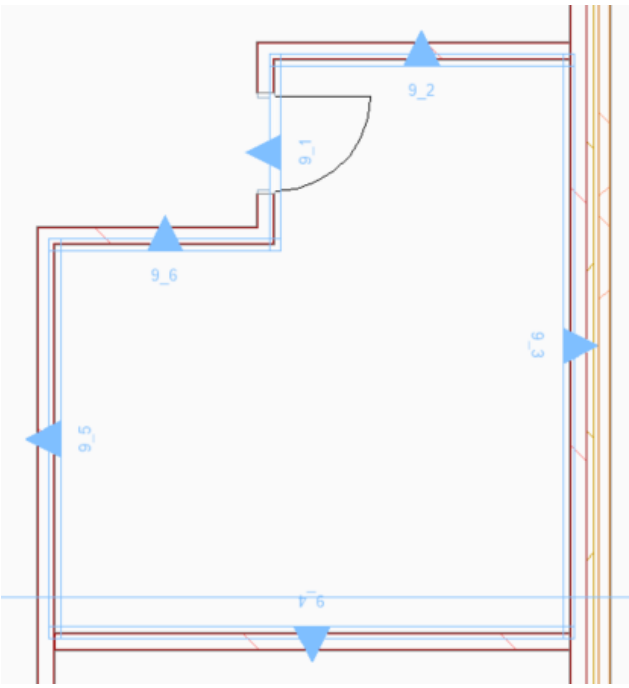
2 Select **Update** in the context menu.



20.7.12.7 Section indicators

A generated plan and reflected ceiling plan provide section indicators for interior elevations and detail sections if they intersect with the section plane.

The section indicators are stored on the dedicated BIM_SECTIONS layer. To change the visibility of indicators, turn the layer ON or OFF.



20.7.13 Working with Point Clouds in BIM

20.7.13.1 Commands

POINTCLOUDDETECTFLOORS, POINTCLOUDFITPLANAR, POINTCLOUDPROJECTSECTION

20.7.13.2 About POINTCLOUDDETECTFLOORS

The new POINTCLOUDDETECTFLOORS command generates volume sections for each floor found in a point cloud representing a building. The detection is based on regions of points with similar Z-coordinates. The generated volume sections can help in navigating point clouds of buildings.

Section planes work on point clouds as well, they can be used to show parts of point clouds. The difference between point cloud crops and section planes is that point cloud crop only clips the point cloud while section planes will clip all geometry in your drawing.

If the point cloud building has multiple floors, there will be multiple section volumes.

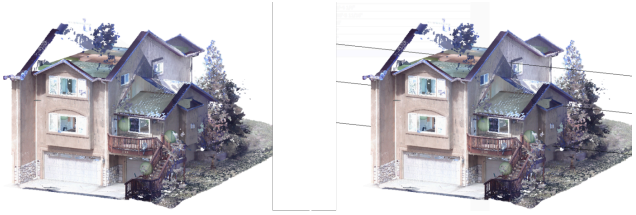
For more information about this command, visit the Command Reference article POINTCLOUDDETECTFLOORS.

Detecting floors from Pointclouds

1 Type POINTCLOUDDETECTFLOORS in the Command line.

2 **You are prompted**: Operate on full point cloud or a selection? [Full/Selection]
   - **Full**: The command will search the entire point cloud for floors and generates a section for each of them.
   - **Selection**: The command only runs on the selected area.

3 Press Enter.



Nota: This volume selection is a detail section of the selected building. You can manipulate the section box if you want to include more or fewer points from the point cloud using the green grips.

20.7.13.3 About POINTCLOUDFITPLANAR

POINTCLOUDFITPLANAR is used to semi-automatically make 3D geometry based on a point cloud scan. It will create a planar surface or solid after a selection of one point in a point cloud. The points that seem to be in a plane are never exactly in one plane, therefore a threshold value is set as a property of the point cloud entity. This also works in bubble view.

Select planes in the point cloud to generate 3D solids based on the given geometries. These 3D solids can be further used in the process of creating a full BIM model.

For more information about this command, visit the Command Reference article POINTCLOUDFITPLANAR.

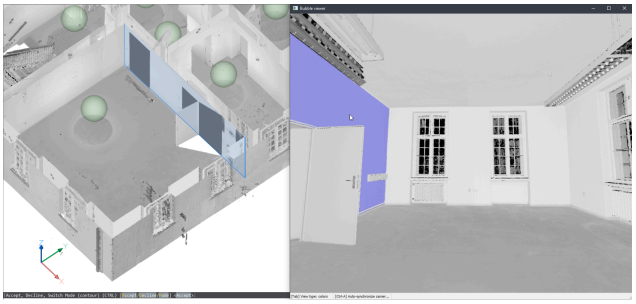
Creating 3D solids from the bubble view

If the bubble viewer is open before launching the command, BricsCAD expects you to select in the bubble viewer. The cursor will give you a preview of the direction of the plane. When you click you get a preview in both bubble view and model view. You can toggle between 2 shape representations using the CTRL-key.

1 Double-click a green point cloud bubble to open the **Bubble viewer**.

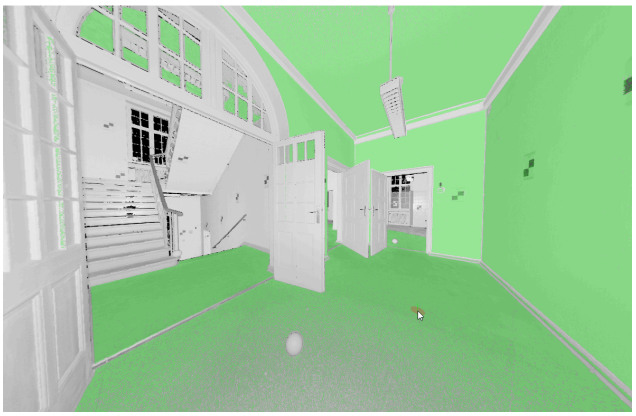
2 Type POINTCLOUDFITPLANAR in the Command line.

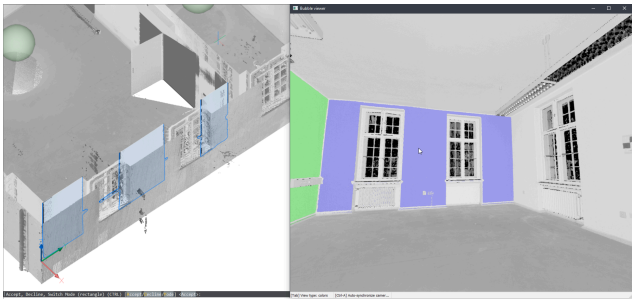
3 Select the desired wall.



4 Press A to accept the selection.

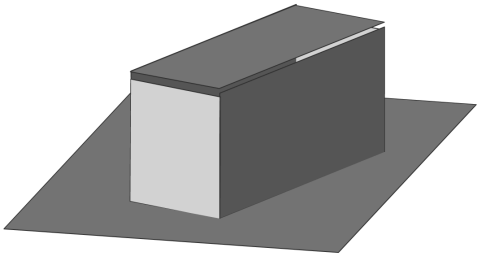
5 **(Optional)** Select more walls.

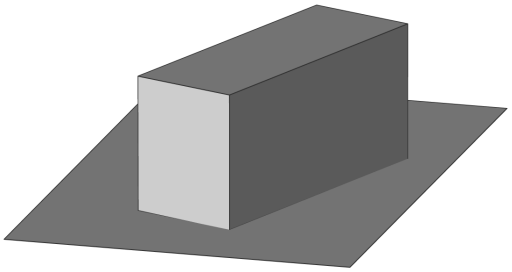




6 Close the **Bubble viewer**.

The surfaces are created and you can edit them to closed spaces with LCONNECT or other connecting tools.





Create 3D entities from the model space

You can also use this command in the model space when the bubble viewer is not open. BricsCAD will ask you to select a point of the point cloud in model space. Depending on the size of the cropped point cloud, it takes more time but it has 2 advantages by searching multiple scan positions:

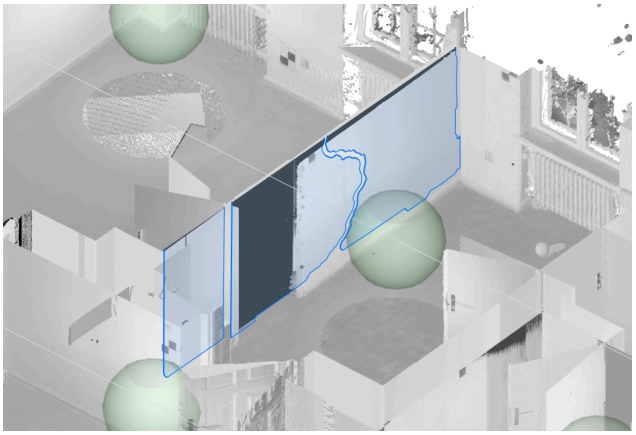
- It can create larger surfaces where only parts are visible in each scan position.
- It can detect wall and slab thickness since it can take the opposite surface into account.

1 Type POINTCLOUDFITPLANAR in the Command line.

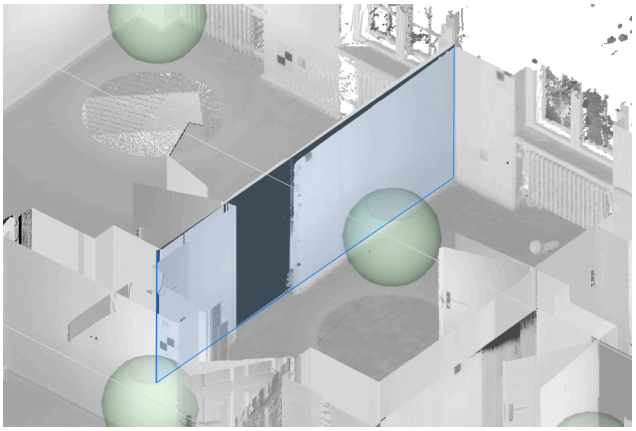
2 Select a wall where you want to create a plane or solid.

3 You are prompted: Accept, Decline, Switch Mode (contour) (CTRL) [Accept/Decline/Mode]:
   - Press A to create the wall.
   - Press D to decline.
   - Press M or CTRL repeatedly to switch between different modes.

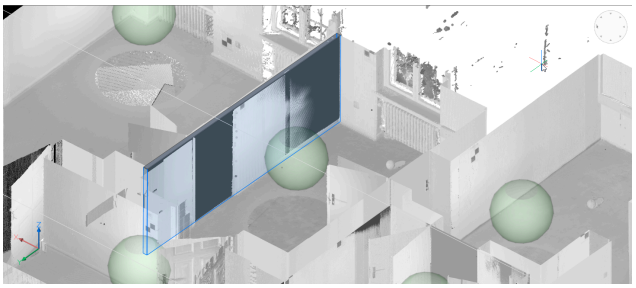
Contour mode will create the best fitting plane:



Rectangular mode will create a rectangular plane around region of planar points:



Solid mode will create a solid from two parallel point cloud planes:



Nota: If the point cloud exists out of multiple rooms, you can use the solid mode to automatically create solid walls.

4 Press A to create the wall.

These solids can be the base for a more complex BIM Model. Use modeling commands to complete the building and add BIM data with **Bimify**, **Automatch** and **Propagate** to quickly transform your point cloud into a complete BIM Building.

It may be helpful to turn **on** the **Point Cloud Closest Point Esnap** setting. This snap setting significantly improves your ability to select relevant point cloud points.

20.7.13.4 About POINTCLOUDPROJECTSECTION

POINTCLOUDPROJECTSECTION automatically generates a raster image with optional contour lines out of a defined section box. These lines can be easily manipulated into a detailed 2D drawing.

The command enables you to detect walls from the volume section of a point cloud based on a variety of wall detection options. You can create volume sections automatically for each floor in a building with POINTCLOUDDETECTFLOORS. You can use these sections to generate 2D lines to create a 2D floorplan or a vertical section. This is a background process and multiple sections can be processed in a queue. This way it is possible to run this command in full resolution on all sections.

For more information about this command, visit the Command Reference article POINTCLOUDPROJECTSECTION.

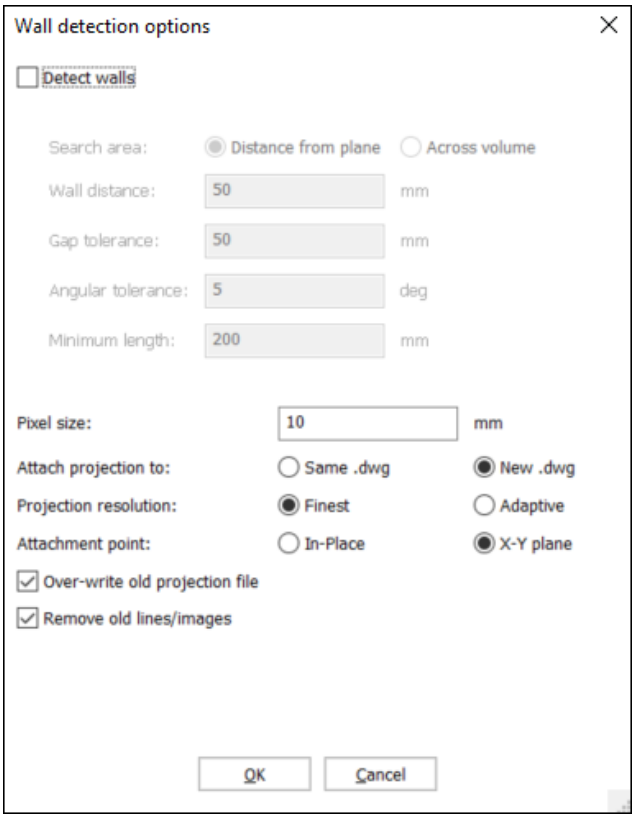
20.7.13.5 Procedure: creating a raster image from a point cloud

1 Type POINTCLOUDPROJECTSECTION in the Command line.

2 Select a **Section** volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section. Turn **Detect Walls** off.



4 Click **OK** to generate the raster image.

Point Cloud scan by Realistic Visual styles.

20.7.13.6 Procedure: detecting walls from a point cloud and generate in 2D

Making horizontal sections from point clouds

1 Type POINTCLOUDPROJECTSECTION in the Command line.

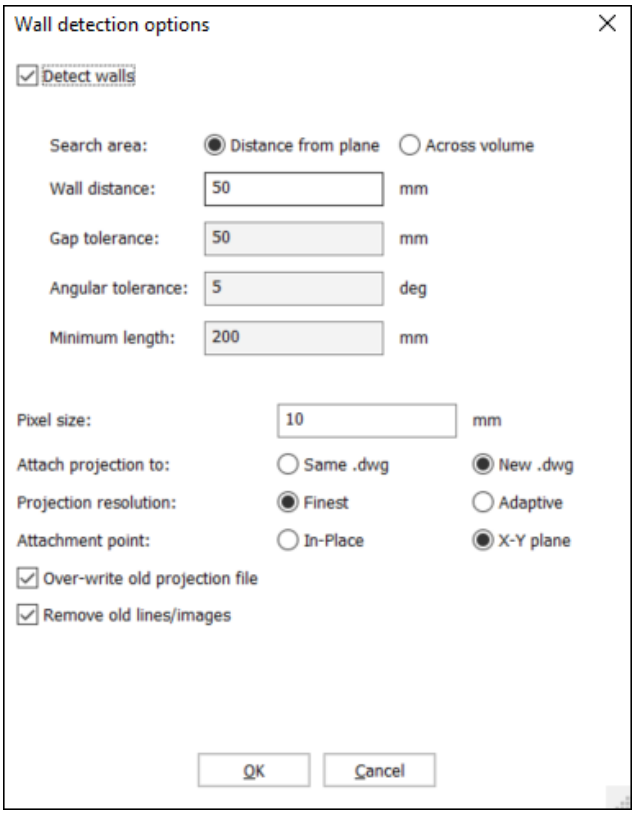
2 Select a section volume and press Enter.

3 The **Wall detection options** dialog appears.

Edit the options to customize your projected section.

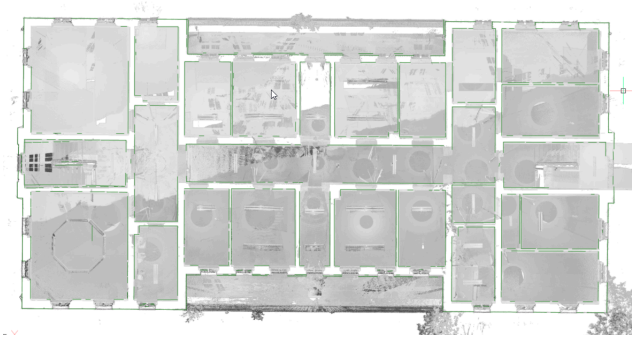
Nota: See the command reference page for more information about the customization options;

Nota: Turn on **Detect Walls** to generate the green lines as seen in the image below.



4 Press Enter to create the projection.

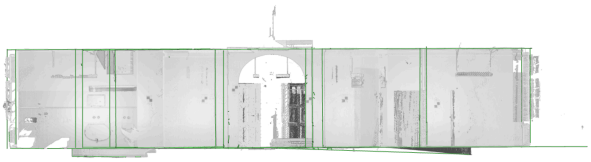
Nota: If there are some missing lines, you can draw them yourself by tracing the background image.



Making vertical sections from point clouds

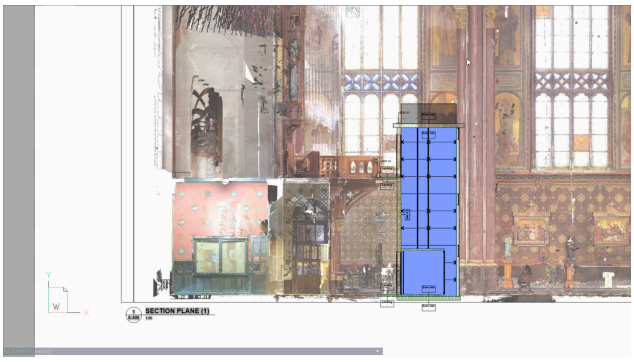
Make a vertical section box on the place you want the vertical projection.

Make sure that the blue round arrow aims to the side you want your projection.



20.7.13.7 Generating BIM sections with point clouds

In some cases it is not necessary to recreate the existing building. Background images can give so much more context to the design documents. These can be used to verify the created 2D geometry, in high quality scans these images can also be used as graphical material. For example, as a background image for a BIM model in renovation projects where modern interventions are made in historical buildings.



Point Cloud scan by Realistic Visual styles.

20.7.14 Interior elevations

20.7.14.1 Command

BIMSECTION

20.7.14.2 About

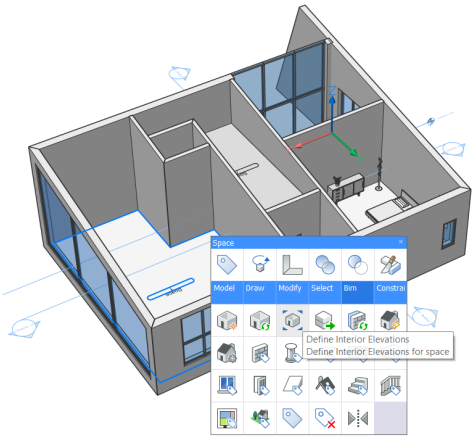
Interior elevations show wall elevations and floor plans for individual Spaces. They provide the details and dimensions of building elements. This can be particularly useful for interior designers or to make a layout of power sockets and light switches. In BricsCAD they can be automatically defined by a BIM Space.

For more information about this command, please read BIMSECTION.

20.7.14.3 Create interior elevations from a BIM space

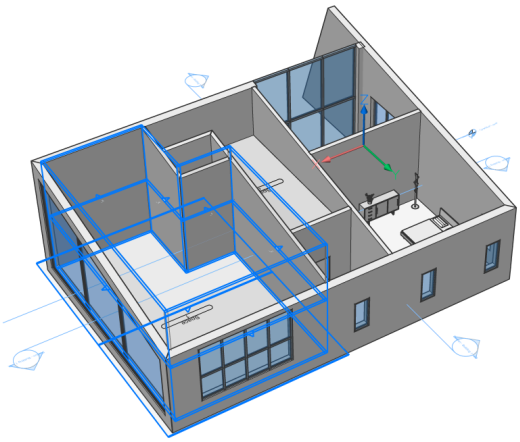
1 Hover the cursor over a Space Tag and select **Define Interior Elevations** from the **BIM** Tab in the Quad **Or** type BIMSECTION in the Command line, followed by i and Enter to choose the interior option.

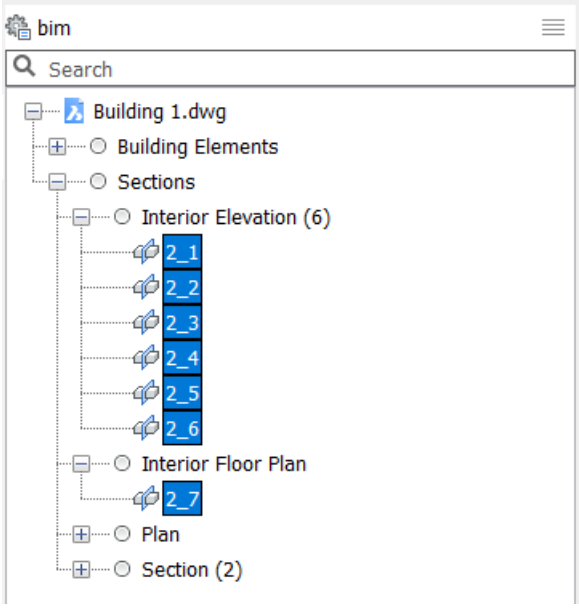
You are prompted: Select BIM Spaces to generate Interior Elevations or [Attach section]:



2 (Skip if you used the Quad) Select the BIM Space for which you want to create the interior elevations and hit Enter.

3 The interior elevations for each wall and the interior floor plan are now defined. They can be found in the **Structure Browser**.

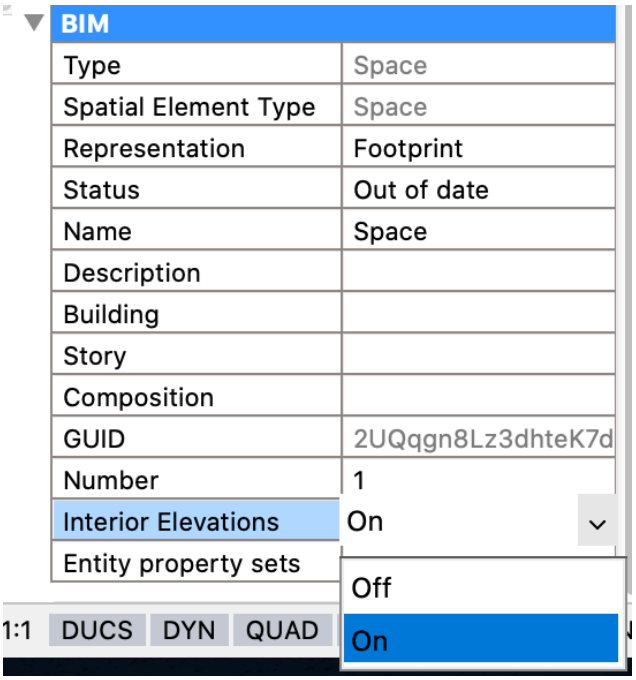




Six interior elevations and one interior floor plan have been created and will appear in the structure browser.

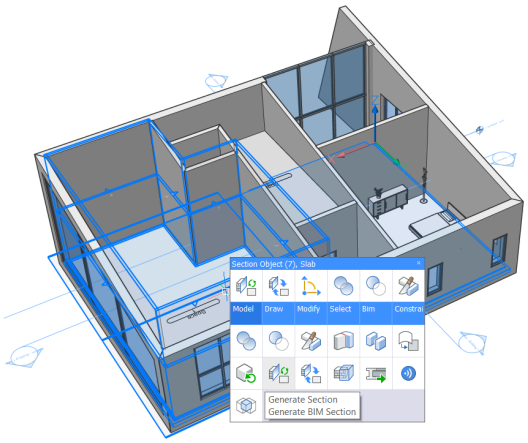
Nota: If there already are interior sections for a Space, running BIMSECTION again on that Space will have no effect. To regenerate the interior elevations, you must delete the existing interior elevations in that Space and run BIMSECTION again.

4 To show or hide the interior elevations you can use the HIDE/ISOLATE/UNISOLATE objects commands **or** you can go to the properties of the Space and switch the interior elevations on and off in the **BIM** tab.



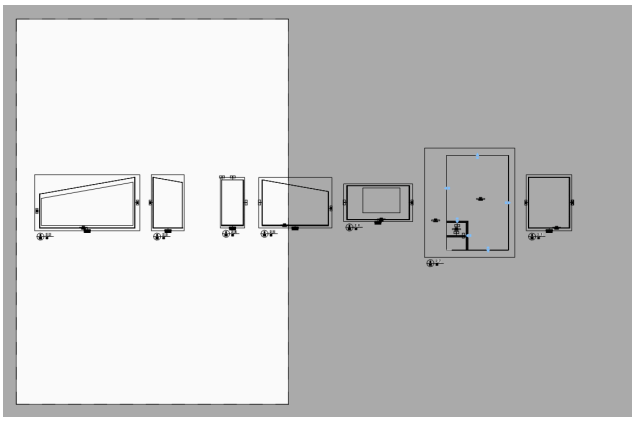
5 Select all elevations, as illustrated above and select **Generate Section** from the Quad to generate the

interior elevations on a sheet.



6 To open the section results click **Open Model**whilst the section planes are selected.

The sheet is automatically saved in the same folder as the model drawing. The following elevations have been produced and can be edited for the selected Space in the model above:



Nota: The section drawings are stored in the file **InteriorElevations** by default. If you generate sections of a second Space, the sections will be added to the same file as the previous ones.

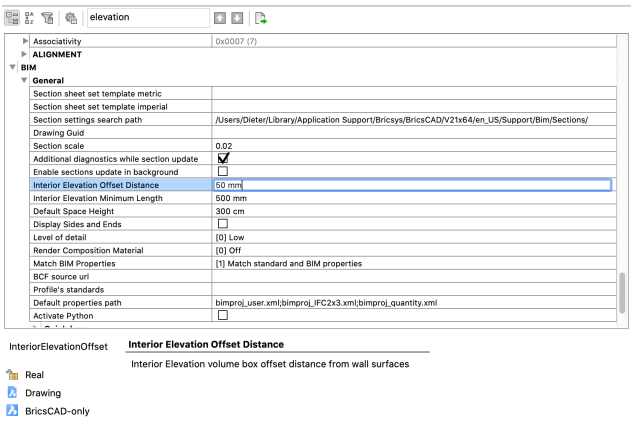
If you need more information on how to generate drawings, visit the Generating Drawings page.
If you want to annotate your drawings, see **Annotate** > **Creating linear dimensions**.

20.7.14.4 Edit the size of the interior elevations

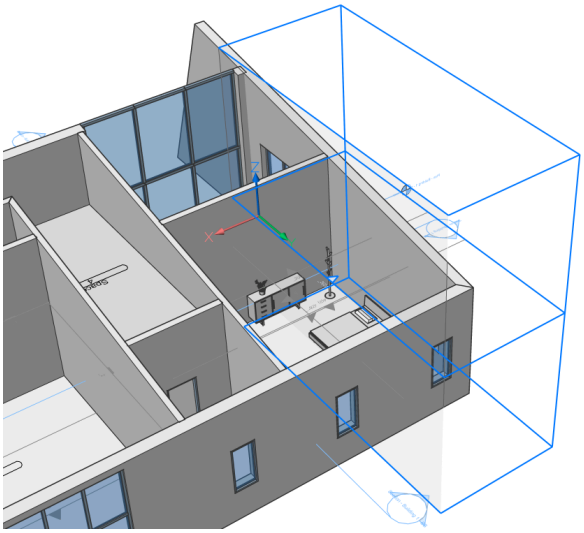
It's possible to edit the size of these sections to include furniture for example. If you create interior elevations for a BIM Space, the offset value is 50mm by default. To include furniture in a section, you have to change this value.

1 To change this setting type INTERIORELEVATIONOFFSET in the Command line and hit Enter. Type a new value and press Enter **or** click on two points in your drawing and the distance between them will be selected.

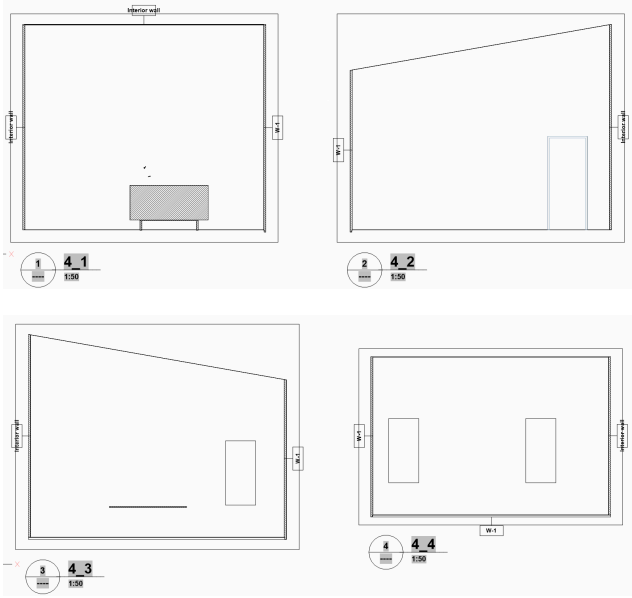
You can also change the offset by typing Settings in the Command line, search for elevation and adapt the **Interior Elevation Offset Distance** value.



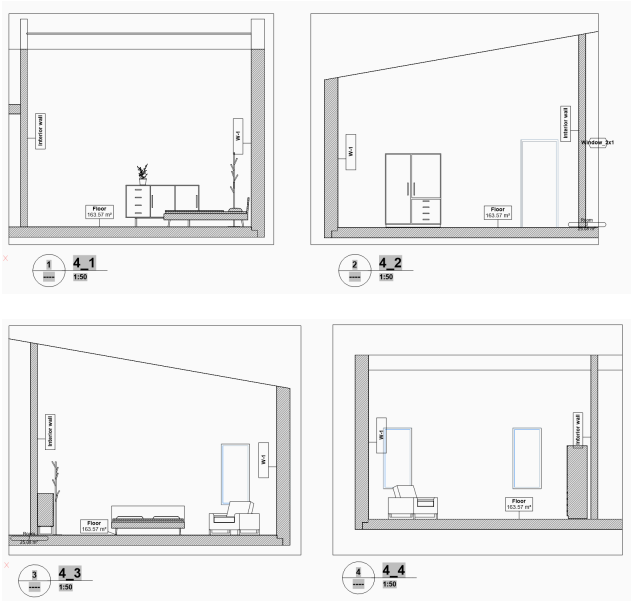
2 If you make new Interior Elevations, the offset distance will be the chosen value. This is also visible when you select the interior elevations in your drawing:



3 This results in more detailed section sheets with the furniture included.



Without specified offset distance (50 mm)



With specified offset distance (2100 mm)

20.7.15 Schedules

20.7.15.1 Open the BIM project browser panel

1 Move the cursor over a toolbar, ribbon item or a dockable panel.

2 Right-click.

3 Choose **Panels** > **BIM Project Browser** in the context menu.

See the article Project browser, for more information about the BIM **Project Browser**.

20.7.15.2 How to create schedules

Schedules are exports, entity properties, block attributes and drawing information to tables on sheets and tables.

Step 1: create a project

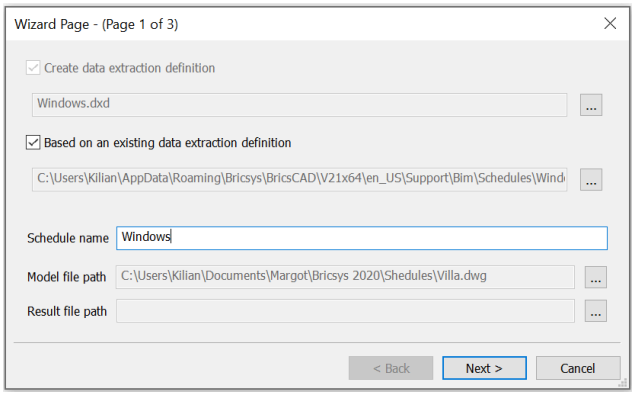
For more information about creating a project, see the article Project browser.

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (Wizard)...** to open the wizard.

Step 2: the schedule wizard

A wizard of 3 pages displays, this will help you to create a schedule.

- Page 1



(1) Create data extraction definition creates a new .dxd (data extraction definition) file:

a Turn the option on.
b Click the **Browse** button (...).
c In the file dialog box, select a folder.
d Enter a name in the **File** name field.
e Click the **Select Folder** button.

Nota: By default, the .dxd file will be placed in the project folder. To change this path click the **Browse** button.

(2) Based on an existing data extraction definition uses an existing .dxd file as the template for the new one:

a Turn the option on.
b Click the **Browse** button (...).

Nota: The .dxd files are stored in ..\bim\schedules serve as templates.

c In the file dialog box, select a .dxd file.
d Click the **Open** button.

Nota: By default .dxd files are saved in the ...\Bim\Schedules subfolder of the **Support** folder. You can use newly created custom .dxd files too. Also, you can create a .dxd files by yourself

using a text editor. Please scroll down for the procedure. "How to create and edit your own Data Extraction file" in this article.

You can either choose a new name for your schedule or keep the default name.

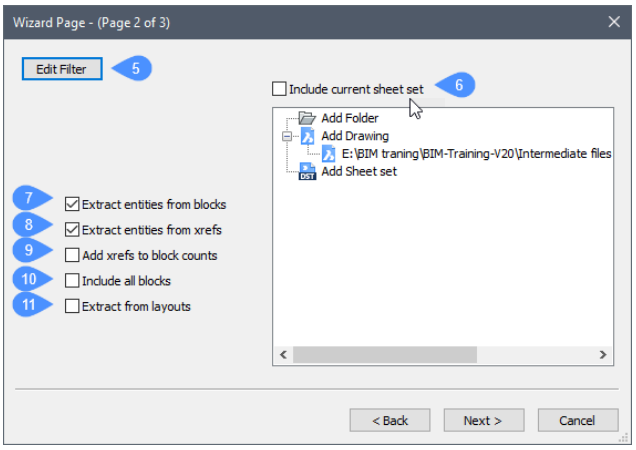
(3) Model file path: The model path links to the model. This is filled in automatically if the wizard is started in the model file. If the wizard is started in another file from the project, link the model file:

a Click the **Browse** button (...).
b In the file dialog box, select the model file.
c Click the **Open** button.

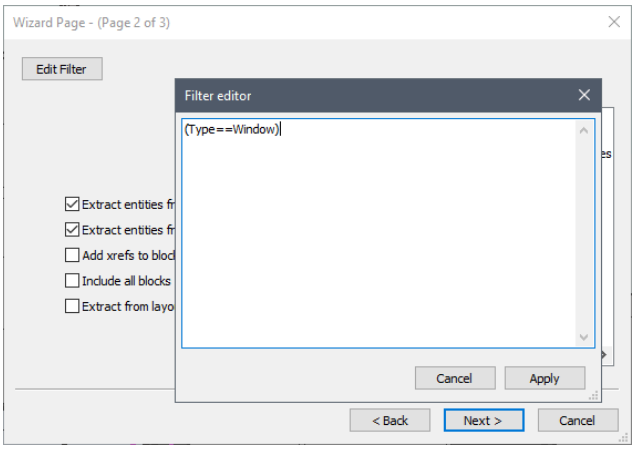
(4) Result file path: The **Result file path** determines where the schedule is created. When left blank, the Schedule is created as a new sheet:

a In the file dialog box, select the sheet file.
b Click the **Open** button.
Click **Next** for the next page.

- Page 2



(5) Edit Filter: The filter selects which BIM entity types are included in the Schedule. Use an existing data extraction definition to automatically define the filter. See the Editing Data Extraction Definition (.dxd) files to learn more about the filter syntax.



(6) Include current sheet set: Adds the current sheet set for data extraction.

(7) Extract entities from blocks: Adds entities in blocks which will also be processed in the schedule.

(8) Extract entities from xrefs: Adds all entities in xrefs which will also be processed in the schedule.

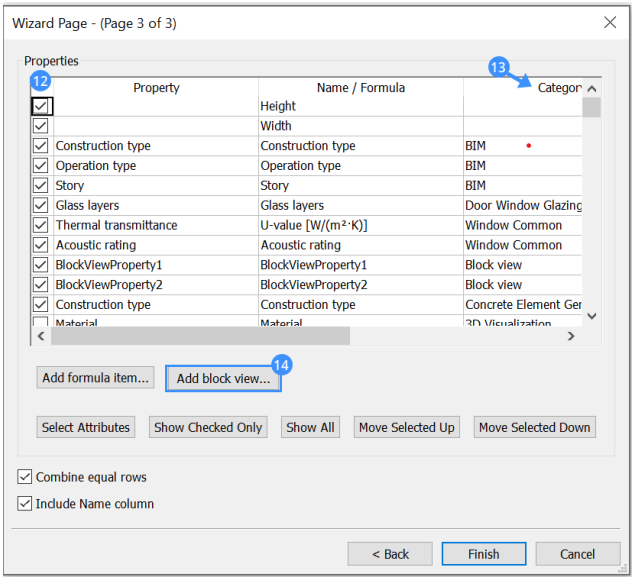
(9) Add xrefs to block counts: Counts selected xrefs as blocks.

(10) Include all blocks: Includes all blocks into the schedule.

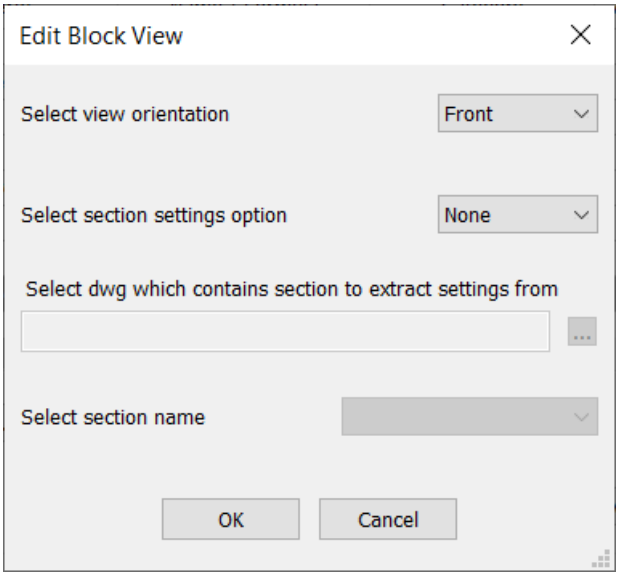
(11) Extract from layouts: When toggled on, all entities layouts will also be processed in the schedule.

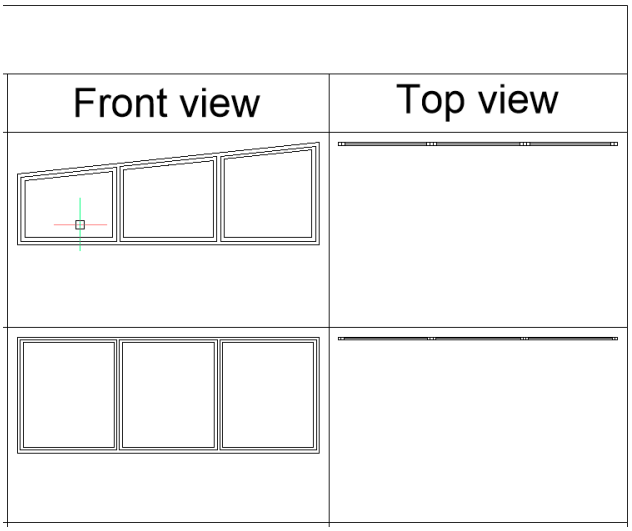
Nota: The Wizard automatically selects all entities in the drawing.

- Page 3



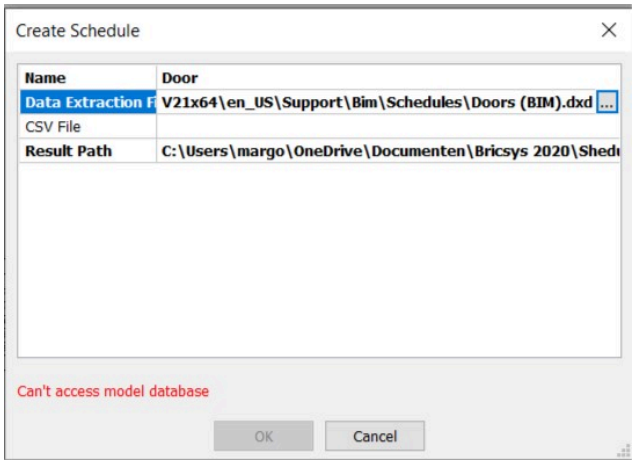
(12) Select the properties to be included in the schedule: click the checkbox to check/uncheck a property: selected (☑) or not selected (☐).
(**13**) Click on the **Category** column to change how the property is shown in the schedule.
(**14**) To include plan and elevation symbols, click on the **Add block view...** button.





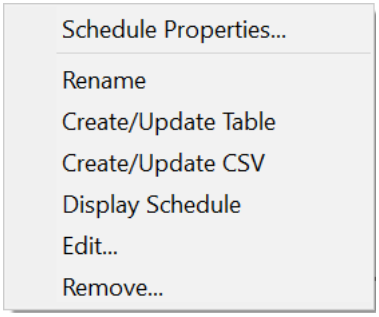
To sort the list, click the title in the column heading. Click again to sort in reverse order.
Click **Finish** to close the wizard (or click **Back** to switch back to page 2).

When the project is created, right-click in the **Schedules** section. Click **Add Schedule (From dxd)...** to open the wizard.



20.7.15.3 Update and display schedules

1 Right-click on the new schedule (created in the BIM **Project Browser**). A context menu appears:



Schedule Properties...	**Name**: Renames the schedule. **Model**: Displays the link to the model file. **DWG Handle**: Displays the DWG Handle. **Data Extraction File**: Displays the link to the Data Extraction Definition. **CSV File**: Displays the link to the .csv file. **Result Path**: Displays the resulting path.
Rename	Renames the schedule.
Create/Update Table	Creates a sheet with the schedule or updates the sheet.
Create/update CSV	Creates a CSV file with the schedule information and saves it in the folder where de model file is located or updates the CSV file if it already exists.
Display Schedule	Displays the schedule in the resulting drawing. (If not already open, the drawing is opened.

Edit...	Opens the **Wizard Page** dialog box where you can edit your schedule. More information about this setup can be found in the procedure in this article.
Remove...	Removes the schedule.

2 Click on **Update**.

3 Once the schedule is updated, click on **Display Schedule**. The schedule is shown in the desired sheet. **Or** double-click on the schedule's name to display the schedule.

20.7.15.4 Procedure: create a window schedule

In this example, you will learn how to make a window schedule from scratch. You can create a door schedule in the same way.

Step 1: open schedule wizard

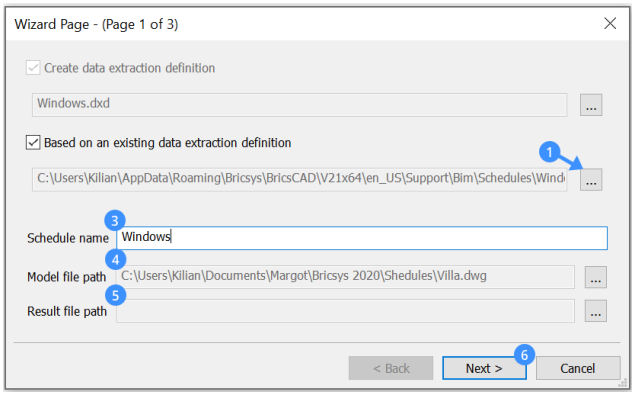
1 Open the BIM **Project Browser**.

2 Right-click on **Schedules**. A context menu appears.

3 Click on **Add Schedule (wizard)...** to open the wizard.

Step 2: the schedule wizard

- Page 1



For this example, use an existing data extraction definition.

a Click on the **Browse** (1) button. A folder pops up. This contains the predetermined data extraction definitions.

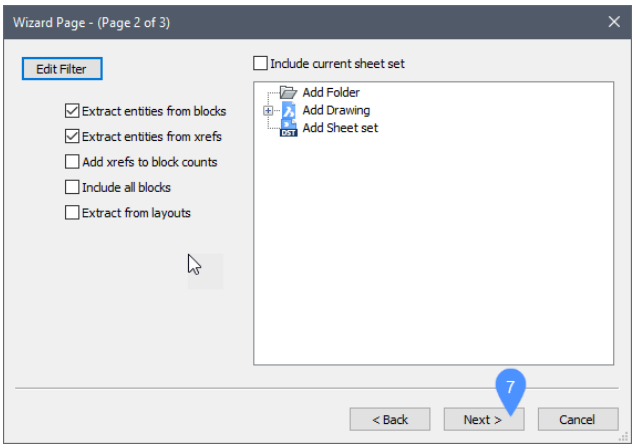
b In the folder, select *'Windows.dxd'*.

c Change the **Schedule name** (3) to, in this case, **Windows**.

d The **Model file path** (4) is correct because it was taken from the model file. To create the schedule on a separate sheet keep the **Result file path** (5) empty.

e Click **Next** (6).

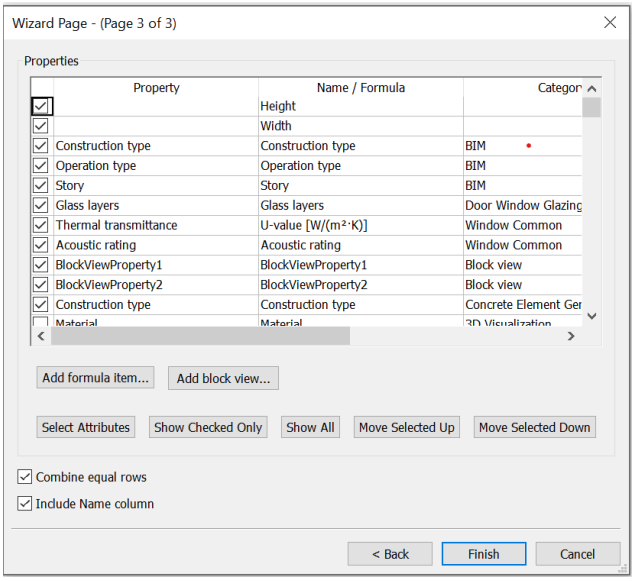
- Page 2



a The wizard automatically selects all the entities in the model.

b Click **Next** (7).

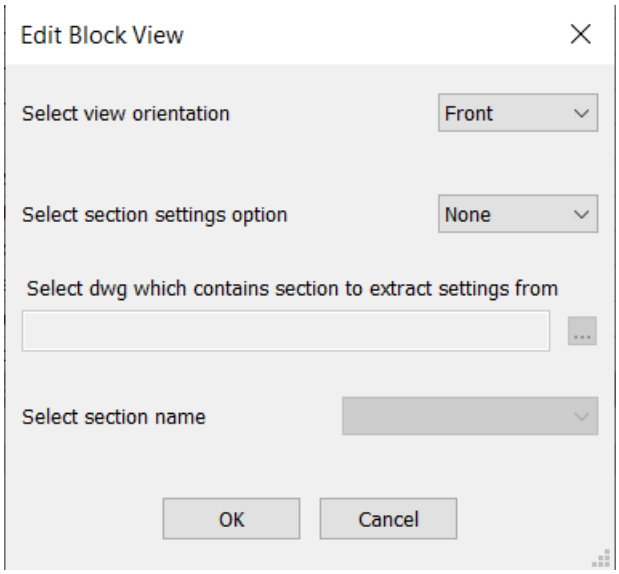
- Page 3



a Because you use an existing .dxd, the properties are already pre-selected. Views such as **Front** and **Top** are also pre-set.

b Click on the **Block view** category, the **Edit Block View** window appears.

c Choose the view.



d Click **Finish**.

Step 3: open the schedule

1 Right-click on the new schedule.

2 In the context menu select **Create/Update Table**.

- Double click on the schedule.
- Right-click on the schedule and select **Display Schedule**.
- Go to **Sheets** and double-click on the schedule sheet. Open the schedule.

3 The result is as follows:

Windows											
Count	Name	Overall height	Overall width	Construction type	Operation type	Story	Glass layers	U-value [W/(m² K)]	Acoustic rating	BlockViewProperty1	BlockViewProperty2
1	Double Panel Vertical Window	2000.000000	3000.000000	Wood	Double panel vertical	Floor 0	3	0.600	32 dB		
1	Double Panel Vertical Window	2000.000000	3000.000000	Wood	Double panel vertical	Floor 0	3	0.600	32 dB		
1	Single Panel Window	1000.000000	1600.000000	Wood	Single panel	Floor 1	3	0.600	32 dB		
1	Single Panel Window	2000.000000	5700.000000	Wood	Single panel	Floor 1	3	0.600	32 dB		
2	Single Panel Window	2000.000000	800.000000	Wood	Single panel	Floor 0	3	0.600	32 dB		
1	Triple Panel Vertical Window	1000.000000	12400.000000	Wood	Triple panel vertical	Floor 1	3	0.600	32 dB		
1	Triple Panel Vertical Window	2000.000000	6500.000000	Wood	Triple panel vertical	Floor 0	3	0.600	32 dB		
1	Triple Panel Vertical Window	2000.000000	7600.000000	Wood	Triple panel vertical	Floor 0	3	0.600	32 dB		
1	Triple Panel Vertical Window	2000.000000	6600.000000	Wood	Triple panel vertical	Floor 0	3	0.600	32 dB		

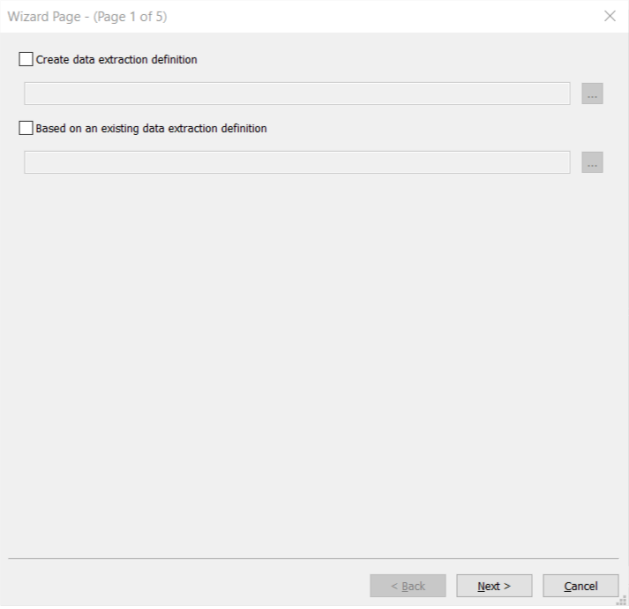
20.7.15.5 Procedure: how to create and edit your own data extraction file: furnishing elements

You can also create a Data Extraction File by yourself and use it to make new schedules. The DATAEXTRACTION command will, just like the Schedules wizard, make a new .dxd file with your settings, but it will also make a .csv file with your data in it.

1 Open your model file.

2 In the **Structure Browser**, select all Furnishing Elements.

3 Type in DATAEXTRACTION in the Command line. A wizard of 5 pages appears:

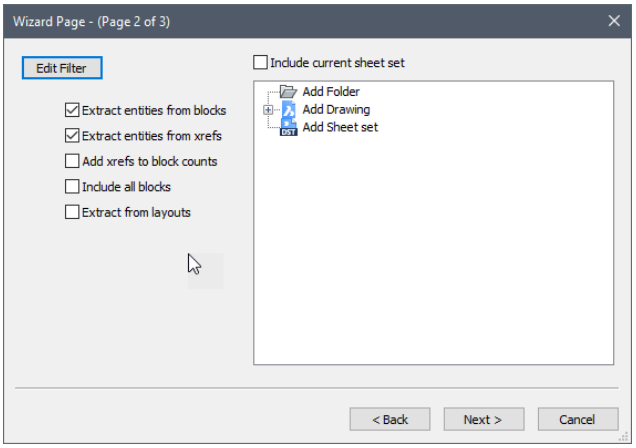


4 Check the **Create data extraction definition** box and choose a file. Name it “FurnishingElements.dxd” and save it in your project folder.

Nota: You can base you dxd file on an existing Data Extraction Definition file by checking **Based on an existing data extraction definition**. And choose the file you want to base on.

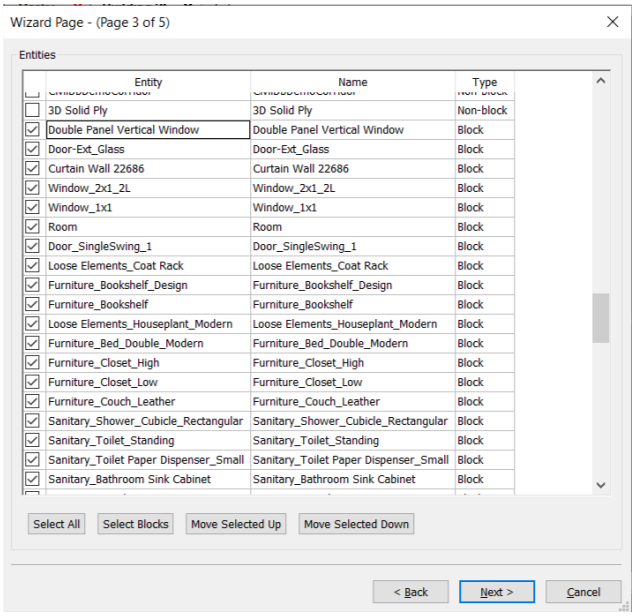
5 Click **Next**.

6 Select all entities you want to include in your schedule.



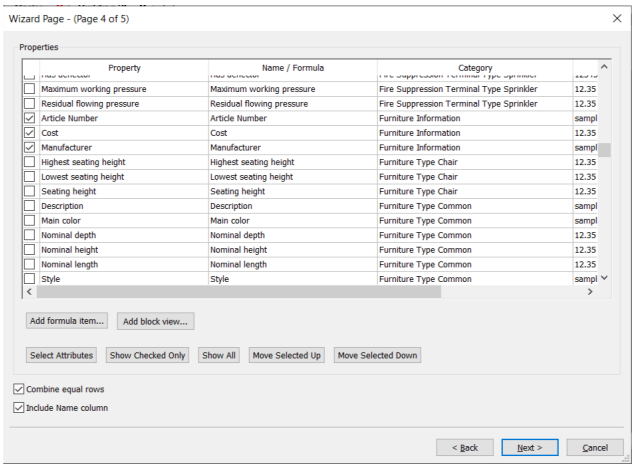
7 Click **Next**.

8 Select all entities you want to have a data extraction of.

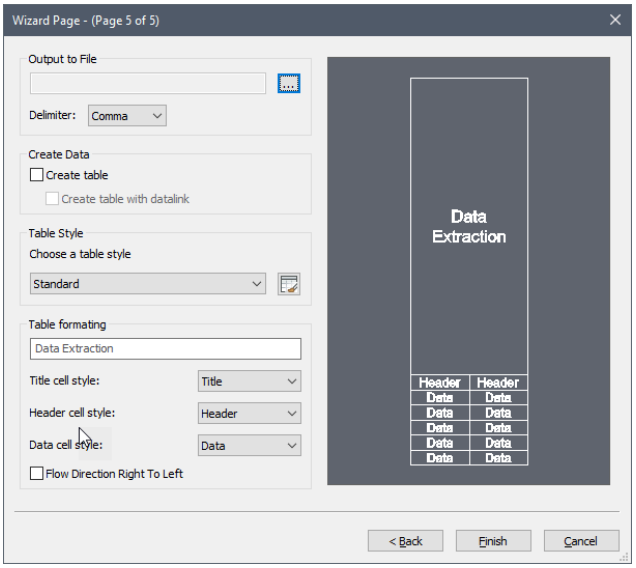


9 Click **Next**.

10 Check the all properties you want in the **Furniture Information**.



11 Click **Next**. The next page appears:



12 Click on the three dots next to the **Output to File**. Give in a new name for your .csv file and click Save. You can choose between different options for the delimiter, but it's best to use a semicolon.

13 You can create a Data table as well. When checking this, you will have the option to place the table any where inside your drawing either model space or paper space.

14 Click **Finish**.

15 This should have created 2 files: a .csv file containing the results of the data extraction (basically the same as a schedule, but in an external file) and a .dxd file. This .dxd file can now be re-used for creating a schedule. (See procedures above).

16 Open the .dxd file in a text editor. We can see 5 sections: **Settings**, **Entity Types**, **Properties**, **TableFormatOptions** and **SELECTION SET**.

17 Leave the **Settings** as they are.

18 The **Entity Types** are chosen because of the selection set you had active when using data extraction.

However, instead of explicitly choosing which types (and handles, see later) you want to extract data from, you can replace the explicit entity types by an asterisk (*), to define that we want to extract information from all entity types.

19 Leave the **Properties** as they are.

20 Leave the **TableFormatOptions** as they are.

21 You can delete the entire **SELECTION SET** section, if you don't want to explicitly define which entities you'll be extracting data from.

22 You can add a filter by adding a new section: The section title is called [Filter] The section content is (Type == Furnishing Element) (in the .dxd files that are given as example in the support folder, a Filter on Door and Window type is used).

23 Thus, the contents of the .dxd file should be as follows:

24 *[Settings]*

ExtractFromBlocks=0

ExtractFromXrefs=1

CountXrefs=0

CombineEqualRows=1

IncludeNameColumn=1

FooterStyle=0

FlowDirectionRtoL=0

CVSdelimiter=1

IncludeLayouts=0

[Entity Types]

*

[Properties]

Furniture Information:Article Number|Article Number|

Furniture Information:Cost|Cost|%1u2%pr2

Furniture Information:Manufacturer|Manufacturer

[TableFormatOptions]

TableStyle=Standard

TitleCellStyle-TITLE

HeaderCellStyle-HEADER

DataCellStyle-DATA

[Filter]

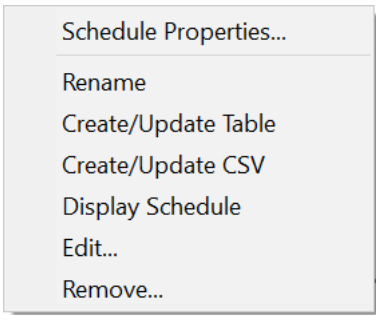
(Type==Furnishing Element)

25 Save this .dxd file in your project folder.

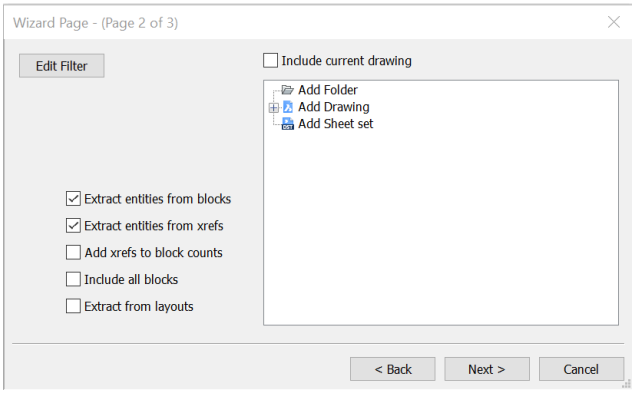
20.7.15.6 Procedure: how to edit a schedule

1 Go to the **Schedules** tab in the **Project Browser**.

2 Right-click the selected schedule. A context menu appears:



3 Click **Edit...** A Wizard of 2 pages appears:



4 Check or uncheck the options you want.

5 Click **Next**.

6 Check all properties you want to be in the schedule or uncheck all properties you want to leave out.

7 Click **Finish**.

8 Right-click on the schedule again and choose **Create/Update Table**.

9 Repeat this action but choose **Create/Update CSV**.

10 The table and CSV file are now updated. You can update and open your sheet with the schedule again and see the changes.

20.7.16 Spot elevation level

20.7.16.1 Command

BIMTAG

20.7.16.2 About

The Spot Elevation tag displays the absolute elevation of a selected point of a BIM object displayed in a section view, based on the **World Coordinate System (WCS)**.

As the Spot Elevation tag is an associative tag type, it can only be used in the **paper space** of a drawing sheet.

For more information on how to set up Sheets and create Projects, please visit the **Project Browser** article.

20.7.16.3 Procedure: how to place a spot elevation tag on a section sheet

1 Open a section sheet of the model. Right-click on the section in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line **or** click on **tag** in the **Draw** menu of the Quad.

3 **You are prompted**: [Manual/selection options (?)]

Type **M**t o choose **Manual** and press Enter.

4 **You are prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

5 **You are prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

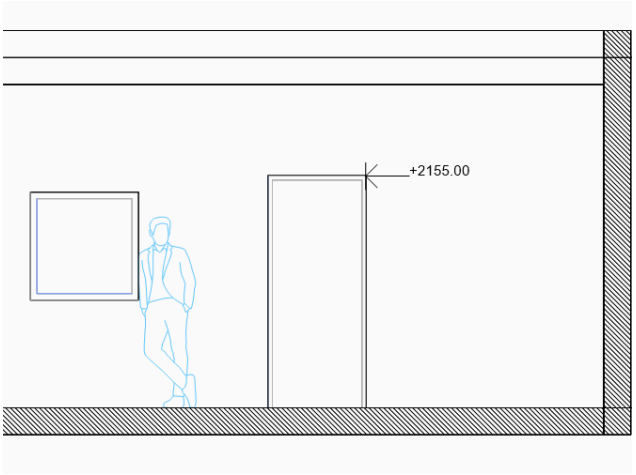
Type **S**to choose **Spot-elevation** and press Enter.

6 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the elevation level measured relative to level 0 of the model.

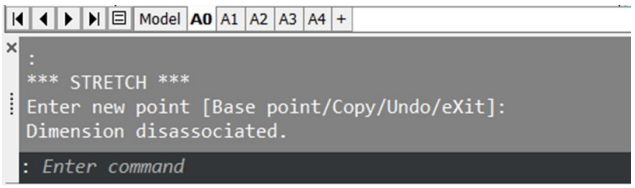
Nota: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

7 When there are several tags possible on the location where you place the tag, hit CTRL to scroll through the different options while the Hotkey Assistant (HKA) appears at the bottom of the screen.

8 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

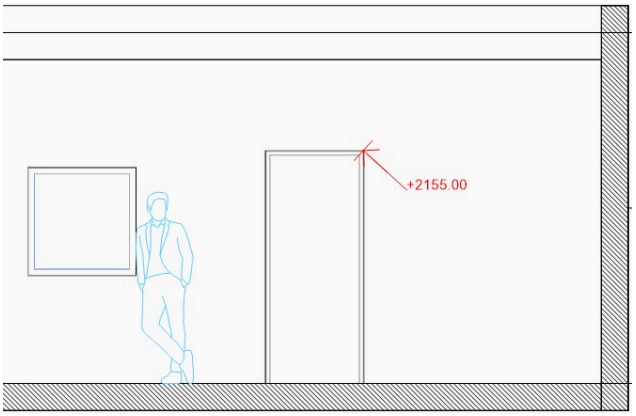


Nota: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value becomes dissociated in the Command line.

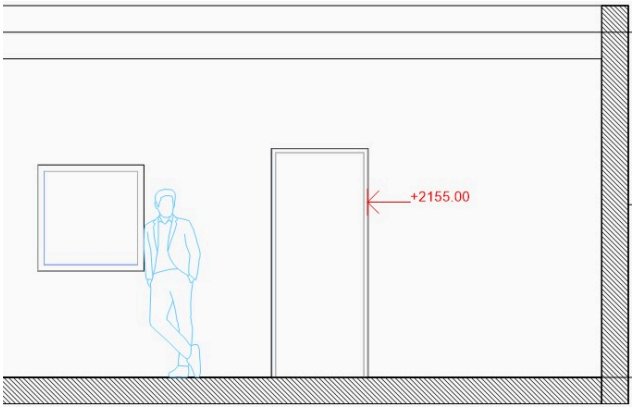


9 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

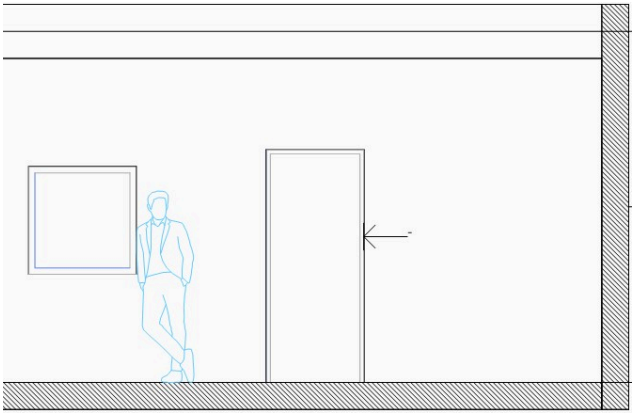
Nota:If the tag has been displaced, it may behave irrationally or haphazardly by either turning red or omitting its value, as illustrated below:



Tag turned red after displacing the arrowhead of the leader and updating the section.



Tag turned red after displacing the entire tag and updating the section.



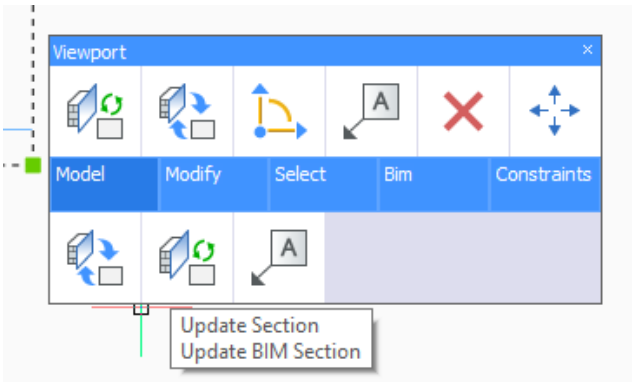
Tag value disappeared after displacing the tag and updating the section.

20.7.16.4 Procedure: how to place a spot elevation tag on an elevation sheet

1 Open an elevation sheet of the model. Right-click on the elevation sheet in the **Project Browser** and select **Display View**.

2 Type BIMTAG in the Command line,

3 **or** click on **Tag** in the **Model** menu of the Quad.



4 **You will be prompted**: [Manual/selection options (?)]

Type **M** to choose **Manual** and press Enter.

5 **Later, you will be prompted**: [Auto/Current mleader style/change Tag type]

Type **T** to change the **Tag** type and press Enter.

6 **Finally, you will be prompted**: [Tag/cOmposition/Load-bearing directions/Spot-elevation]

Type **S** to choose **Spot-elevation** and press Enter.

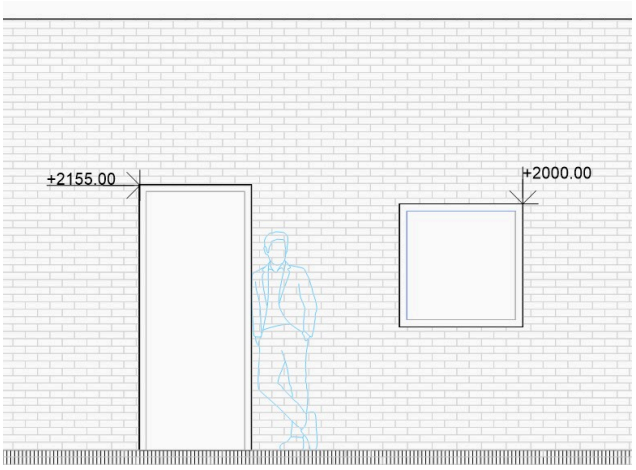
7 Click on a point on the edge of a BIM object to display the elevation height. The displayed value is the absolute elevation level measured relative to Floor 0 of the model.

Nota: You can select any point of the edge of a BIM object. Turn off ESNAP if you don't only want to select snapping points.

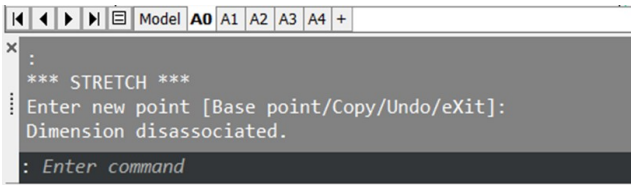
8 When there are several tags possible on the location where you place the tag, hit CTRL to scroll

through the different options while the HotKey Assistant (HKA) appears at the bottom of the screen.

9 Move your cursor in order to adjust the position of the tag and left-click to place the tag.

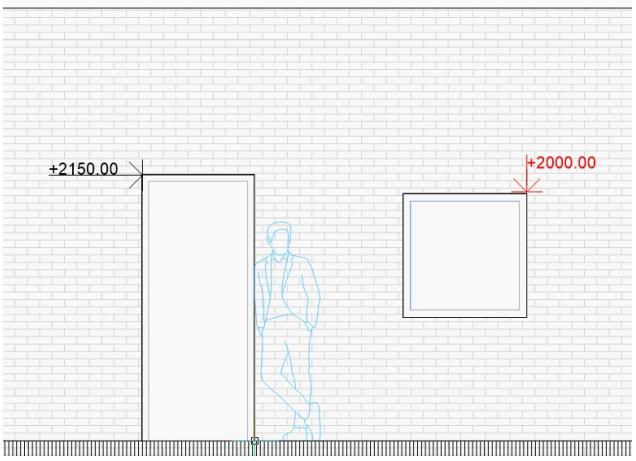


Nota: The spot elevation level is a non-dynamic value. When you move the tag, the value does not always change along. The moment you move a tag that will not be updated, you can see the tag value is dissociated in the Command line.

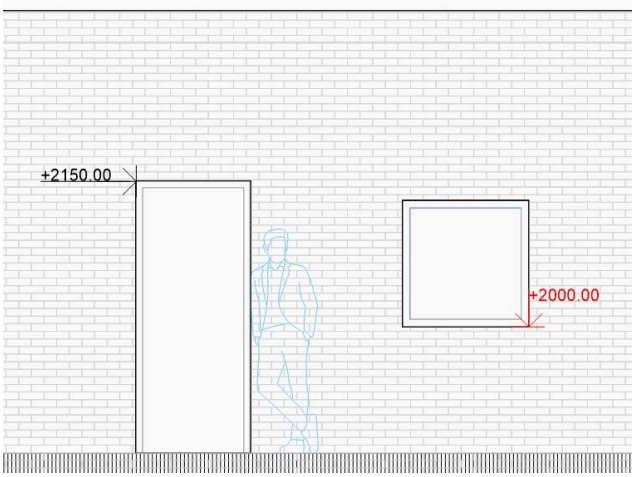


10 Update the section to make sure the right values are displayed. Select the section, open the Quad and click on **Update Section** in the **Model** tab.

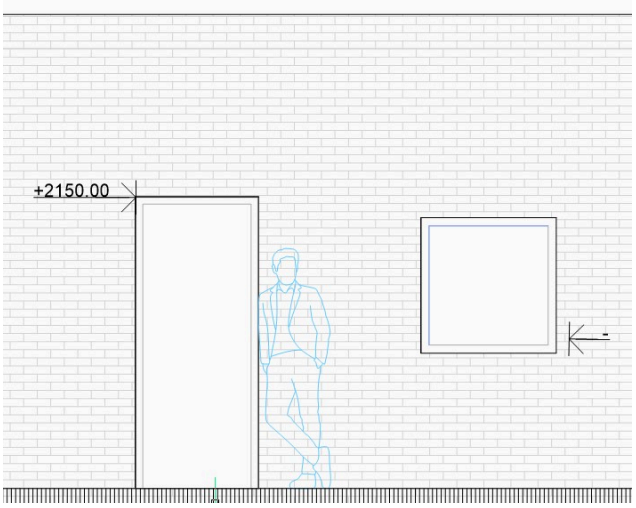
The tag should not behave irrationally or haphazardly by either turning red or omitting its value when it's displaced.



Tag turned red after displacing the arrowhead of the leader and updating the section.



Tag turned red after displacing the entire tag and updating the section.



Tag value disappeared after displacing the tag and updating the section.

20.7.16.5 Procedure: placing spot elevation level tags on floor plans or reflected ceiling plans

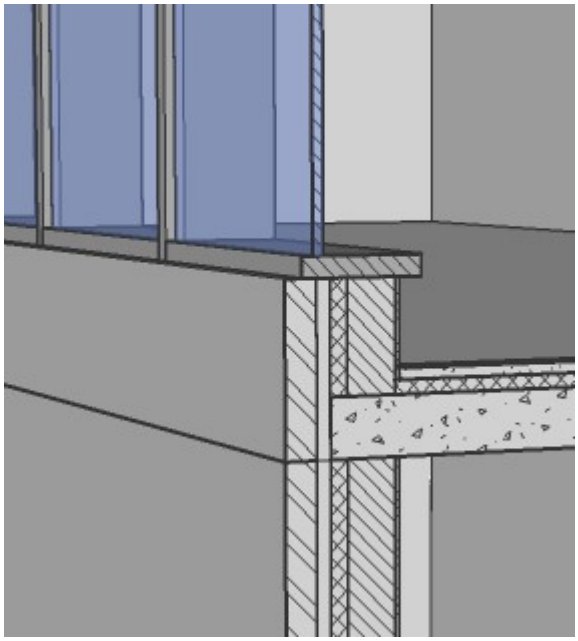
1 Open a floor plan or a reflected ceiling plan from the **Project Browser**.

2 Go through step 2-7 of 'How to place spot elevation level tag in a section sheet'.

3 In a floor or ceiling plan, there are 2 different tags possible.

- **With MLeader**

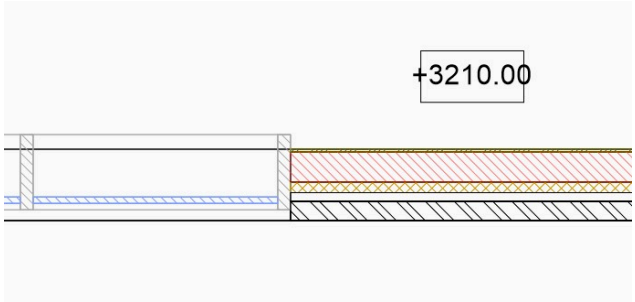
  This annotation tag type, which illustrates with a leader line, is also normally used in the section or elevation sheets. You can use this tag on entities that do not reach or aligned to the floor/ceiling level, such as a window sill in the picture below.

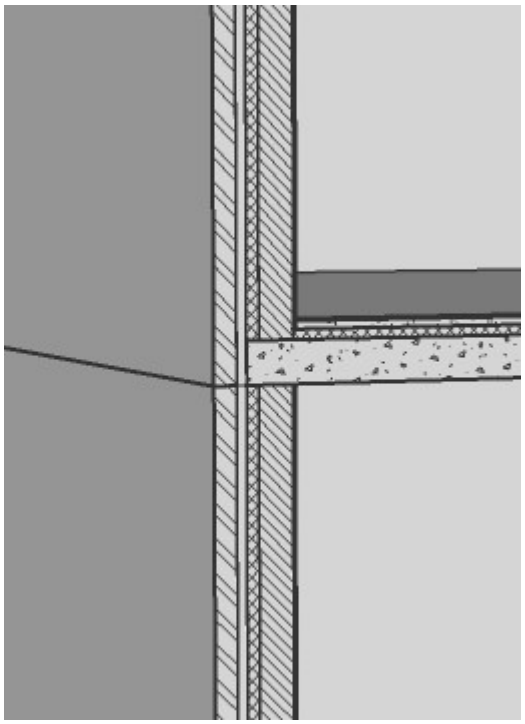


Spot elevation level tag on window sill level which is not on floor level.

- **Without MLeader**

  This tag has a different layout to the tags found in the elevation or section sheets. This tag appears when you place a spot elevation tag on entities that reaches the floor/ceiling plane, thereby displaying the elevation level of the floor plane.



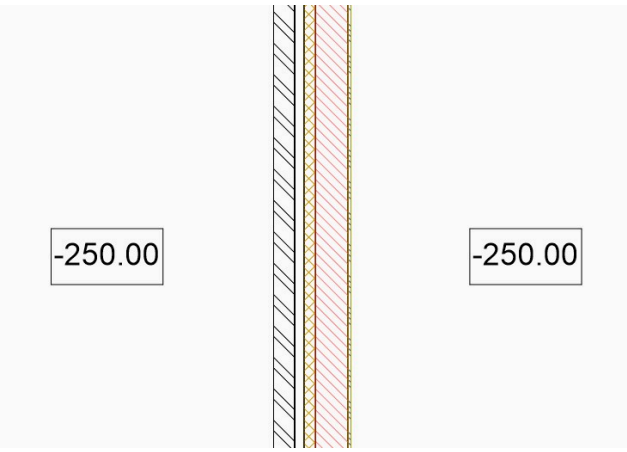


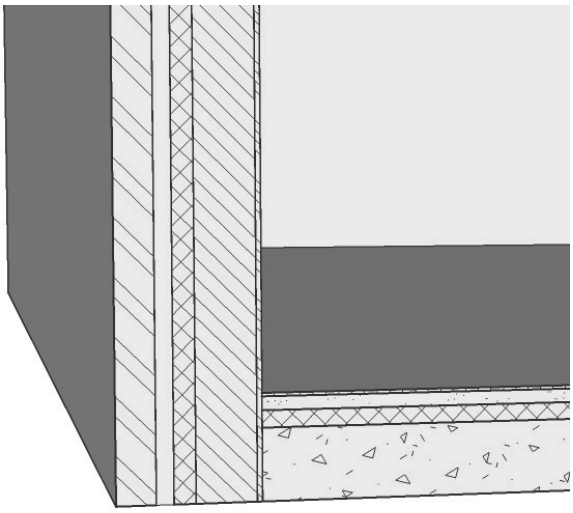
Spot elevation level tag on wall that reaches till the floor slab.

4 When placing a tag on a wall in plan view, you may encounter different results for different plies of the wall. This is due to how elevation levels are measured In BricsCAD. When placing a spot elevation tag on an entity in plan view, it refers to the elevation value of the lower level of the entity's ply, if applicable. Similarly, when you place a tag on a ceiling plan, the upper level of the ply is measured. Since plies of a wall may differ in their lengths based on how they are designed at detail junctions, the spot elevation tag value may differ according to the ply you have selected.

- **Construction detail with unedited plies**

  The interior and exterior plies have the same length. All the spot elevation tags you place on this wall in plan view will therefore display the same value.

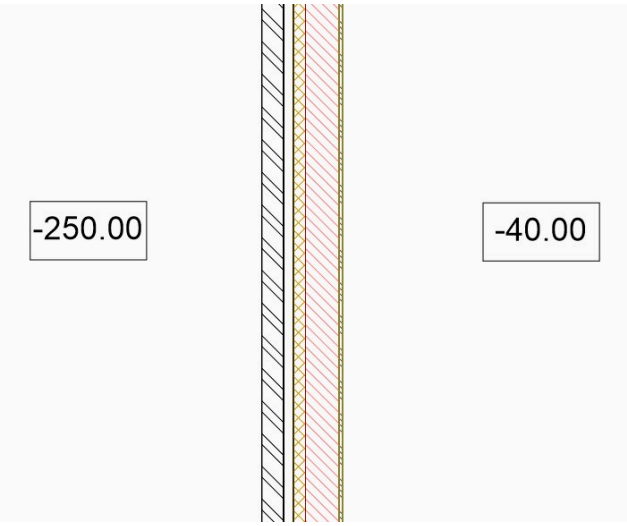


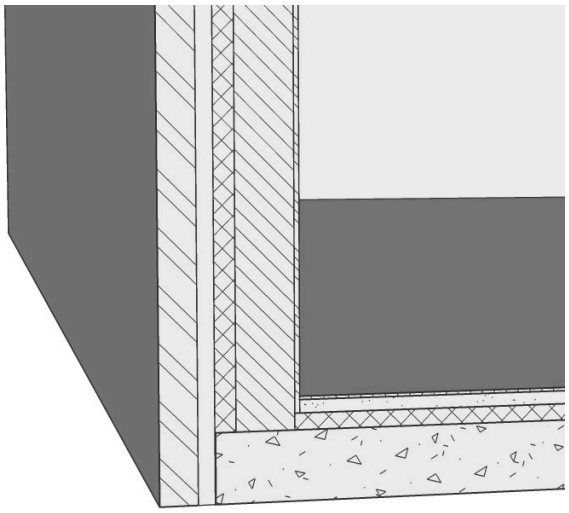


Nota: The value is -250 instead of 0 because Floor 0 is defined as the level of the upper side of the ground floor in this model.

- **Edited construction detail**

  If you have adjusted how the wall would connect to a floor slab based how it would be constructed, the plies of the wall will therefore have different lengths. Depending on which ply you select to place the spot elevation level tag, a different value will be displayed.

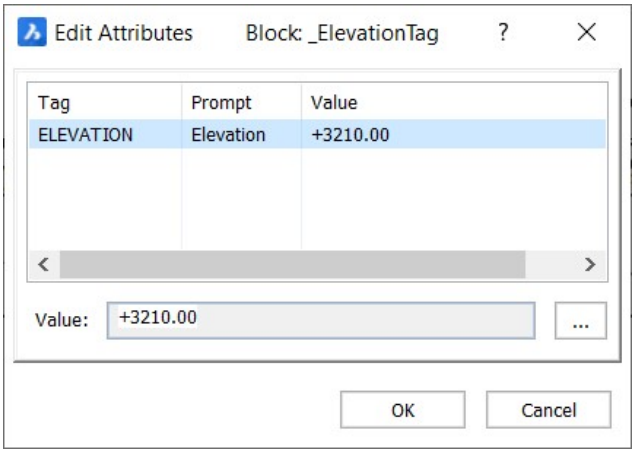




Elevation level of interior ply is higher because ply does not reach till the base level of the floor.

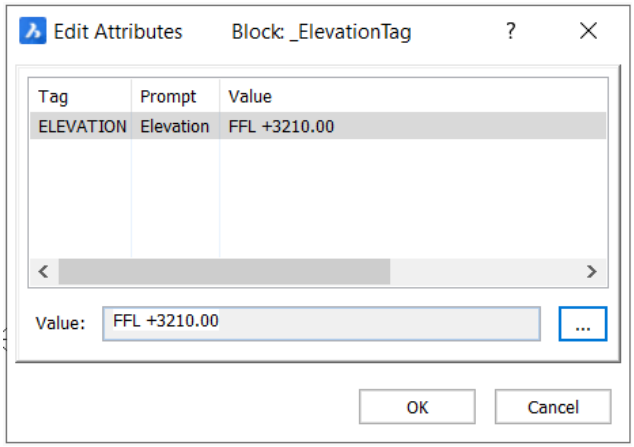
20.7.16.6 Procedure: adjusting the layout of the tag

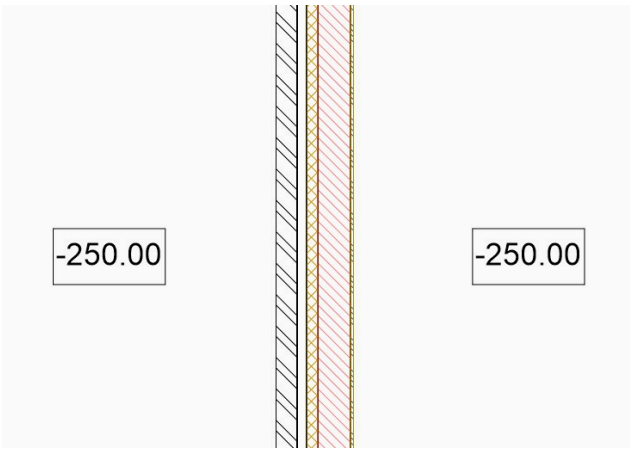
1 Double-click on the value of the spot elevation tag to adjust the layout. A dialog box will pop-up where you can edit the attributes.



2 Click on the three dots to open the **Text Formatting** dialog box.

3 Here you can edit the attributes of the displayed value of the spot elevation level tag. You can put prefixes and/or suffixes to the values, e.g. FFL, SSL, to better illustrate the levels you are referring to.





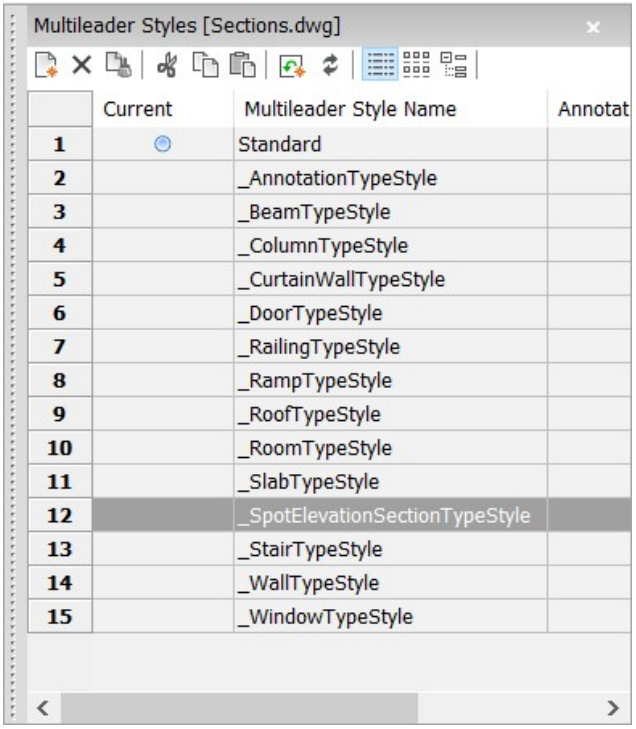
4 If the prefixes and/or suffixes are commonly used, you can create or edit the tag blocks so you do not have to repeat Step 3. For more info, visit the command article BEDIT.

5 For a detailed explanation about attributes, visit the command article MText.

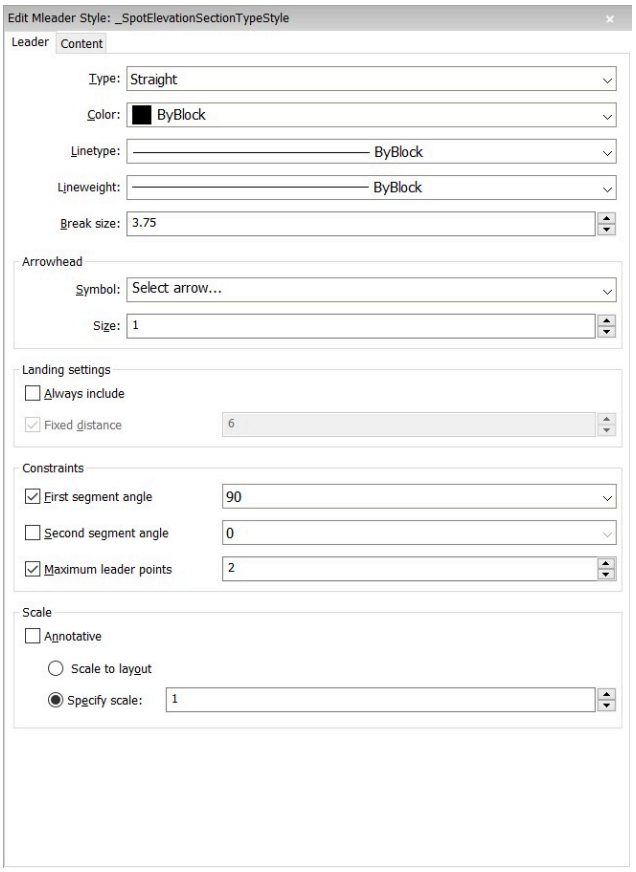
20.7.16.7 Procedure: adjusting the layout of the leader

1 Type MLEADERSTYLE in the Command line and press Enter. The **Drawing Explorer** dialog box will pop-up where you can adjust the layout of the leader.

2 Click on **SpotElevationSectionTypeStyle** in the **Multileader Styles** dialog box.

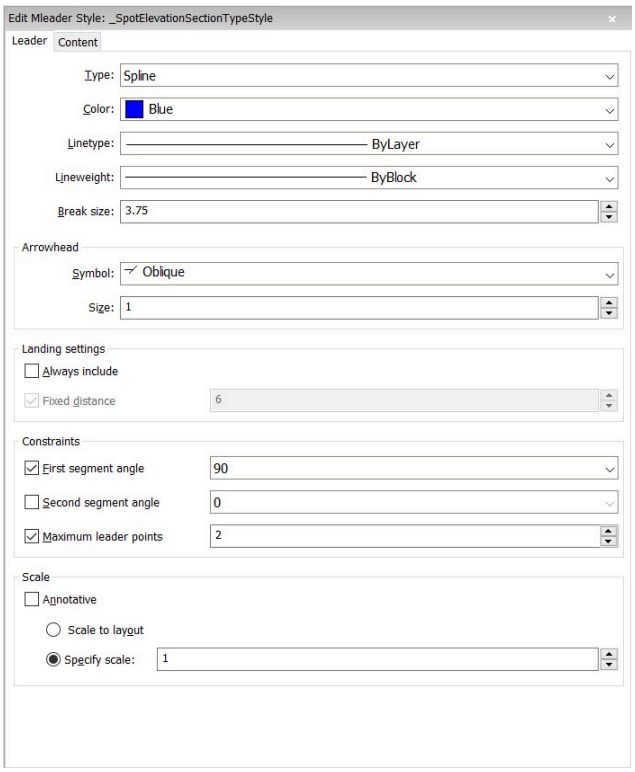


3 You can edit the settings of the leader in the **Edit MLeader Style** dialog box. For more info about the MLeader style settings, visit the article MLeaderStyle.

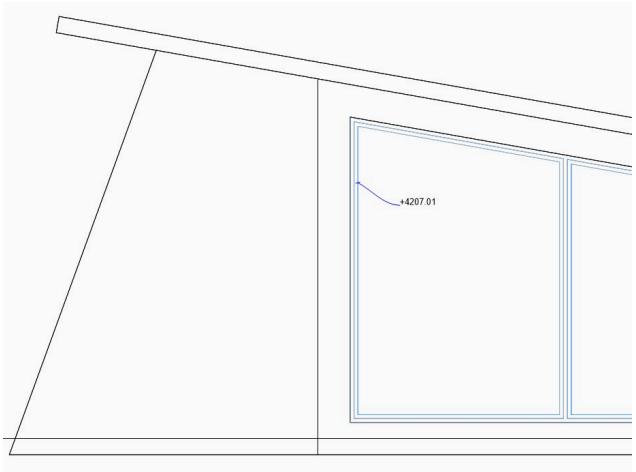


4 After adjusting the settings, click on regen, close the **Drawing Explorer** and update the section to load the new settings.

Nota: If the new settings don't load, type BIMTAG in the Command line, select the viewport, click **R** to select **Regenerate All** and press Enter to load the new settings.



New Settings



Resulting BimTag

20.7.17 Templates for generated drawings

20.7.17.1 About

BricsCAD BIM has a series of templates to customize your generated drawings. These templates are stored in your **Support** folder in the following path:

C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\Bim\Sections.

The **Appdata** folder is a hidden folder by default, you may have to unhide folders first. This path can also be accessed by the SUPPORTFOLDER command.

To understand how it works, we must know how BricsCAD creates layers, hatches, tags, and call-out blocks. Then, for each template, we will discuss how to customize them.

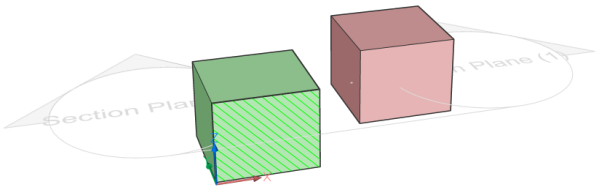
20.7.17.2 Procedure: understanding the created layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

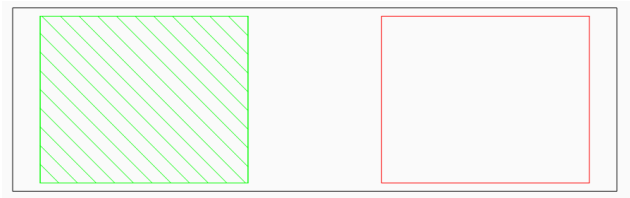
2 Draw two boxes of 500x500x400, one being a bit behind the other.

3 Put the first box on the layer **Layer1** (with layer color green) and the second box on the layer **Layer2** (with layer color red).

4 Create a BIM section, aligned with the front face of the first, green box and cutting through it in the middle, but not cutting through the second, red box.



5 Generate and open the section drawing.



6 Take a look at the **Layers** panel: three groups of layers have been created: **Background**, **Boundary**, and **Fill**.

	Current	Name	D..	On/Off	Freeze	Locked	Color
1	◉	0					White
2		Background_Layer1					Green
3		Background_Layer2					Red
4		BIM_SECTIONS_PREVIOUS					Magenta
5		BIM_VIEWPORTS					White
6		Boundary_Layer1					Green
7		Defpoints					White
8		Fill_Layer1					Green
9		Title Block Frame					175
10		Title Block Labels					157
11		Title Block Text					251

7 The generated layers for the green solid are:
- **Boundary_Layer1**: the boundary of the section through the box.
- **Fill_Layer1**: the hatching of the section.
- **Background_Layer1**: shows subentities (e.g., vertices of solids) that are not sectioned.

8 The layers all inherit the color of the **Layer1** layer.

9 Since the red solid is not sectioned, only the **Background_Layer2** layer is created. It inherits the color of the **Layer2** layer.

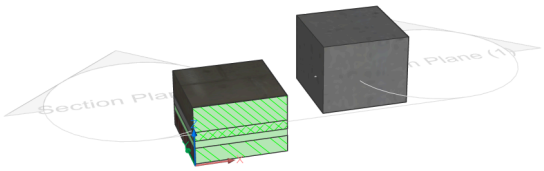
10 Drag the **Cavity Wall, Brick** composition from the **Compositions** panel onto the green box.

Nota: Set the LEVELOFDETAIL system variable to 1 or 2, to display the ply materials.

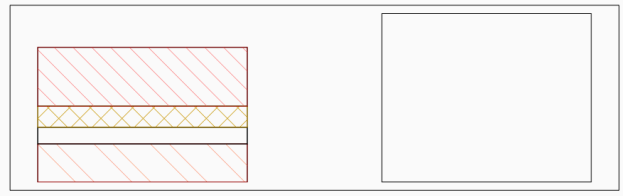
11 Double-click the **Cavity Wall, Brick** composition in the **Compositions** panel.

This composition consists of 4 plies, each with its own material. A physical material is associated with a hatch pattern, used for floor plans and sections. Another pattern is specified for elevations.

12 Assign the **Concrete, Reinforced** composition to the red box.



13 Update and open the section.



14 Go to your layers and execute the PURGE command to remove the unused layers.

	Current	Name	D..	On/Off	Freeze	Locked	Color
1		0					White
2		Background_Air					White
3		Background_Concrete_ Reinforced					White
4		Background_Facing Bricks_ Hand-form					White
5		Background_Insulation_ Polyurethane					White
6		Background_Layer1					Green
7		Background_Layer2					Red
8		Background_Supporting Wall_ Brick					White
9		BIM_SECTIONS_PREVIOUS					Magenta
10		BIM_VIEWPORTS					White
11		Boundary_Air					White
12		Boundary_Cavity Wall_ Brick					White
13		Boundary_Facing Bricks_ Hand-formed					14
14		Boundary_Insulation_ Polyurethane Fo					42
15		Boundary_Layer1					Green
16		Boundary_Supporting Wall_ Brick					14
17		Fill_Facing Bricks_ Hand-formed					21
18		Fill_Insulation_ Polyurethane Foam					42
19		Fill_Layer1					Green
20		Fill_Supporting Wall_ Brick					11
21		Title Block Frame					175
22		Title Block Labels					157
23		Title Block Text					251

You see the three categories (**Background**, **Boundary**, and **Fill**) again. Now the second part of the layer name is the name of the physical material used in the solid, instead of the layer name of the box:

- **Background_Facing Bricks_Hand-formed**, **Background_Insulation_Polyurethane Foam**, and **Background_Supporting Wall_Brick**.
- **Boundary_Facing Bricks_Hand-formed**, **Boundary_Insulation_Polyurethane Foam**, and **Boundary_Supporting Wall_Brick**.
- **Fill_Facing Bricks_Hand-formed**, **Fill_Insulation_Polyurethane Foam**, and **Fill_Supporting Wall_Brick**.

15 Set the **Generate elevation fill** property of the section **ON** to add an elevation hatch.

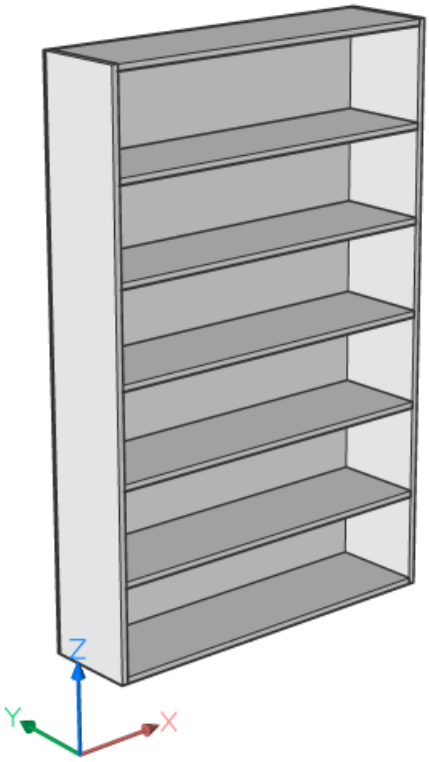
16 Update the section.

17 The outline of the second box is now hatched and the layer **Elevation_Concrete_Reinforced** is added.

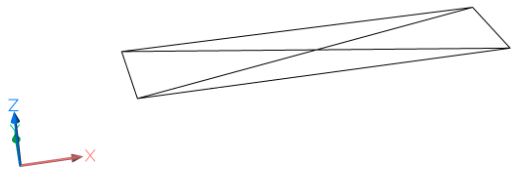
20.7.17.3 Procedure: understanding the BRX_2D layers

1 Open a millimeter template **BIM-mm.dwt** and save the drawing.

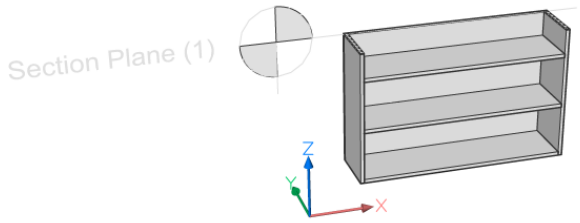
2 Drag the component **Furniture Bookshelf** into the drawing. You can find the component in the **Furnishing Elements** category in the **Library** panel (LIBRARYPANELOPEN command).



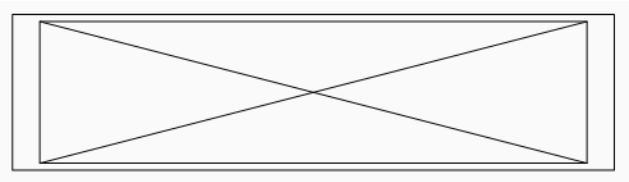
3 If you now show and isolate the **BRX_2D_Symbols** layer, you will see a symbol drawn at the insertion point of the bookshelf.



4 Now show all layers again (LAYUNISO command) and make a section through the bookshelf.



5 Generate and open the Plans drawing.



6 You will see (if the **Boundary_BRX_2D_Symbols** layer is on) that only the symbol is drawn onto the plan, and not the real section. It happens because this component had a 2D symbol on the BRX_2D layer attached to it.

7 Now return to your 3D model and Open a copy of the bookshelf.

8 Hide the bookshelf itself and make sure the **BRX_2D_Symbols** layer is on.

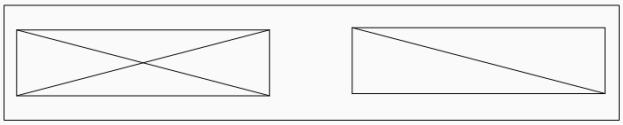
9 Delete one of the lines (of the cross).

10 **(Optional)** Draw extra 2D lines on the **BRX_2D_Symbols** layer.

11 Save the drawing.

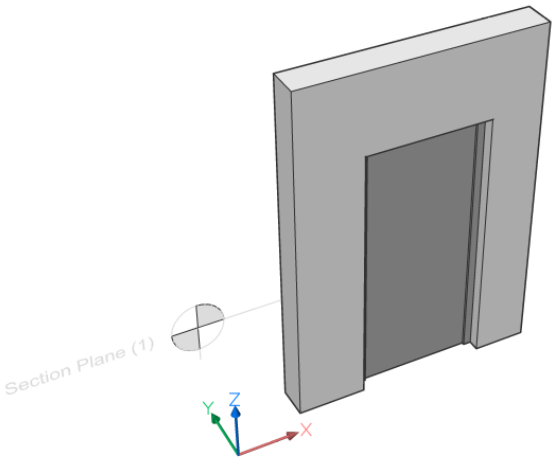
12 Go back to the original 3D model and type in BMINSERT in the Command line and go to the file path where you just saved the new bookshelf. Click **OK** and choose a point to insert the bookshelf.

13 Regenerate the section and open it. Widen the viewport if necessary, by dragging the existing viewport.



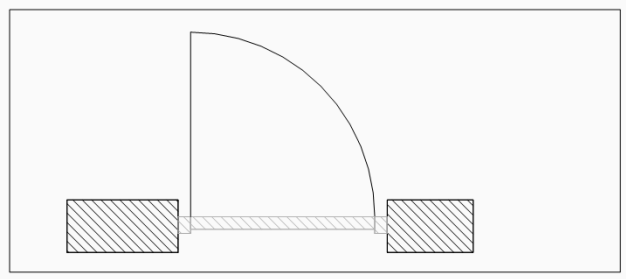
14 You will now see that the second bookshelf has the adapted symbol displayed.

15 Draw a wall in your 3D model and insert the door **Door Ext Plain** into this wall.



16 Repeat steps 3 to 5 but now use the **BRX_2D+_Swing** layer instead of the **BRX_2D_Symbols** layer.

17 The generated plan displays the symbol of the door swing as well as the door section itself. The **BRX_2D+** layer of a component makes sure you can show the real section together with a symbol.
   - Symbols on a layer with prefix **BRX_2D_** will be used to replace the geometry resulting from the actual section.
   - Symbols on a layer with prefix **BRX_2D+_** will be added to the section result. Only those 2D entities in a plane parallel to the section plane are.



20.7.17.4 Procedure: customizing the colors and line weights of hatch patterns and lines

1 Firstly, in the above procedure 'Understanding the created layers', you saw that for objects without a composition, the colors and line weights of the **Fill_**, **Background_**, and **Boundary_** layers are copied from the layer properties in the 3D model.

2 However, for objects with a composition, the default colors are set by a template found by typing in SUPPORTFOLDER command and browsing to *Bim/Sections/_SectionSettings.dwg*.

3 Open this drawing.

4 Go to the **Layers** panel, you will find a list of all the possible layers that can be created for every material present in the material library. They all have a color and line weight. Changing the color or line weight and saving the file will change the default color/line weight for this layer in your future generated drawings. You can fully customize how your drawing will look for every type of line (**Fill**,

Background, **Boundary**, or **Elevation**) per material.

195		Fill_Ceiling_ Plasterboard					253
196		Fill_Concrete_ Beam and Pot Floor					85
197		Fill_Concrete_ Plain					85
198		Fill_Concrete_ Precast					85
199		Fill_Concrete_ Reinforced					85
200		Fill_Copper					151
201		Fill_Facing Bricks_ Concrete Blocks					21
202		Fill_Facing Bricks_ Hand-formed					21
203		Fill_Facing Bricks_ Sand-lime					21
204		Fill_Floor Joist_ Wood					33
205		Fill_Floor_ Boarded					9
206		Fill_Floor_ Ceramic Tiles					251
207		Fill_Floor_ Concrete Tiles					251
208		Fill_Floor_ Laminate					251
209		Fill_Floor_ Parquet					251
210		Fill_Floor_ Stone Tiles					43
211		Fill_Freestone					43
212		Fill_Glass					141
213		Fill_Glass Brick					141
214		Fill_Gold					151
215		Fill_Granite					43
216		Fill_Gravel					251
217		Fill_Gypsum Board					251
218		Fill_Insulation_ Extruded Polystyrene					42
219		Fill_Insulation_ Glass Wool					42
220		Fill_Insulation_ Polyurethane Foam					42
221		Fill_Insulation_ Rock Wool					42
222		Fill_Iron					251
223		Fill_Lead					151
224		Fill_Marble					43

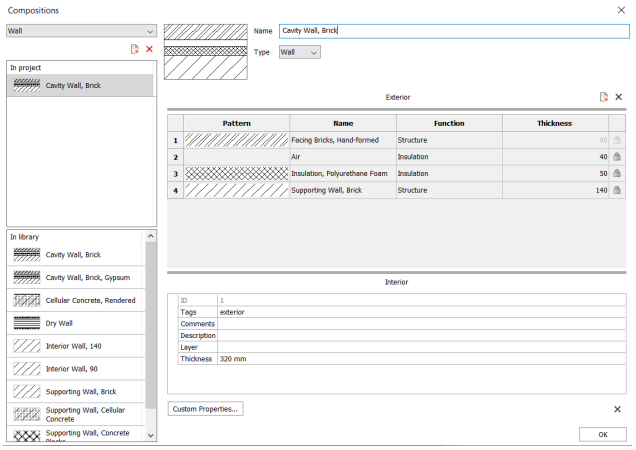
20.7.17.5 Procedure: customizing the hatch patterns

1 Open the drawing made in the above procedure 'Understanding the created layers'.

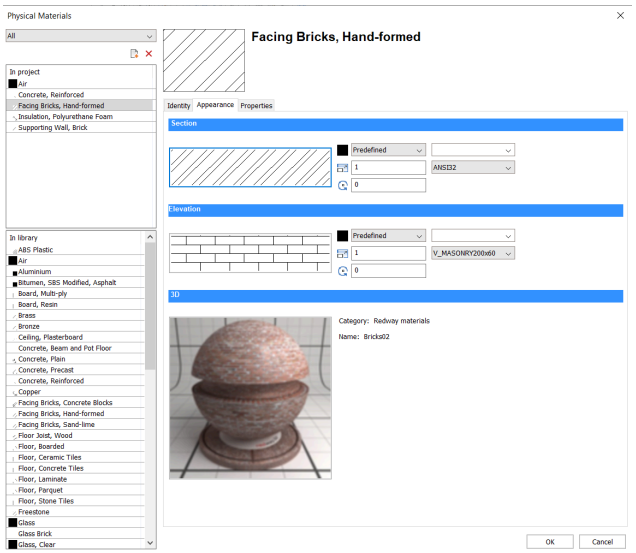
2 Open the **Compositions** panel.

3 Double-click the composition **Cavity Wall, Brick**.

4 In the procedure 'Understanding the created layers said that you can change the hatch patterns of the materials here. How you need to do this will follow in the next steps.



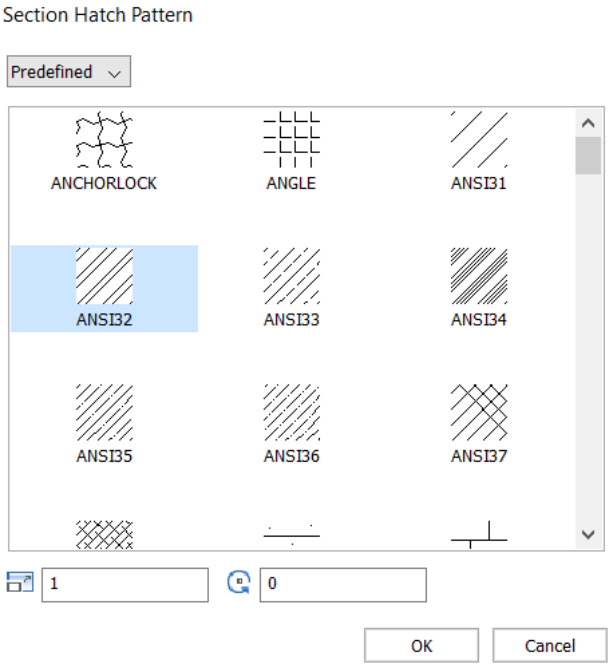
5 Double-click on the row of **Facing Bricks, Hand-formed**. The **Physical Materials** dialog will pop-up on the material **Facing Bricks, Hand-formed**. You could have entered this dialog box by typing in the command BLMATERIALS, after which you could have selected the **Facing Bricks, Hand-formed** material.



6 Go to the tab **Appearance**.

7 Under the title **Section**, double-click on the hatch pattern.

8 You can now choose your new standard section hatch representation for this particular material. In every composition with this material, the same hatch representation will be used.



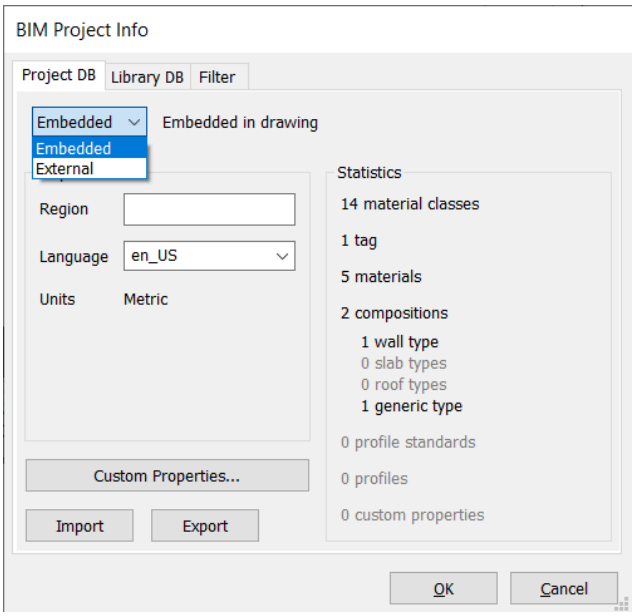
9 If you want to change the appearance of an elevation hatch, you can do so by double-clicking the hatch pattern under the **Elevation** title. The outer most visible material of a composition will be used to determine the elevation hatch.

10 Note that the representation of these hatches will not be copied into new drawings as the **Physical Materials** dialog is not a template, but rather a way to define the representation in a specific drawing.

If you would like to store this information, it is recommended to create a library of compositions. The best way to do so is to make a drawing where you attach the compositions you would like to store to little cubes. Make sure each composition has the hatch appearance you want by going through the process described above.

11 Once this is finished, type in BIMPROJECTINFO in the Command line. You will enter the **BIM Project Info** dialog box.

12 In the **Project DB** tab, switch the value for the database from **Embedded** to **External**. You will be asked to save the database on your computer and give it a name, for example, 'CompositionsTemplate.bsyslib'.



13 If you now want to use this database (Project DB) in a new drawing, switch the value for the database from **Embedded** to **External**. This time select the .bsyslib file you created.

20.7.17.6 Procedure: customizing tags per entity type for all section drawings

For this procedure, please visit the following procedures from the BIM Tags page:

- Create the source data for a BIM entity type.
- Customize tag blocks.
- Edit the source data to customize BIM Tags for all section drawings.

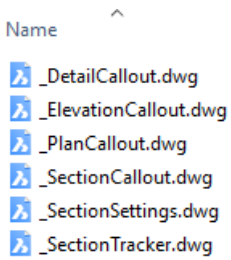
To change the tags in the current drawing, visit the procedure 'Customize tag blocks for the current drawing only'.

20.7.17.7 Procedure: customizing call-out blocks

1 Source files of the various call-out blocks and the section tracker exist in the **..\Support\Bim\Sections** subfolder of the **Roamable Root** folder.

By default this is: **C:\Users\<user_name>\AppData\Roaming\Bricsys\BricsCAD\Vxx\en_US\Support**

\Bim\Sections. You can easily access this file by typing in SUPPORTFOLDER command and browsing to **Bim** > **Sections**.



2 You can edit these files to customize the layout of the call-outs and the section tracker blocks.

3 When a drawing is generated, the needed call-out blocks are created from the call-out blocks source file(s). When a BIM section is updated, the existing call-out blocks in the target drawing are used (type BEDIT to choose one to edit).

20.7.17.8 Procedure: customizing the sheet set template

1 The sheet set templates are stored in the **C:\Users\<user_name>\AppData\Local\Bricsys\BricsCAD \Vxx\en_US\Templates** folder.

2 If you want to edit an existing template, you have two options. Either you edit it directly, or you make a copy.
   - To edit it directly, you can go to **Open** under the BricsCAD icon, choose **.dwt** in the **Files of type** drop-down list and choose the file you want to edit. You can then make the modifications and save the file.
   - To make a copy, double-click on the .dwt file you want to copy. A new .dwg with this template will open. Make your modifications and **save your file as .dwt**. Save it in the templates folder to easily use the copied template in a new drawing.

3 If you want to start from scratch, using a wizard, follow the procedure 'To create a drawing using a wizard'. After the creation and modification of the dwg, save it as a .dwt file in your sheet set templates folder.

4 If you have saved the template .dwt to the sheet set templates folder, you will see it appear when you use the **New Wizard** to open a new drawing. You will be able to select it as the template for the creation of generated drawings in the sheet set set-up of the **Project Browser**.

5 What can you store into a template?

6 You can make a title block for your template, with sheet set properties, by following the procedure 'Using sheet set properties in a title block'. Do this for every layout in your template. You can also store the following things by using them in your .dwt file:
   - Layers
   - Colors
   - Line types
   - Hatch patterns

- Blocks
- Text styles
- Dimension styles
- Page set-ups

21. BricsCAD Mechanical

21.1 Cosa c'è di nuovo

21.1.1 Panoramica

BricsCAD Mechanical V21 offre tutti gli strumenti nuovi e migliorati disponibili in BricsCAD V21 e ulteriori miglioramenti specifici per i flussi di lavoro meccanici.

Questa sezione offre una panoramica di tutte le nuove funzionalità e miglioramenti nei strumenti di **BricsCAD V21 Mechanical**:

Gestore BOM alla pagina 1539

Librerie di parti alla pagina 1542

Parametri e Vincoli alla pagina 1543

Parti standard alla pagina 1543

Lamiera alla pagina 1543

Per informazioni sulle nuove funzionalità e miglioramenti in **BricsCAD V21 Core e Civile**, andare alla sezione Novità in BricsCAD V21. Contiene i seguenti articoli:

Overview alla pagina 31

User interface alla pagina 32

Productivity alla pagina 35

Drafting alla pagina 49

Modeling alla pagina 51

Civil tools alla pagina 60

Performance and compatibility alla pagina 65

Per informazioni sulle nuove funzionalità e miglioramenti in **BricsCAD V21 BIM**, visitare la sezione Novità di BricsCAD BIM V21. Contiene i seguenti articoli:

Overview alla pagina 967

Modeling techniques alla pagina 968

Building data alla pagina 975

Project collaboration alla pagina 984

Design documentation alla pagina 985

Point clouds alla pagina 994

21.1.1.1 Note di rilascio

Clicca qui per una panoramica delle note di rilascio.

21.1.1.2 Diagnostica e raccolta dati di utilizzo

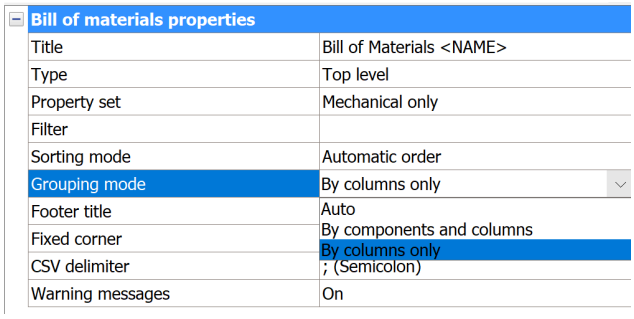
Molti dei miglioramenti sono stati ispirati da richieste e suggerimenti da parte degli utenti BricsCAD. In Bricsys, continuiamo ad espandere i nostri metodi di raccolta dei feedback degli utenti con nuovi strumenti di raccolta dati. Quando si lancia BricsCAD per la seconda volta, viene visualizzata una finestra di dialogo che invita a partecipare al Programma di raccolta dati di diagnostica e utilizzo. Il processo di raccolta dei dati consente di condividere con Bricsys, dati diagnostici e di utilizzo anonimi per migliorare BricsCAD. Puoi revocare il tuo consenso e far rimuovere i tuoi dati dai nostri database in qualsiasi momento.

21.1.2 Gestore BOM

Creare e modificare le distinte base è più facile che mai.

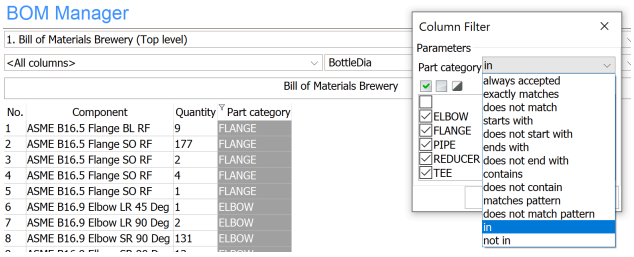
21.1.2.1 Raggruppamento

Con l'aggiunta della proprietà raggruppamento, è ora possibile raggruppare rapidamente il contenuto della tabella BOM in righe raggruppate in base alle proprietà comuni delle colonne esistenti.

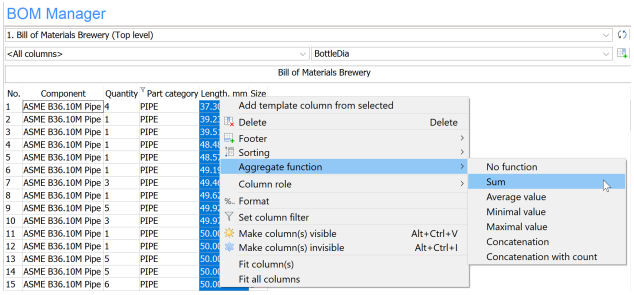


21.1.2.2 Colonne

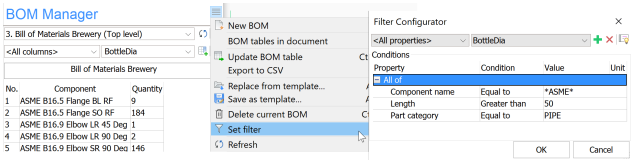
È ora possibile impostare uno dei diversi filtri di colonna predefiniti per qualsiasi colonna in una BOM.



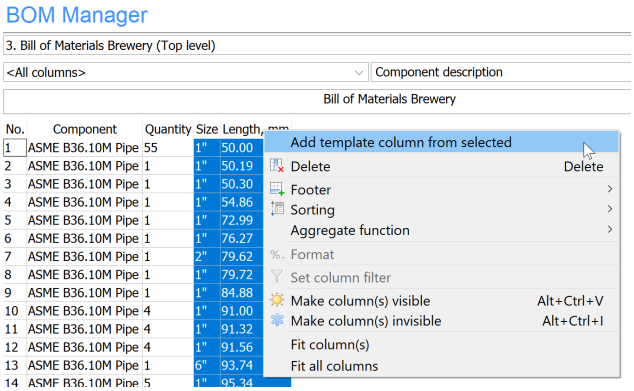
Le colonne aggregate sono ora supportate nelle tabelle BOM di livello superiore e inferiore create con il comando BMBOM. Consentono di calcolare la somma, i valori minimi, massimi o medi di una determinata proprietà rispetto al raggruppamento corrente.



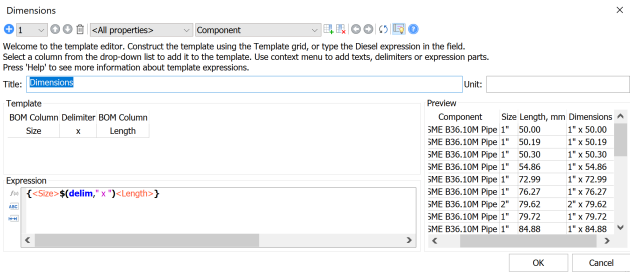
Un'opzione nel menu principale del Gestore BOM apre la nuova finestra di dialogo Configuratore filtro. Nel Configuratore filtro è possibile visualizzare il contenuto della tabella BOM; le espressioni jolly sono supportate per le proprietà stringa in un filtro BOM.



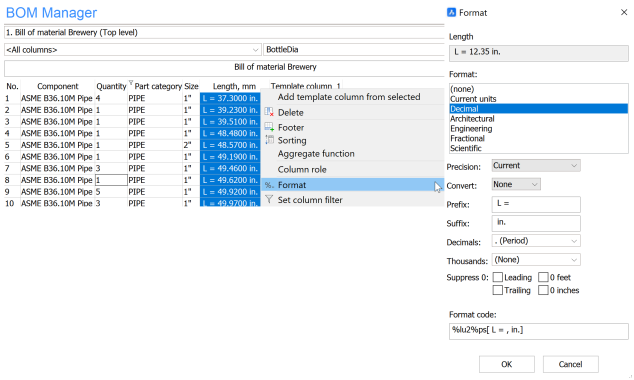
Le Colonne del Modello sono un nuovo tipo di colonna valutata, che consente di combinare i valori di più proprietà in una singola colonna BOM, come testo, utilizzando espressioni DIESEL.



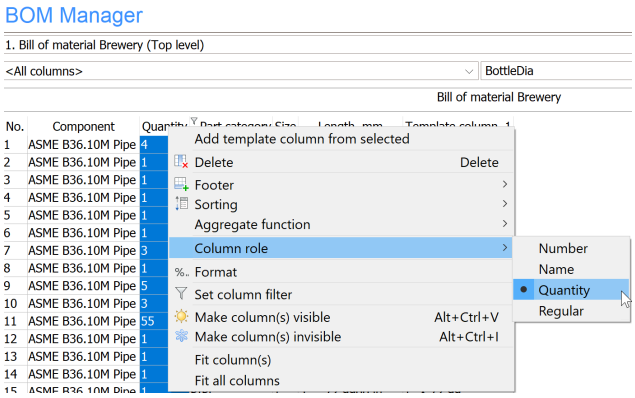
È possibile configurare e visualizzare in anteprima le espressioni di colonna formula o modello utilizzando la finestra di dialogo Editor espressioni.



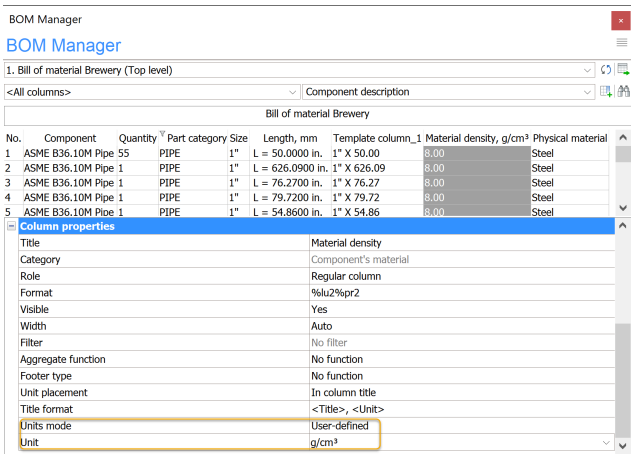
Con l'aggiunta della finestra di dialogo Formato, è ora possibile personalizzare l'espressione dei dati nelle tabelle BOM con prefisso, suffisso e delimitatori.



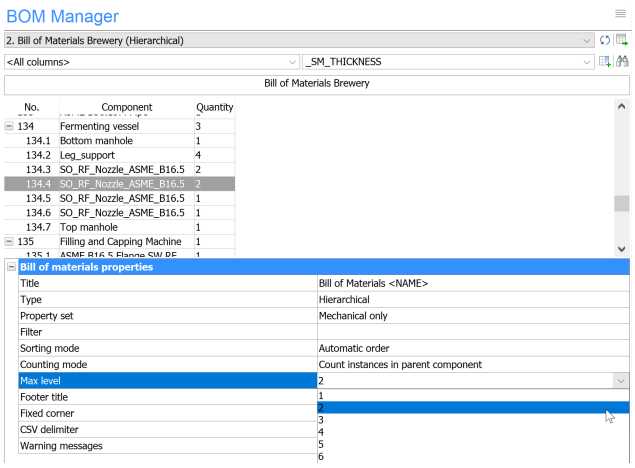
La nuova opzione Ruolo Colonna consente di specificare quale colonna deve essere l'origine dei richiami nelle viste di disegno.



Ora è possibile selezionare un'unità per la densità del materiale, sia nella finestra di dialogo Materiali Fisici che nelle tabelle BOM nella sezione delle proprietà della colonna. Le unità supportate sono: kg/m³, kg/dm³, g/cm³, lbs/ft³ e lbs/in³.

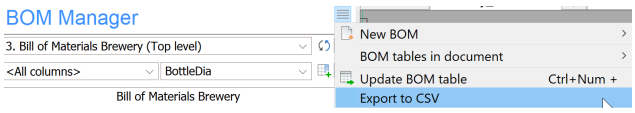


È stata introdotta un'opzione per limitare il livello riportato di una BOM gerarchica.



21.1.2.3 Esportazione

Ora è possibile esportare la tabella BOM in un file CSV.

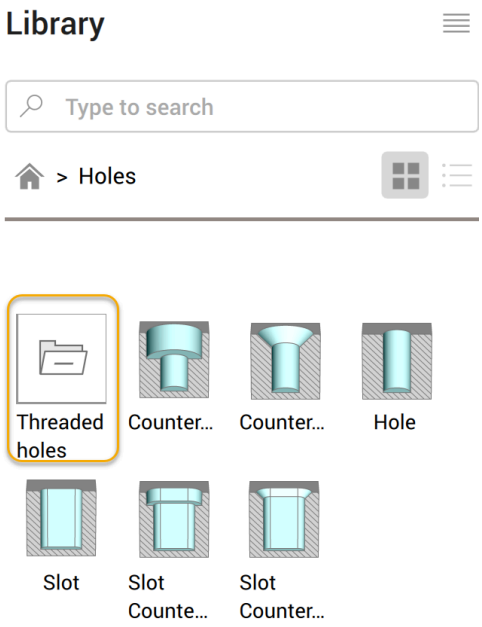


21.1.3 Librerie di parti

Il lancio di BricsCAD V21 ha registrato un aumento significativo del numero di parti standardizzate disponibili.

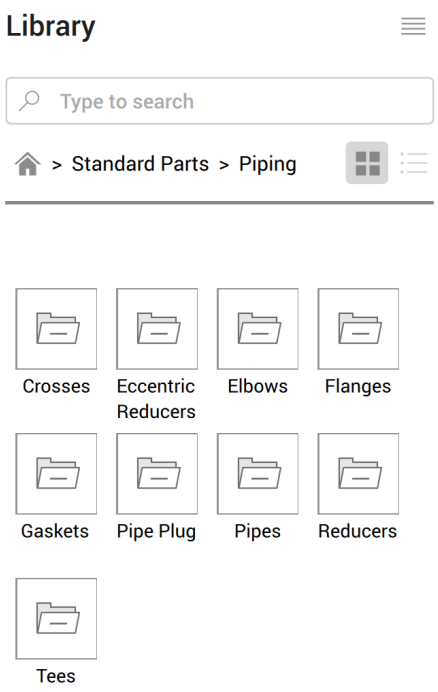
21.1.3.1 Fori

È stata aggiunta una libreria di componenti di fori filettati.



21.1.3.2 Tubazioni

È stata aggiunta una libreria di componenti delle tubazioni.



21.1.4 Parametri e Vincoli

21.1.4.1 Tubazioni

Insieme al rilascio delle parti standardizzate delle tubazioni, un nuovo comando BMCONNECT vincola automaticamente le parti selezionate in base alla loro vicinanza l'un all'altra.

21.1.5 Parti standard

I vincoli vengono ora ripristinati automaticamente dopo aver modificato i parametri di una parte standard.

21.1.6 Lamiera

Gli strumenti per la lamiera in BricsCAD Mechanical V21 offrono maggiore flessibilità e flussi di lavoro più rapidi.

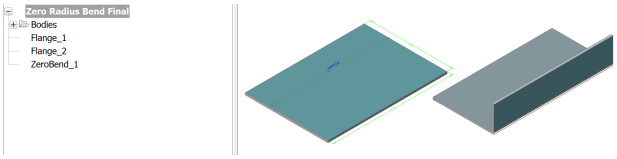
Ad esempio, una specifica tabella di piegatura può essere assegnata a ogni solido e le arti con uno spessore diverso possono esistere nello stesso blocco.

21.1.6.1 Smusso

La nuova funzione Smusso aggiunta è supportata nel comando SMCONVERT, nel comando SMREPAIR e nel comando SMUNFOLD.

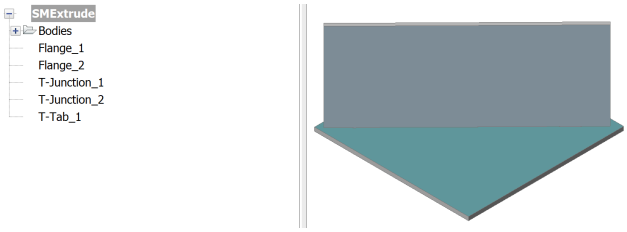
21.1.6.2 Piegatura Raggio Zero

Una nuova caratteristica PiegaturaZero consente di creare una piegatura senza raggio interno per i componenti Lamiera.



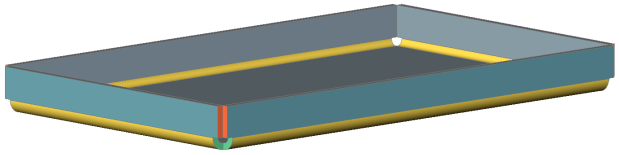
21.1.6.3 Giunzione-T

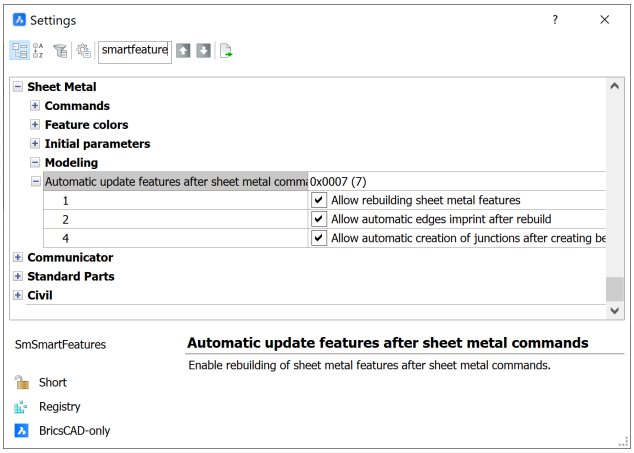
È possibile creare una nuova caratteristica di Giunzione-T utilizzando SMEXTRUDE su una polilinea a segmento singolo che giace su una flangia per creare una nuova flangia ortogonale all'esistente. Utilizzando la caratteristica di Giunzione-T, è possibile convertirla in una Linguetta-T che aggiungerà automaticamente linguette alla flangia ortogonale e rimuoverà il materiale necessario dall'originale.



21.1.6.4 Caratteristiche Intelligenti Aggiornate

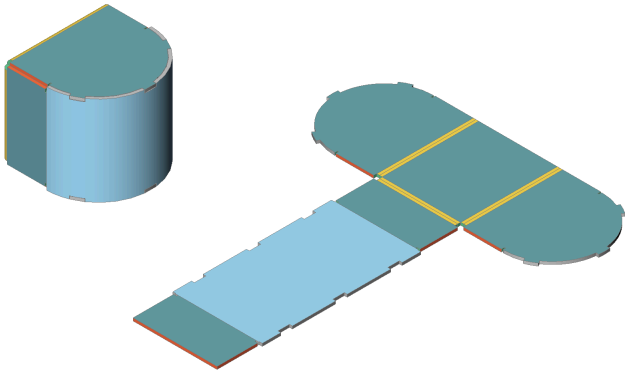
È ora possibile creare disegni di lamiera ancora più velocemente utilizzando il comando SMSMARTFEATURES aggiornato che consente di creare automaticamente giunzioni quando si utilizza lo strumento di piegatura.



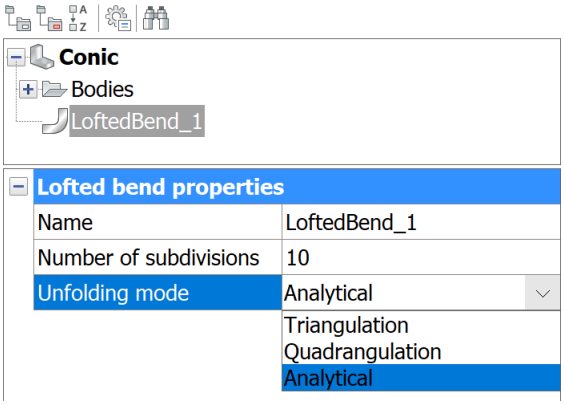


21.1.6.5 Linguette

Il comando SMTABCREATE dispone ora di un'opzione per creare una linguetta curva su un bordo rigido curvo, collegando una flangia e una curva, comprese le piegature loft.

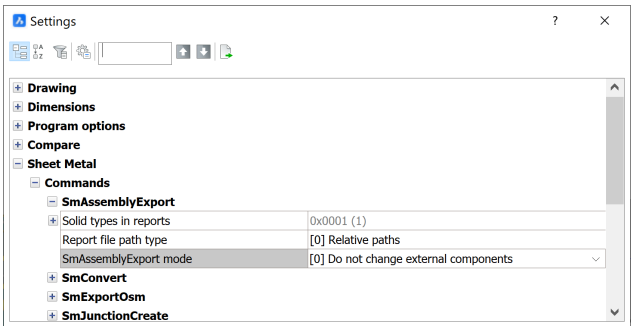


Il comando SMUNFOLD offre un comportamento migliore per le curve coniche con loft. Ora possono essere spiegate in modo analitico, per evitare di suddividerli in un determinato numero di curve e flange.



21.1.6.6 Esportazione Assieme

La variabile di sistema SMASSEMBLYEXPORTMODE consente di mantenere le caratteristiche riconosciute per le parti in lamiera e in lamiera modellata povera quando si utilizza il comando SMASSEMBLYEXPORT.



21.2 Guida introduttiva

21.2.1 Area di lavoro meccanica

21.2.1.1 Avvia un progetto di progettazione meccanica

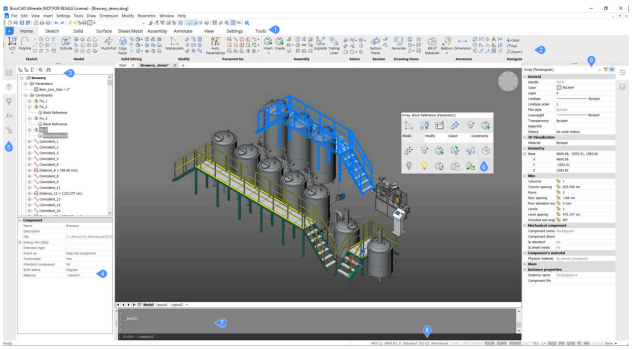
È possibile avviare un progetto di progettazione meccanica:

1 Avvia BricsCAD.

Viene visualizzata la finestra *Avvio di BricsCAD*.

2 Cliccare su **Inizio** nella sezione **Mechanical**.

21.2.1.2 Finestra applicazione BricsCAD mechanical



1 **Schede della barra multifunzione**: fare clic per passare a una determinata scheda.

2 **Barra Multifunzione**: ogni scheda contiene diversi pannelli, che combinano strumenti simili.

3 **Navigatore meccanico**: permette di navigare attraverso una rappresentazione ad albero del modello, con la possibilità di evidenziare un particolare nodo nell'area del modello e modificarne le proprietà meccaniche.

4 **Proprietà Meccaniche** di un particolare nodo: alcune sono di sola lettura. Controllare le proprietà del nodo principale nella struttura.

5 **Pannelli degli strumenti agganciabili**: navigatore meccanico e Libreria.

6 **Menu cursore Quad**: permette di applicare uno strumento particolare quando si passa il mouse sull'entità interessata.

7 **Barra dei Comandi**: digitare il comando.

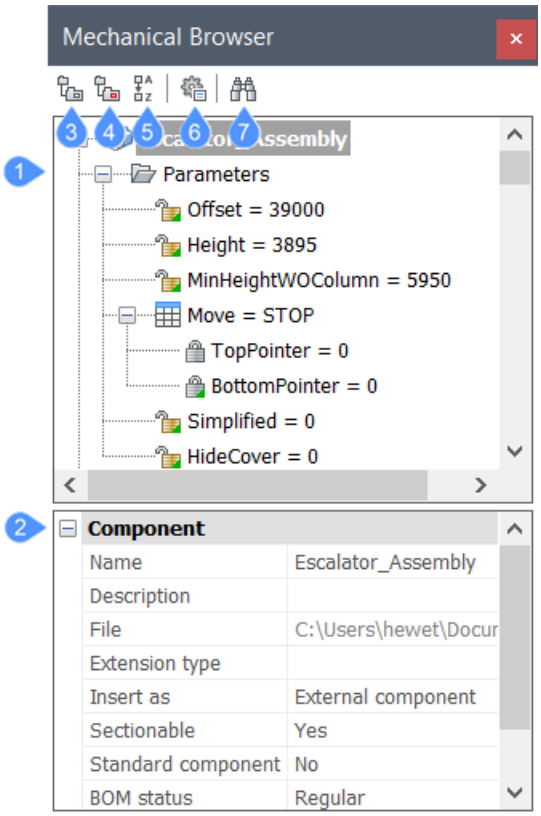
8 **Barra di stato**: stato e commutazione veloce delle opzioni.

9 **Pannello Proprietà**: Regola le proprietà non meccaniche delle entità selezionate, come colore e layer.

Figura 4: Navigare in BricsCAD Mechanical

21.2.2 Mechanical browser panel

The Mechanical browser panel offers a central location to view and modify parametric properties of mechanical parts and assemblies.



1 Parameter list

2 Properties

3 Group by entity

4 Group by type

5 Sort

6 Settings

7 Show search

21.2.2.1 Parameter list

Displays a list of all the parameters and constraints in the drawing. They are organized in expandable categories which may include user parameters,2Ddimensional constraints, 3D dimensional constraints, 3D geometric constraints,block parameters and arrayparameters.

A context menu displays when right-click the main component name:

Update
Updates the hierarchy of mechanical components for the current drawing in case referenced drawing files of sub-components have been modified.

Visual style > All by Viewport
Applies the current visual style to all components in the assembly.

Insert standard hardware
Inserts a standard hardware part as a mechanical component in the current drawing.

Switch all to local
Converts all external components in the model to local components.

Switch all to external…
Switches all internal components to external components.

Select all
Selects all components with the same definition.

Select same
Selects all components with the same name and the same parameter values.

Highlight all
Highlights all components with the same definition.

Highlight same
Highlight all components with the same name and the same parameter values.

Create exploded view
Creates a block with an exploded representation of the current.

Dissolve
The selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Delete
This is the analogue of the SMDELETE command. In this case the feature is removed from the browser and geometry is changed depending upon the type of the feature.

Disable
The feature will remain in the Mechanical Browser, but the part will behave like the feature was dissolved. It is beneficial in contrast with dissolve, that feature still updates on geometry changes and you can turn it on when you want, avoiding recreating it or recognizing by SMCONVERT.

Right-clickmenusoffer additional toolsfor components:

Open
Opens the referenced drawing.

Open a copy
Opens a copy of a component insert as a new drawing.

Update
Reloads all referenced components from external files and updates BOM tables.

Replace…
Replaces a component insert.

Nota: Replacing a local insert turns it into an external insert.

Set material to component
Opens the Physical Materials dialog box, which allows you to assign a physical material to the component.

BOM status
Controls the appearance of the component in BOM tables.

Right-clickmenusoffer additional toolsfor parameters:

Geometry-driven
If checked, it makes the parameter geometry-driven.

Create design table
Creates a design table to drive parametric block parameters.

Animate
Animates models by means of parameters.

Link to parameter
Links subcomponent parameter to the main level parameter.

21.2.2.2 Properties

Specifies the properties of the selected item.

21.2.2.3 Group by entity

Groups 3D constraints by entity. Expand an entity to view its associated constraints.

21.2.2.4 Group by type

Groups 3D constraints by type. Expand a constraint to view the affected entities.

21.2.2.5 Sort Alphabetically

Lists mechanical components and 3D constraints alphabetically. Otherwise, they are listed in the order they are added to the assembly.

21.2.2.6 Settings

Expressions of constraints
Controls whether the numeric value or the assigned parameter name displays.

Components parameters
Expressions at sub-components control the visual representation of sub-component parameters: numeric value or assigned parameter name.

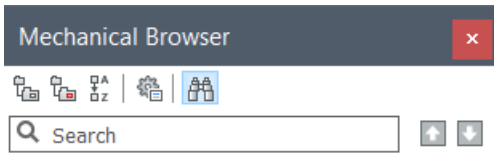
Expressions of components parameters
Sub-component parameters control the visibility of sub-component parameters.

Sub-components of standard parts
Parameters at properties add a parameters section for the selected instance to the Mechanical browser properties

21.2.2.7 Show search

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

Search field
Searches the parameter list for the character string you enter. The arrow tools to the right of the search field enable you find the next or previous occurrence of the text string.



21.2.3 Confronto 3D

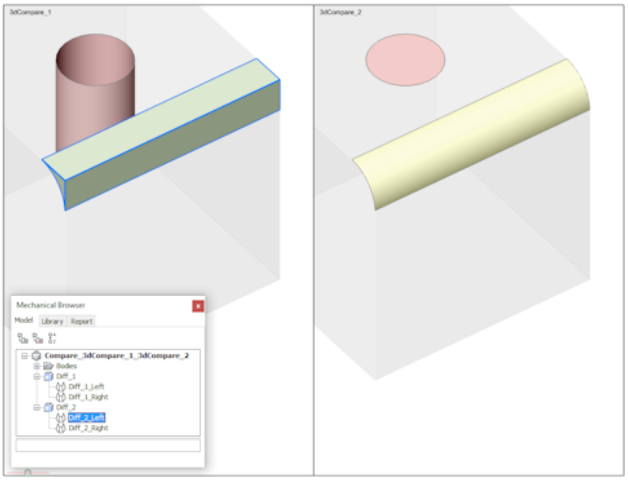
Comando: *3DCOMPARE*.

È possibile confrontare due modelli 3D, ad esempio solidi 3D e superfici memorizzati in 2 file DWG, inclusi i modelli importati in BricsCAD da altri formati di file. Lo strumento può elaborare modelli di grandi dimensioni e segnalare le differenze nel **Navigatore Meccanico**. Sono supportate diverse modalità di visualizzazione.

21.2.3.1 Definizioni

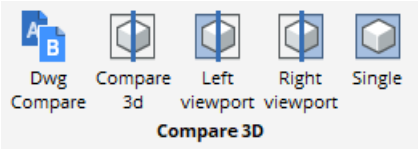
Un modello 3D è un file DWG con uno o più solidi o superfici 3D. Quando si confrontano due modelli 3D, BricsCAD tenta innanzitutto di allinearli automaticamente nello spazio 3D per ottenere la massima somiglianza. Successivamente, tutte le facce di questi solidi 3D e superfici vengono confrontate geometricamente con una data tolleranza. Se per ogni faccia del primo file esiste una faccia uguale nel secondo file e viceversa, tali modelli sono considerati uguali. Tuttavia, se per una determinata faccia in un file non è presente una faccia uguale in un altro file, o se solo una parte di questa faccia è uguale a una parte di una faccia nell'altro file, viene rilevata una differenza. Pertanto, una differenza è una faccia intera o una parte di una faccia di un solido 3D o di una superficie, per la quale non esistono parti uguali di facce complete trovate nell'altro file. Due o più differenze sono raggruppate insieme se sono geometricamente adiacenti l'una all'altra.

Un nodo differenza viene creato nel **Navigatore Meccanico** per ogni gruppo di differenze adiacenti. Se si espande questo nodo, è possibile vedere le particolari differenze di cui è costituito. Selezionando la differenza nel **Navigatore Meccanico**, la caratteristica corrispondente viene evidenziata nel disegno.



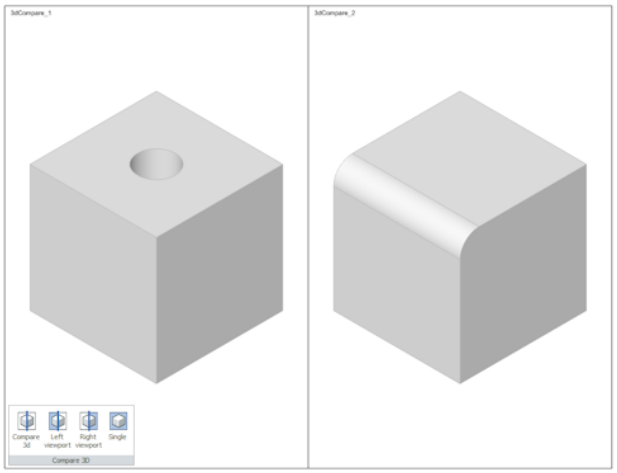
21.2.3.2 Barre degli Strumenti

È possibile accedere alla procedura di Confronto 3D utilizzando gli strumenti del pannello **Confronta 3D** nella scheda **Strumenti**.



21.2.3.3 Visualizzazione

Utilizzare il comando *3DCOMPARE* per creare un nuovo documento DWG denominato **Confronta_<File_1>_<File_2>.dwg**. I modelli confrontati vengono collegati come riferimenti esterni a questo disegno. Un layout speciale denominato **Confronto** viene creato automaticamente con due finestre, una per ogni disegno, illustrando le differenze tra i disegni utilizzando i colori.

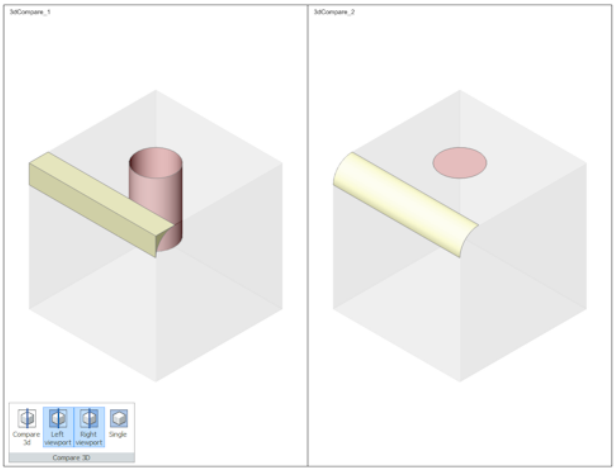


La visualizzazione è controllato tramite la variabile di sistema *3DCOMPARIMODE*. Se il valore di *3DCOMPAREMODE* è diverso da 0 (zero), il modello corrispondente diventa semitrasparente, mentre le differenze vengono visualizzate utilizzando colori diversi. Ogni differenza è rappresentata nel suo colore specifico. È possibile ingrandire una particolare differenza utilizzando il menu contestuale del Navigatore Meccanico.

Per visualizzare le differenze:

1 Fare clic sul pulsante dello strumento **Finestra Sinistra** ( ) e/o sul pulsante dello strumento **Finestra**

Destra ( ).



2 Fare doppio clic su una delle finestre.

3 Utilizzare il mouse/scorciatoie da tastiera per manipolare la visualizzazione.

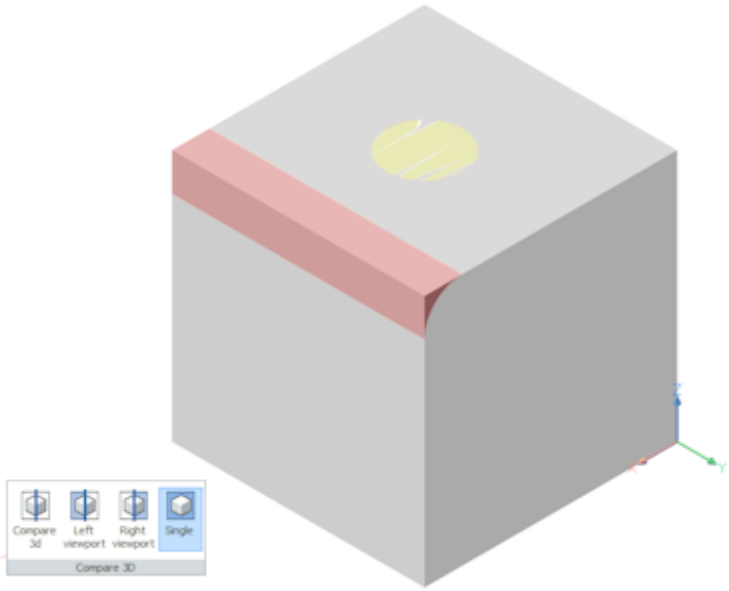
Esempio: Zoom avanti/indietro o rotazione.

La visualizzazione nell'altra finestra viene sincronizzata automaticamente. Questo consente di confrontare in dettaglio la stessa parte in entrambi i modelli.

In alternativa, è possibile:

Fare clic sul pulsante dello strumento **Singola** ( ).

Vengono visualizzate le differenze nello spazio modello.

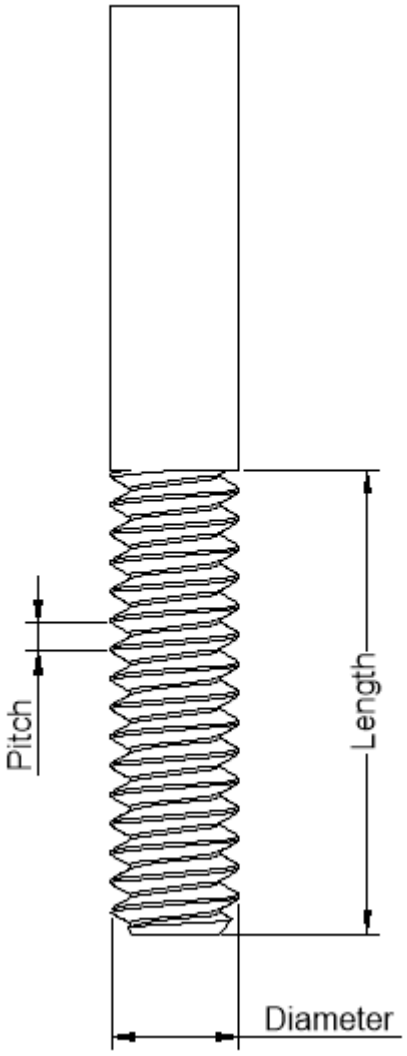


21.3 Progettazione di Parti

21.3.1 Caratteristica filetto

Utilizzare il comando DMTHREAD per creare una caratteristica filetto.

L'immagine seguente mostra la struttura del filetto:



Una caratteristica filetto possiede i seguenti parametri:

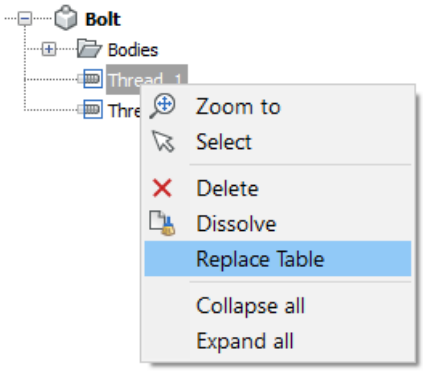
- **Filettatura standard**

  È in un gruppo di filetti standard. Questo elenco può essere esteso con un filetto personalizzato. Se cambi lo standard, la geometria verrà ricostruita automaticamente.
- **Passo**del filetto.
- **Lunghezza**del filetto.
- **Diametro**del filetto.

- **Tipo** del filetto: maschio o femmina. Se il tipo è impostato come auto, il tipo verrà determinato automaticamente.

21.3.1.1 Aggiungere una configurazione filetto

1 Cliccare il pulsante destro del mouse sulla caratteristica nel **Navigatore Meccanico**.



2 Selezionare **Sostituisci Tabella**.

3 Selezionare un file CSV con la seguente configurazione:

```
Dimensione,Diametro,Passo Nome1, Valore_diametro1, Valore_passo1 Nome2,
 Valore_diametro2, Valore_passo2 Nome3, Valore_diametro3, Valore_passo3
```

21.3.2 Metodi di progettazione delle Parti

BricsCAD Mechanical ti consente di creare e modificare modelli 3D.

Metodi per creare un modello 3D:

- Crea un solido primitivo 3D, come un parallelepipedo e un cilindro.
- Disegnare un profilo 2D ed estruderlo o rivoluzionarlo per creare un solido 3D.
- Modellare una superficie 3D ed ispessirla per creare un solido 3D, o unire più superfici insieme per creare un solido 3D.
- Importare la geometria di solidi 3D e di superfici esistente da altri programmi CAD.

Metodi per modificare e aggiungere ulteriori dettagli ai tuoi modelli 3D:

- Premendo/Tirando, spostando e ruotando le facce.
- Creando Raccordi o Smussi sugli spigoli del modello.
- Applicando Fori.
- Aggiungendo protrusioni e depressioni e utilizzando le operazioni Booleane, strumenti di Modellazione Diretta per Modificare e Deformare.

Nessuna dipendenza padre-figlio = approccio senza cronologia

BricsCAD Mechanical non crea dipendenze padre-figlio tra primitive geometriche e feature. Tutte le modifiche apportate alla parte sono locali; non impongono la rigenerazione della geometria. Questo approccio senza storia ti consente di concentrarti sulla progettazione invece di pensare a come può essere modificato in futuro.

Vincoli 3D = Parametrico

BricsCAD Mechanical fornisce un potente set di strumenti per parametrizzare la parte aggiungendo vincoli 3D alle facce e agli spigoli e collegandoli con parametri globali utilizzando espressioni e tabelle di progettazione.

21.3.3 Schizzo (Sketching)

Il gruppo di strumenti Schizzo consente di disegnare profili 2D, che possono essere estrusi o usati in rivoluzioni per creare solidi 3D e superfici, o per aggiungere/sottrarre volume a/da entità esistenti. Gli strumenti schizzo sono disponibili nella scheda **Schizzo** della barra multifunzione dell'area di lavoro **Meccanico**:

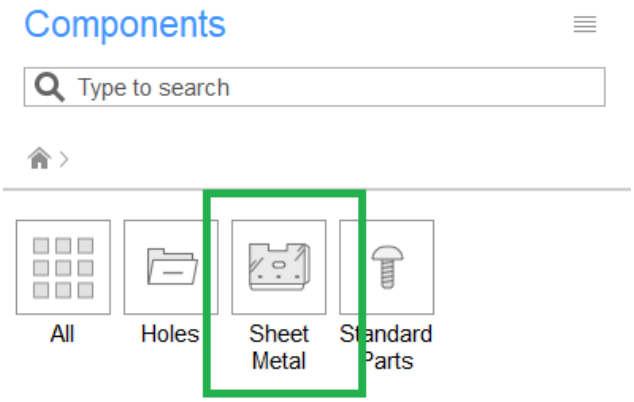


Figura 5: Barra Multifunzione per area di lavoro Mechanical

Gli schizzi vengono creati nel piano XY del Sistema di Coordinate Utente (UCS) corrente. Poiché l'UCS dinamico è abilitato per impostazione predefinita, è possibile creare entità 2D sulla faccia del solido 3D sotto il cursore. Utilizzare lo strumento Allinea UCS per posizionare l'UCS in base alle proprie esigenze.

Gli strumenti di disegno includono primitive 2D familiari, come un rettangolo/poligono, un cerchio/arco/ellisse, una polilinea e una spline.

Utilizzare gli strumenti Sposta, Copia, Ruota 2D, Specchio 2D e Serie per manipolare gli schizzi.

Utilizzare i comandi *OFFSET*, *STIRA*, *TAGLIA*, *ESTENDI*, *RACCORDO/CIMA* e *SPEZZA* per modificare gli schizzi.

Utilizzare i vincoli 2D per rendere gli schizzi parametrici.

Nota: Il valore della variabile di sistema *DELOBJ* (Delete Entity) controlla se le entità dello schizzo vengono eliminate quando vengono utilizzate da *DMEXTRUDE* o *DMREVOLVE* per creare solidi o superfici 3D.

21.3.4 Modellazione Solida

BricsCAD Mechanical fornisce un ricco set di strumenti per la modellazione solida, che include strumenti di modellazione 3D tradizionali come Estrudi o Sottrai, ma anche intuitivi strumenti di Modellazione Diretta

come Premi/Tira, Copia Facce e potenti vincoli 3D che consentono di regolare la forma e le dimensioni del tuo modello 3D.

Gli strumenti per solidi 3D sono disponibili nella scheda **Solido** del nastro dell'area di lavoro **Meccanica**:

Figura 6: Solido

Tabella 1: Modellazione Solida

Strumenti/Gruppo di Strumenti	Descrizione
Estrudi	Crea solidi 3D da schizzi.
Rivoluzione	
Loft	
Sweep	
Ispessisci	
Gruppo di strumenti primitivi: Parallelepipedo, Cilindro, Cono, Piramide, Cuneo, Sfera e Toro	Crea solidi 3D direttamente, senza schizzi sottostanti.

Tabella 1: Modellazione Solida

Strumenti/Gruppo di Strumenti	Descrizione
Premi/Tira	Modifica solidi 3D (applicati a facce e/o spigoli).
Sposta	
Ruota	
Copia Facce	
Raccordo	
Cima	
Svuota	Modifica solidi 3D.
Trancia	
Gruppo di strumenti booleani: Unione, Sottrai, Intersezione e Separa	
Scala	
Specchio	
Serie	
Cancella	Può essere applicato all'intero solido 3D e a un gruppo di facce che formano una caratteristica (ad esempio un raccordo o un foro).
Verifica	Trova e corregge gli errori nella geometria importata.
Semplifica	Riconosce automaticamente le facce canoniche (planari, cilindriche, coniche, sferiche, toroidali) dalle facce spline del tuo solido 3D. Questo è spesso necessario prima di applicare i vincoli 3D.
Cuci	Cuci trasforma superfici e regioni, che formano un contorno a tenuta stagna, in un unico solido 3D.

Tabella 1: Modellazione Solida

Strumenti/Gruppo di Strumenti	Descrizione
Gruppo di strumenti per la deformazione: Torsione, Sposta Punto, Sposta Spigolo, e Trasforma Curva	Deforma il tuo solido 3D in modo uniforme.
Selezione Intelligente	Seleziona facce e spigoli dei tuoi solidi 3D in base a particolari criteri geometrici e topologici, come lunghezza, area e levigatezza.
Parametrizza	Aggiunge vincoli 3D alle facce e agli spigoli dei solidi 3D per allinearli e ridimensionarli. Rende parametrica la tua parte 3D.
Intento del Disegno	Applica automaticamente vincoli 3D geometrici, come parallelo, perpendicolare, coassiale e tangente.

Nota: BricsCAD è un modellatore libero dalla cronologia; non crea dipendenze genitore-figlio tra le caratteristiche, né rigenera la geometria sul cambiamento dei parametri. È possibile incorporare il controllo parametrico completo in qualsiasi parte 3D (incluse quelle importate) utilizzando vincoli 3D.

21.3.5 Modellazione di Superfici

BricsCAD Mechanical fornisce un potente gruppo di strumenti per la modellazione di superfici, la conversione di solidi 3D in superfici e viceversa.

Gli strumenti di modellazione delle superfici sono disponibili nella scheda **Superficie** della barra multifunzione dell'area di lavoro **Meccanico**:

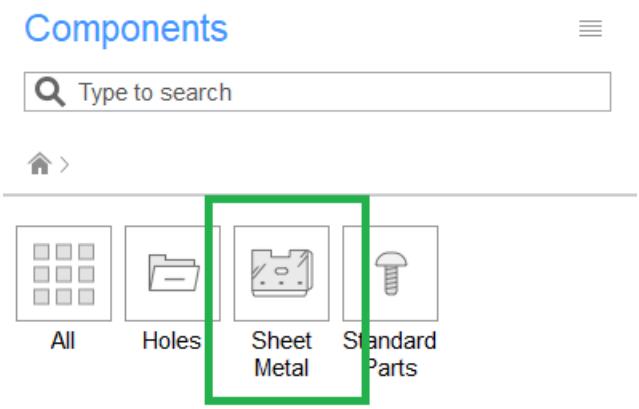


Figura 7: Superficie

La Modellazione delle Superfici e la Modellazione Solida sono strumenti simili che riguardano la modifica e l'applicazione di vincoli 3D. L'unica differenza è che gli strumenti Estrudi/Rivoluzione/Loft/Sweep possono creare entità superficie da profili 2D chiusi e accettare profili non clonati.

Il comando Regione crea un'entità regione da un profilo 2D chiuso e tutte le regioni possono essere cucite insieme per creare una superficie più grande.

Altri metodi per creare una superficie consistono nell'estrarre una o più facce adiacenti di un soldo/superficie 3D e di esplodere un solido 3D in un insieme di superfici e regioni.

21.3.6 Modellazione di Superfici a Forma Libera

21.3.6.1 Panoramica

Aggiungi distorsioni complesse ai tuoi modelli solidi o alle tue superfici utilizzando gli strumenti meccanici BricsCAD. Puoi creare forme uniformi o forme organiche. È possibile deformare i solidi e le superfici spostando, ruotando e torcendo gli spigoli per modificarne la geometria in modo indipendente senza utilizzare la cronologia di progettazione.

L'approccio senza storia ti consente di concentrarti sulla progettazione, invece di pensare a come può essere modificato in futuro.

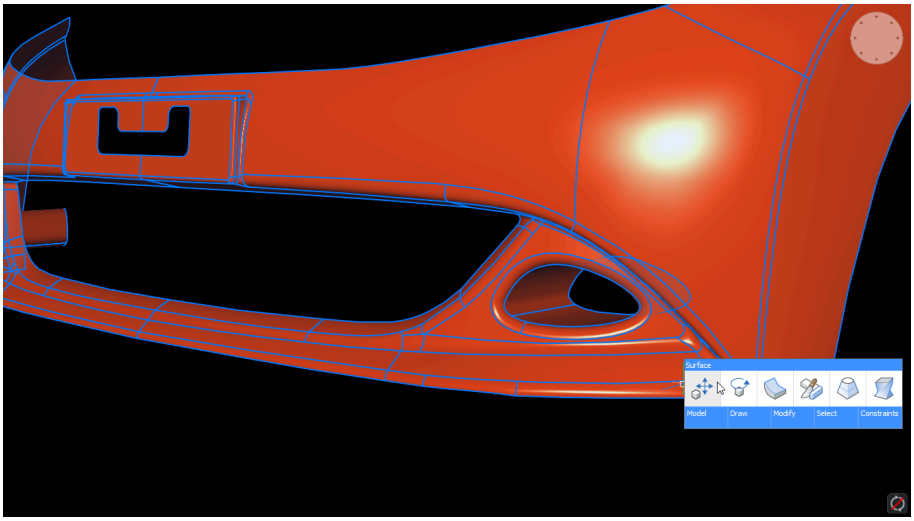


Figura 8: Modellazione di Superfici a Forma Libera

21.3.6.2 Strumenti di deformazione

È possibile utilizzare i seguenti strumenti di deformazione:

Figura 9: Deforma

Tabella 2: Strumenti di deformazione

Strumento	Comando	Descrizione
Sposta Spigolo	*DMDEFORMMOVE*	Deforma facce mediante lo spostamento e rotazione dei loro spigoli.
Sposta Punto	*DMDEFORMPOINT*	Deforma il gruppo di facce selezionato spostando un punto.
Trasforma Curva	*DMDEFORMCURVE*	Deforma le facce cambiando la geometria dei loro spigoli.
Torsione	*DMTWIST*	Deforma l'oggetto 3D ruotando una porzione attorno a un dato asse.

Suggerimento: Guarda il tutorial: utilizza la modellazione di superfici e la modellazione deformabile.

21.4 Progettazione di Assiemi

21.4.1 Progettazione Assiemi Bottom-up e Top-down

21.4.1.1 Panoramica

La modellazione di un assieme rappresenta la struttura di un prodotto progettato ed è costituita dall'utilizzo di componenti per raggruppare le entità geometriche nelle gerarchie. La modellazione dell'assieme viene comunemente utilizzata nelle applicazioni Mechanical Computer-Aided Design (MCAD), semplificando la progettazione di prodotti complessi contenenti migliaia o milioni di entità geometriche.

In questa guida vengono utilizzati i seguenti termini:

- Un assieme è un componente che contiene elementi di altri componenti
- Un sottoassieme è un assieme separato inserito in un altro assieme
- Una parte è un componente che non contiene inserti di un altro componente.

Un assieme con componenti inseriti (che a loro volta possono essere assemblati con altri componenti inseriti) forma un albero chiamato struttura meccanica del modello 3D. Questo albero viene visualizzato nel Navigatore Meccanico.

Esistono 2 modi principali per progettare prodotti meccanici: approcci **top-down** e **bottom-up**. È anche possibile aggiungere una struttura meccanica alla geometria esistente, progettata in BricsCAD o importata da un'altra fonte.

Utilizzare il comando *BMNEW* per creare un nuovo documento per un nuovo componente meccanico.

Utilizzare il comando *BMMECH* per convertire i blocchi esistenti e i riferimenti esterni in componenti. Utilizzare il comando *BMUNMECH* per eseguire l'opposto.

21.4.1.2 Progettazione Bottom-Up (Basso-Alto)

Con l'approccio dal basso verso l'alto, puoi iniziare a progettare un prodotto dettagliando il componente di basso livello. Questi componenti vengono quindi raggruppati in componenti di livello superiore. Il processo continua fino a quando il componente principale non corrisponde al prodotto stesso.

Il comando principale per la progettazione bottom-up (dal basso verso l'alto) è *BMINSERT*. Utilizzare questo comando per inserire un componente esistente nel disegno corrente.

21.4.1.3 Progettazione Top-Down (Alto-Basso)

Con l'approccio top-down (dall'alto verso il basso), puoi iniziare a progettare un prodotto definendo, ma non dettagliandoli, i suoi componenti di primo livello. Ogni componente viene quindi raffinato in modo più dettagliato definendo i suoi sottocomponenti. Il processo continua fino a quando tutti i componenti di basso livello sono definiti e dettagliati.

Utilizzare il comando *BMFORM* per creare un nuovo sottocomponente per un componente esistente.

21.4.1.4 Aggiunta di una Struttura Meccanica alla Geometria Esistente

È possibile aggiungere la gerarchia dei componenti a qualsiasi disegno esistente, creato in BricsCAD o importato da un formato CAD diverso.

Utilizzare il comando *BMFORM* per convertire un gruppo selezionato di entità geometriche in un sottocomponente del disegno corrente.

21.4.1.5 Modificare la Struttura Meccanica

Durante il processo di progettazione, è possibile comprendere che un particolare componente deve essere spostato verso l'alto o verso il basso nella gerarchia dei componenti.

Utilizzare il comando *BMFORM* per formare un nuovo sottocomponente per il disegno corrente che conterrà un gruppo selezionato di un componente.

Utilizzare il comando *BMDISSOLVE* per eliminare un componente e spostare tutti i suoi componenti secondari al livello del componente dissolto.

21.4.1.6 Utilizzo di Componenti Standard

Alcuni componenti standard sono:

- viti
- bulloni
- rondelle
- cuscinetti
- fori.

BricsCAD fornisce un'enorme libreria di componenti standard, con oltre 30.000 dettagli in diverse dimensioni. È possibile aumentare significativamente le prestazioni della progettazione utilizzando parti standard.

Utilizzare il pannello Componenti per inserire componenti standard nelle proprie parti e assiemi.

21.4.1.7 Vincoli di Assemblaggio

I vincoli dell'assieme sono Vincoli 3D ordinari, che collegano facce e spigoli dei corpi per rappresentare i componenti di un assieme. È possibile collegare Vincoli 3D a livello superiore e a livello di sottocomponenti.

21.4.1.8 Visibilità Componente

Un componente all'interno di un assemblaggio complesso potrebbe sembrare invisibile a causa di altri componenti che lo circondano.

Esempio: un pistone è invisibile all'interno di un blocco di cilindro. Per rendere visibile tale componente, potrebbe essere necessario nascondere gli altri componenti circostanti.

E' possibile nascondere qualsiasi sottocomponente inserito nel disegno corrente o qualsiasi suo sottocomponente su qualsiasi livello di nidificazione.

C'è anche la possibilità di rappresentare un componente in trasparenza per visualizzarlo in wireframe.

Utilizzare il comando *BMHIDE* per nascondere un particolare componente secondario.

Utilizzate il comando *BMSHOW* per rendere visibile un sottocomponente nascosto.

21.4.2 Comandi e Strumenti di Progettazione dell'Assieme

21.4.2.1 Comandi

Tabella 3: Comandi Progettazione Assieme

Icona	Comando	Descrizione
	BMNEW	Crea un nuovo componente meccanico come un nuovo file di disegno.

Tabella 3: Comandi Progettazione Assieme

Icona	Comando	Descrizione
	BMMECH	Converte il disegno corrente in un componente meccanico principale.
	BMINSERT	Inserisce un componente meccanico esistente nel disegno corrente.
	BMINSERT	Inserisce un componente meccanico esistente nel disegno corrente con ridimensionamento automatico della nuova parte e lo connette alla parte associata.
	BMHARDWARE	Apre il Pannello Componenti con i Componenti Meccanici Standard
	BMFORM	Crea un nuovo componente meccanico e lo inserisce nel disegno corrente.
	BMCONNECT	Collega due parti standard di Tubazioni creando vincoli 3D tra le loro entità di connessione.
	BMREPLACE	Sostituisce un elemento di componente.
	BMLOCALIZE	Commuta componenti esterni in componenti locali.
	BMEXTERNALIZE	Commuta componenti locali in componenti esterni.
	BMOPEN	Apre il disegno di origine di un componente meccanico esterno.
	BMOPENCOPY	Verrà aperta una copia di un componente come nuovo disegno.
	BMDISSOLVE	Dissolve un componente meccanico inserito nel disegno corrente.
	BMLINK	Modifica i solidi 3D di destinazione delle caratteristiche basate sui componenti.
	BMUNLINK	Interrompe la connessione tra un componente (ad esempio una finestra) ed un solido 3D (ad esempio un muro) mantenendo l'apertura.

Tabella 3: Comandi Progettazione Assieme

Icona	Comando	Descrizione
	BMHIDE	Nasconde un componente meccanico o un sottocomponente nel disegno corrente.
	BMSHOW	Mostra un componente o sottocomponente meccanico precedentemente nascosto nel disegno corrente.
	BMVSTYLE	Applica uno stile di visualizzazione a un componente meccanico.
	BMBALLOON	Crea entità richiamo associativi per i componenti dell'assieme nello Spazio Modello e per le viste generate degli stessi nei layout dello Spazio Carta.
	BMBALLOON	Consente il posizionamento automatico di richiami associativi su tutti i componenti in una particolare vista di disegno in un layout dello Spazio Carta.
	BMBOM	Inserisce la tabella Distinta dei Materiali (BOM) nel disegno corrente.
	BMMASSPROP	Calcola le proprietà di massa del modello corrente utilizzando le densità assegnate ai suoi componenti.
	BMUPDATE	Ricarica tutti i componenti referenziati da file esterni ed aggiorna le tabelle BOM.
	BMUPDATE	Ricarica tutti i componenti referenziati selezionati da file esterni ed aggiorna le tabelle BOM.
	BMEXPLODE	Crea una rappresentazione esplosa di un assieme spostando i componenti per renderli tutti visibili, lo memorizza in un nuovo blocco ed inserisce il blocco nello Spazio Modello.
	BMEXPLODEMOVE	Sposta le parti selezionate per formare una rappresentazione esplosa in una determinata direzione, tenendo conto delle possibili collisioni fisiche tra i componenti (simili all'opzione Lineare del comando *BMEXPLODE*).
	BMTRAILINGLINES	Crea tutte le linee guida necessarie per le parti selezionate.

Tabella 3: Comandi Progettazione Assieme

Icona	Comando	Descrizione
	MECHANICALBROWSEROPEN	Visualizza il Navigatore Meccanico.
	PARAMETERSPANELOPEN	Visualizza il Gestore Parametri.
	BMDEPENDENCIES	Elenca tutti i file nella finestra dei comandi, comprendendo le definizioni dei componenti inseriti nell'assieme.
	BMRECOVER	Recupera una struttura meccanica rotta.
	BMUNMECH	Converte l'attuale componente meccanico in un disegno semplice.
	BMXCONVERT	Converte solidi X-Hardware nel disegno corrente in componenti meccanici.

21.4.2.2 Strumenti

È possibile utilizzare i seguenti strumenti per lavorare con i componenti meccanici:

1 Nella barra degli strumenti **Assemblaggio**:

- **Assemblaggio**

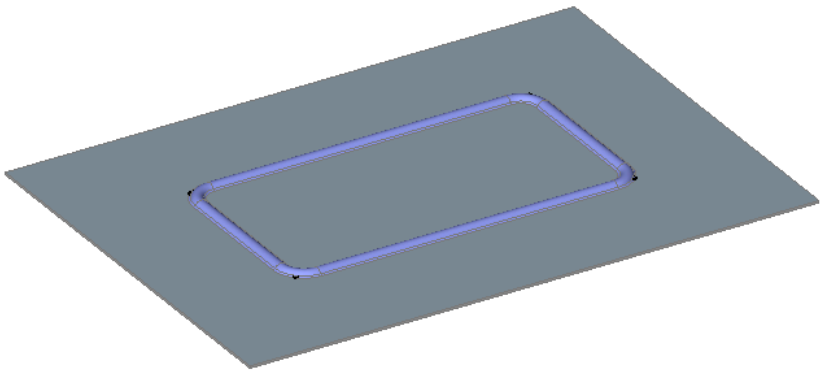


Figura 10: Assemblaggio

- **Assemblaggio - Elementi**

Figura 11: Assemblaggio - Elementi

- **Assemblaggio - Esplodi**

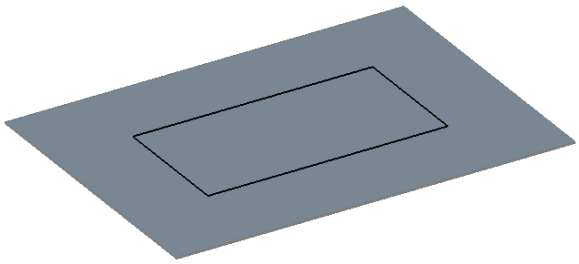


Figura 12: Assemblaggio . Esplodi

- **Assemblaggio - Visualizzazione**

Figura 13: Assemblaggio - Visualizzazione

2 Nel menu **Assieme** (solo Area di lavoro Meccanico):

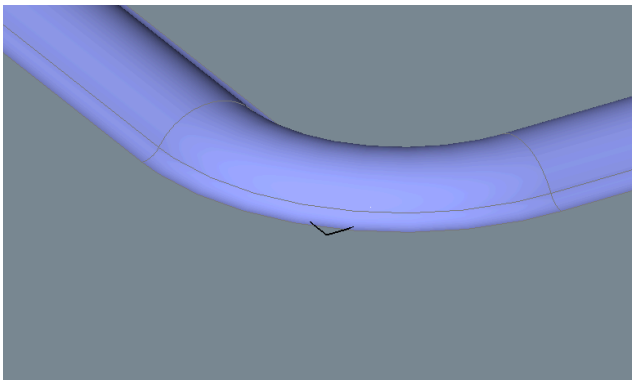


Figura 14: Menu Assieme

3 Nella scheda **Assemblaggio** della barra multifunzione dell'Area di lavoro Meccanico:

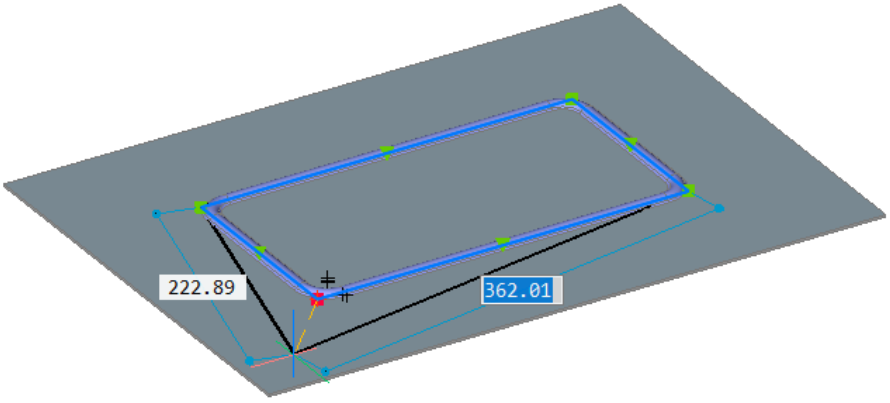


Figura 15: Assemblaggio

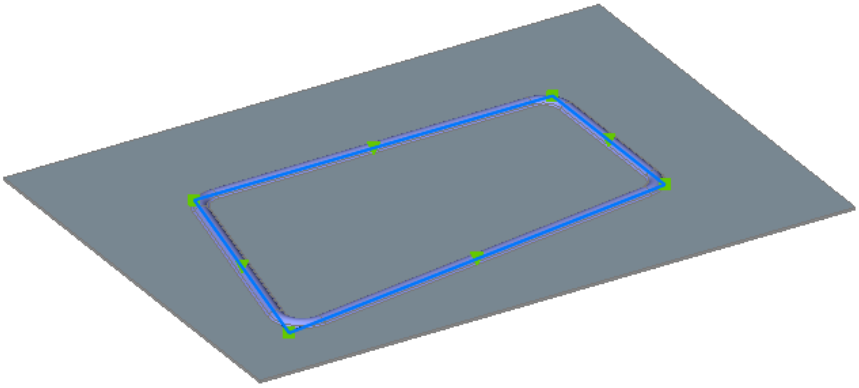


Figura 16: Strumenti

21.4.3 Proprietà dei Componenti

Le proprietà dei componenti e i menu contestuali si trovano nel pannello **Navigatore Meccanico**.

I menu contestuali dei componenti vengono visualizzati quando si clicca il pulsante destro del mouse sul componente. Le proprietà dei componenti vengono visualizzate automaticamente quando si selezionano uno o più componenti nel pannello Navigatore Meccanico.

Figura 17: Navigatore Meccanico

I componenti possono essere: **Principali**, **Elementi** e **Sottocomponenti**.

Tabella 4: Componenti

Tipo di componente	Stile di visualizzazione Renderizzato	Stile di visualizzazione Wireframe	Nascosto
Componente senza sottocomponenti	*	*	*
Componente con sottocomponenti	*	*	*
Componente parte standard			

* I componenti esterni contengono una freccia verde: o .

Tabella 5: Proprietà dei Componenti

Menù contestuale	
Menu contestuale del Componente Principale	Cliccare il pulsante destro del mouse sul nome del componente principale: **Figura 18:** Menu contestuale del Componente Principale • **Aggiorna**: aggiorna la gerarchia dei componenti meccanici per il disegno corrente nel caso in cui i file di disegno referenziati dei sottocomponenti siano stati modificati. • **Stile di Visualizzazione**: **Tutto da Finestra**: applica l'attuale **Stile di Visualizzazione** a tutti i componenti dell'assieme (vedere il comando *SHADEMODE*). • **Inserisci hardware standard**: inserisce una parte hardware standard come componente meccanico nel disegno corrente. • **Commuta tutto in locale**: commuta tutti i componenti in componenti interni (vedere il comando *BMLOCALIZE*). • **Commuta tutto in esterno**: commuta tutti i componenti in componenti esterni (vedere il comando *BMEXTERNALIZE*). • **Aggiungi nuovo parametro**: crea un nuovo parametro nell'assieme. • **Seleziona tutto**: seleziona tutti i componenti con la stessa definizione. • **Seleziona uguale**: seleziona tutti i componenti con lo stesso nome e gli stessi valori dei parametri. • **Evidenzia tutto**: evidenzia tutti i componenti con la stessa definizione. • **Evidenzia uguale**: evidenzia tutti i componenti con lo stesso nome e gli stessi valori dei parametri. • **Crea vista esplosa**: crea un blocco con una rappresentazione esplosa dell'assieme corrente. • **Comprimi tutto**: Comprime il componente principale e tutti i componenti e sottocomponenti. • **Espandi tutto**: Espande il componente principale e tutti i componenti e sottocomponenti.

Tabella 5: Proprietà dei Componenti

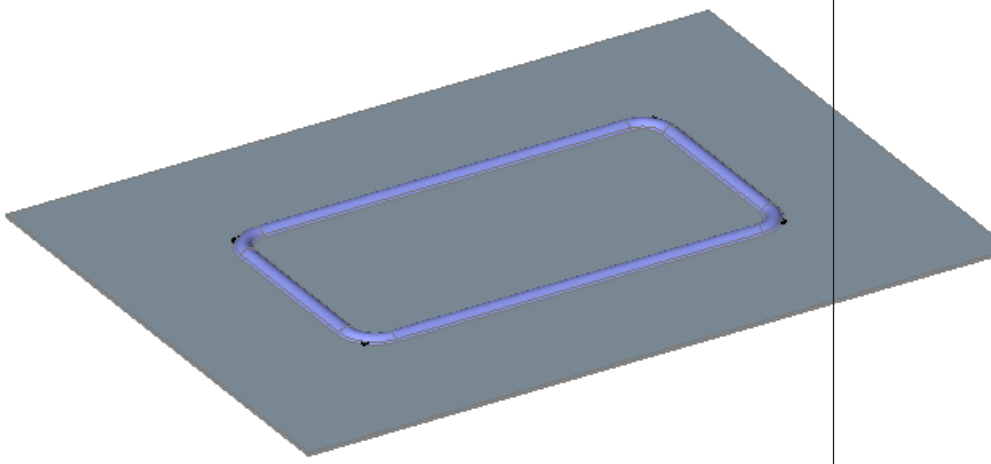
Menù contestuale	
Menu contestuale del Componente	Cliccare il pulsante destro del mouse sul nome di un componente Menu contestuale dei componenti esterni (a sinistra), menu contestuale dei componenti locali (a destra):  • **Apri**: apre il disegno del riferimento (vedere il comando *BMOPEN*). • **Apri una copia**: apre una copia di un componente come nuovo disegno (vedere il comando *BMOPENCOPY*). • **Aggiorna**: ricarica tutti i componenti referenziati da file esterni e aggiorna le tabelle BOM (vedere il comando *BMUPDATE*). • **Sostituisci**: sostituisce l'elemento di un componente (vedere il comando *BMREPLACE*). La sostituzione di un elemento locale lo trasforma in un elemento esterno. • **Sostituisci tutti gli elementi**: sostituisce tutti gli elementi che fanno riferimento alla stessa fonte (vedere il comando *BMREPLACE*). Sostituendo un elemento locale lo si trasforma in un elemento esterno. • **Commuta in locale**: commuta un componente esterno in un componente interno (vedere il comando *BMLOCALIZE*). • **Commuta in esterno**: commuta un componente interno in un componente esterno (vedi il comando *BMEXTERNALIZE*). • **Imposta il materiale al componente**: assegna un materiale fisico al componente locale. • **Rimuovi materiale dal componente**: rimuove la definizione del materiale fisico dal componente locale. • **Stato BOM**: parametro per controllare l'aspetto del componente nelle tabelle BOM. • **Dissolvi**: dissolve un componente meccanico inserito nel disegno corrente (vedere il comando *BMDISSOLVE*). • **Scollega**: interrompe la connessione tra un elemento componente (ad esempio una finestra) e un solido 3D (ad esempio un Muro) mantenendo l'apertura. • **Nascondi** ( ) / **Mostra** ( ): nasconde o mostra il componente selezionato. • **Escludi tutti gli elementi dalla sezione**: imposta la proprietà **Sezionabile** di tutti gli elementi simili su *NO*. Definisce se un elemento è influenzato dal comando *VISTASEZIONE*. • **Stile di Visualizzazione**: mostra il menu **Stile di Visualizzazione**. Sono disponibili tutti gli Stili do Visualizzazione salvati nel disegno corrente. - **Da componente principale**: consente di eseguire il rendering del componente selezionato in base allo shademode del componente

Tabella 5: Proprietà dei Componenti

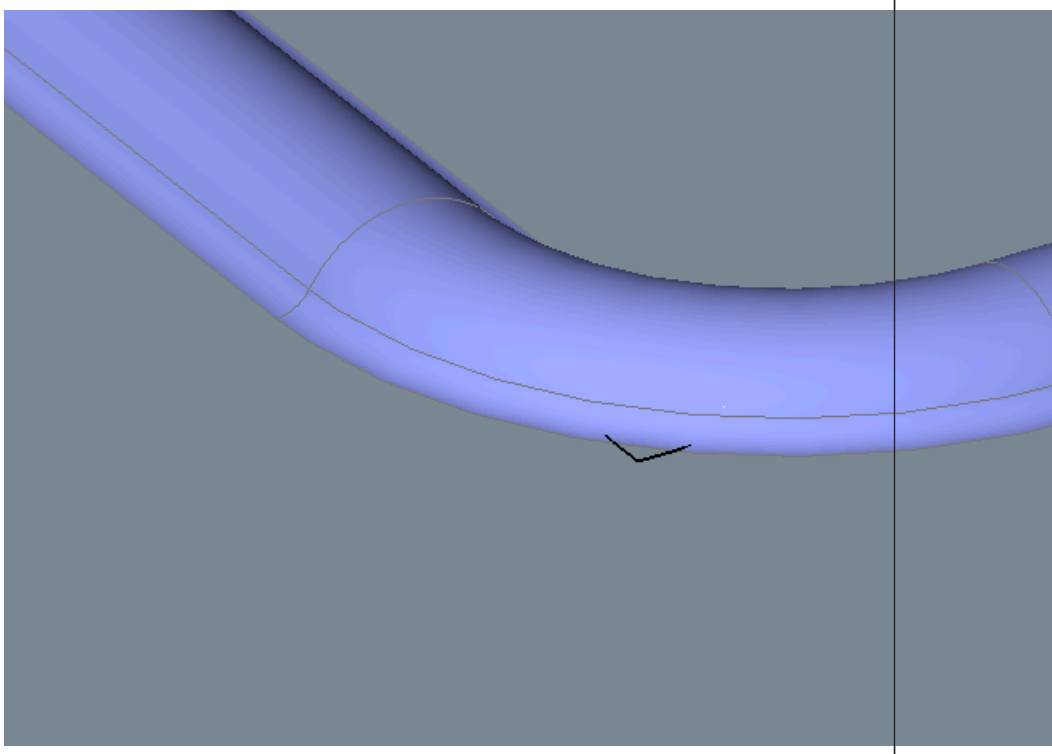
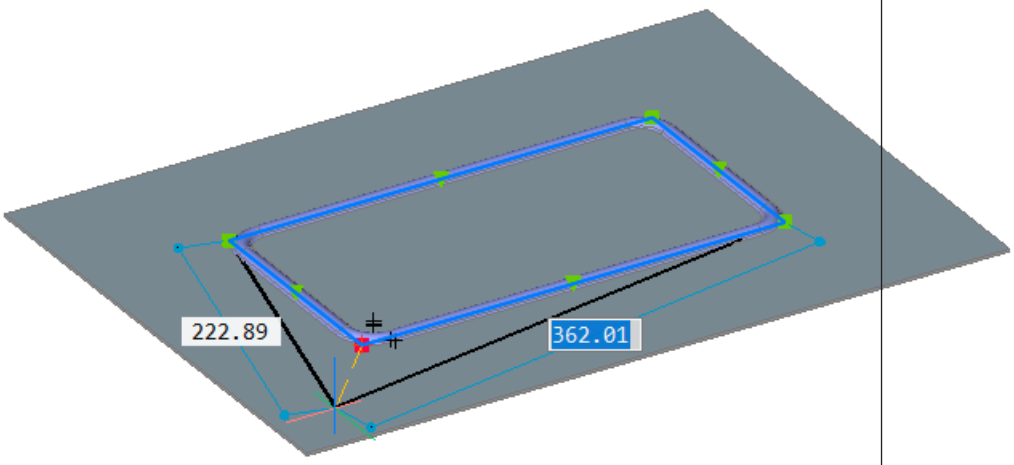
Menù contestuale	
Menu contestuale dei sottocomponenti	Menu contestuale dei sottocomponenti esterni (a sinistra), menu contestuale dei sottocomponenti locali (a destra):   • **Apri**: apre il disegno di riferimento (vedere il comando *BMOPEN*). • **Apri una copia**: apre una copia di un componente come nuovo disegno (vedere il comando *BMOPENCOPY*). • **Nascondi** ( ) / **Mostra** ( ): nasconde o mostra il sottocomponente selezionato. • **Escludi tutti gli elementi dalla sezione**: imposta la proprietà **Sezionabile** di tutti gli elementi simili su *NO*. Definisce se un elemento è interessato dal comando *VISTASEZIONE*. • **Stile di Visualizzazione**: vedere Stile di Visualizzazione Componente. • **Zoom a**: ingrandisce il gruppo di selezione • **Seleziona tutto**: seleziona tutti i componenti con la stessa definizione. • **Seleziona uguale**: seleziona tutti i componenti con la stessa definizione e gli stessi valori dei parametri. • **Evidenzia tutto**: evidenzia tutti i componenti con la stessa definizione. • **Evidenziare uguale**: evidenzia tutti i componenti con la stessa

Tabella 5: Proprietà dei Componenti

Menù contestuale	
Menu contestuale dei parametri del componente	Cliccare il pulsante destro del mouse su un parametro: • **Link al parametro**: stabilisce un collegamento tra uno o più parametri del componente e i parametri dell'assieme. • **Comprimi tutto**: Comprime il componente principale e tutti i componenti e sottocomponenti. • **Espandi tutto**: Espande il componente principale e tutti i componenti e sottocomponenti.
Proprietà	

Tabella 5: Proprietà dei Componenti

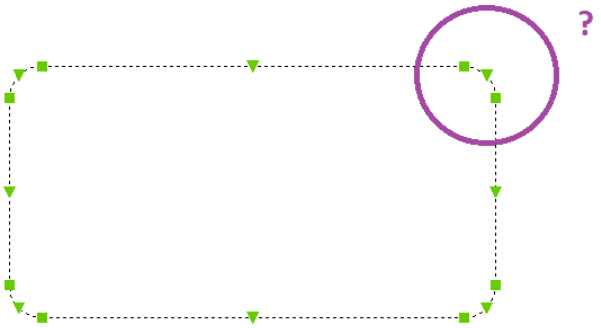
Menù contestuale	
Proprietà dei componenti principali	Cliccare sul nome del componente principale:  **Figura 19:** Proprietà dei componenti principali • **Nome**: digitare un nome per il componente principale. Per impostazione predefinita, il nome del componente principale è uguale al nome del file. • **Descrizione**: descrizione facoltativa del componente principale. • **File**: mostra il percorso e il nome del disegno. • **Tipo estensione**: consente di identificare il componente come componente BIM o Caratteristica di Lamiera. • **Inserisci come**: selezionare **Componente Locale** o **Componente Esterno**. • **Sezionabile**: definisce se il componente è influenzato dal comando *VISTASEZIONE*. • **Componente standard**: definisce se questo componente è standard o meno. • **Stato BOM**: definisce il comportamento di questo componente in una tabella BOM. • **Materiale**: materiale del componente principale. Il **Nome**, la **Densità** e la **Descrizione** opzionale sono definiti nella finestra di dialogo Materiali Fisici. Selezionare il nodo **Materiale** ed eseguire una delle operazioni seguenti: - Cliccare sul pulsante **Sfoglia** (...) per aprire la finestra di dialogo Materiali Fisici e scegliere un materiale nell'elenco Progetto o Libreria. La proprietà Densità del materiale selezionato viene utilizzata dal comando *BMMASSPROP*. - Cliccare sul pulsante **Elimina** (x). Il campo **Materiale** ora riporta<*Eredita*>. Quando il componente viene inserito in un assieme, la definizione del **Materiale** viene copiata dal componente principale dell'assieme.

Tabella 5: Proprietà dei Componenti

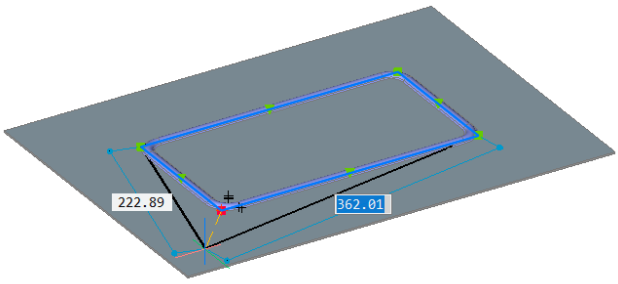
Menù contestuale	
Proprietà del componente elemento	Fare clic sul nome del sottocomponente:  **Figura 20:** Proprietà del componente elemento • **Nome**: digitare un nome per il componente. Per impostazione predefinita, il nome del componente è uguale al nome del file di origine. • **Nome componente**: nome del componente come definito nel file di origine. • **Descrizione**: descrizione del componente come definito nel file di origine. • **Visibile**: mostra o nasconde il componente nell'assieme. Scegliere tra: - Cliccare sul campo **Visibile**, fare clic sul pulsante freccia giù e scegliere *Sì* o *No*. - Fare doppio clic sul campo **Visibile** per disattivare la visibilità del componente. • **Sezionabile**: definisce se il componente è influenzato dal comando *VISTASEZIONE*. • **Componente standard**: definisce se questo componente è standard o meno. • **Stato BOM**: definisce il comportamento di questo componente in una tabella BOM. • **Stile di Visualizzazione**: consente di controllare lo stile di visualizzazione del componente. Come impostazione predefinita lo stile di visualizzazione viene ereditato dalla finestra. Fare clic sul campo **Stile di Visualizzazione**, cliccare sul pulsante freccia giù e scegliere uno stile di visualizzazione nell'elenco. • **File**: mostra il percorso e il nome del disegno di riferimento. • **Materiale**: mostra il materiale del componente. Se impostato su *<Eredita>*, il materiale viene copiato dal componente principale. Se il materiale è definito esplicitamente nel file di origine, viene utilizzato quest'ultimo.

Tabella 5: Proprietà dei Componenti

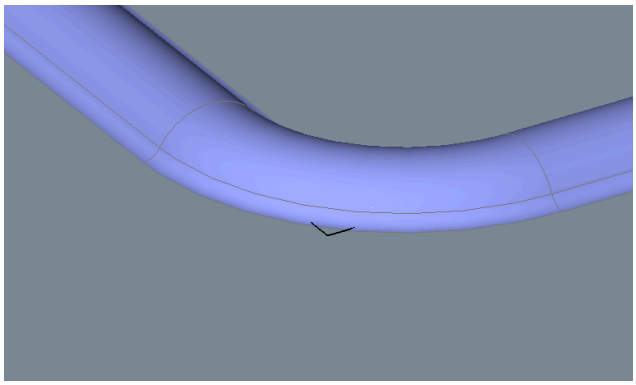
Menù contestuale	
Proprietà sottocomponente	Cliccare sul nome del sottocomponente:  **Figura 21:** Proprietà sottocomponente • **Nome**: nome del componente come definito nel file di origine del componente principale. • **Nome componente**: nome del componente come definito nel file di origine. • **Descrizione**: descrizione del componente come definito nel file di origine. • **Visibile**: mostra o nasconde il componente nell'assieme. Scegliere tra: - Cliccare sul campo **Visibile**, fare clic sul pulsante freccia giù e scegliere *Sì* o *No*. - Fare doppio clic sul campo **Visibile** per disattivare la visibilità del componente. • **Sezionabile**: definisce se il componente è influenzato dal comando *VISTASEZIONE*. • **Componente standard**: definisce se questo componente è standard o meno. • **Stato BOM**: definisce il comportamento di questo componente in una tabella BOM. • **Stile di Visualizzazione**: controlla lo stile di visualizzazione del componente. Come impostazione predefinita lo stile di visualizzazione viene ereditato dalla finestra. Fare clic sul campo **Stile di Visualizzazione**, cliccare sul pulsante freccia giù e scegliere uno stile di visualizzazione nell'elenco. • **File**: mostra il percorso e il nome del disegno di riferimento. • **Materiale**: mostra il materiale del componente secondario. Se impostato su *<Eredita>*, il materiale viene copiato dal componente principale. Se il materiale è definito esplicitamente nel file di origine, viene utilizzato quest'ultimo.

Tabella 5: Proprietà dei Componenti

Menù contestuale	
Metodi di Selezione	
Selezionare il/i componente/i nel modello	I componenti vengono evidenziati nel Navigatore Meccanico.
Cliccare su un componente nel pannello Navigatore Meccanico.	Il componente si evidenzia il disegno. Premere la barra spaziatrice per selezionare il componente.

21.4.4 Proprietà di massa degli assiemi

21.4.4.1 Panoramica

Utilizzare il comando *BMMASSPROP*.

Un assieme può essere costituito da più parti realizzate con diversi materiali. Per calcolare correttamente la massa, il centro di gravità (baricentro) e l'inerzia di tali assiemi, BricsCAD supporta densità individuali per i componenti.

Un materiale fisico per un componente meccanico è definito nella proprietà Materiale nel **Navigatore Meccanico**.

Per impostazione predefinita, un componente eredita la densità dell'Assieme, in cui è stato inserito o formato da.

21.4.4.2 Impostare le unità di massa

1 Aprire la finestra di dialogo *Impostazioni*.

2 Espandere le impostazioni **Proprietà unità** sotto le unità **Disegno / Disegnare/ Unità di Disegno**.

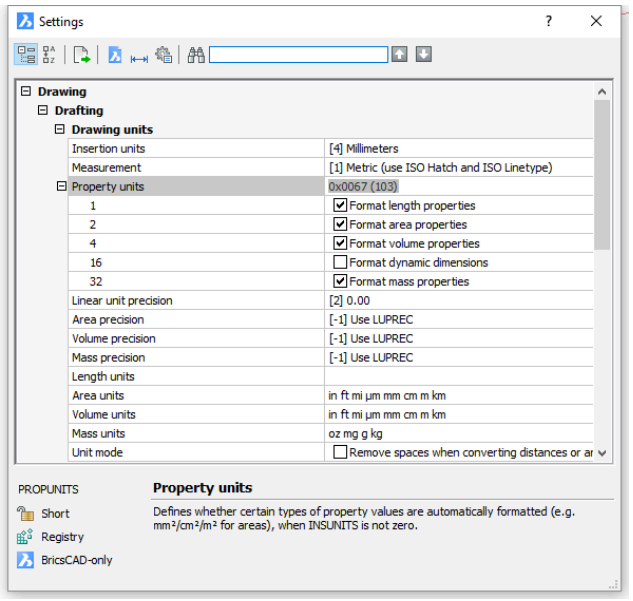


Figura 22: Impostazioni: Proprietà Unità

3 Selezionare la casella di controllo **Formato proprietà di massa** per abilitare la formattazione dei valori di massa con le unità appropriate.

4 Selezionare il campo **Unità di massa**.

5 Fare clic sul pulsante **Sfoglia** ...

6 Specificare le unità di massa desiderate nella finestra di dialogo *MASSUNITS*.

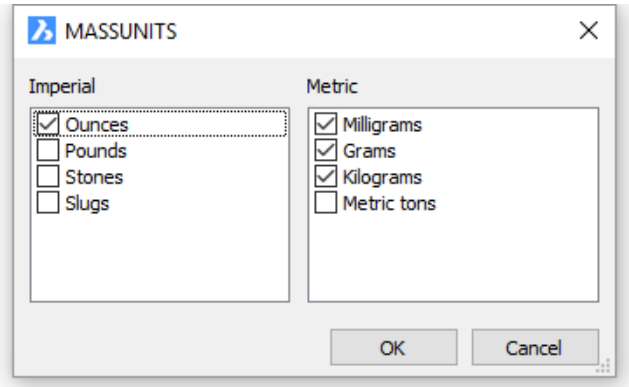


Figura 23: MASSUNITS

Nota: L'impostazione delle **Unità di Massa** influisce solo sui valori di massa. Altre proprietà di massa come densità o momenti di inerzia verranno formattate in unità SI per il sistema metrico e in unità imperiali per il sistema imperiale, indipendentemente dal valore delle **Unità di Massa** impostate.

7 Specificare la precisione desiderata nel campo **Precisione di massa**.

8 Chiudere la finestra di dialogo *Impostazioni*.

21.4.4.3 Calcola le proprietà di massa

Utilizzare il comando *BMMASSPROP*.

È possibile calcolare le proprietà di massa per l'intero modello o per una selezione di solidi 3D e/o sottocomponenti. Nella barra dei comandi sono riportati i valori di massa, volume, baricentro, momenti di inerzia, prodotti di inerzia, raggi di rotazione, momenti principali e direzioni X-Y-Z rispetto al baricentro.

Facoltativamente è possibile allineare il sistema di Coordinate Utente con gli assi di inerzia. Di conseguenza, il centro di massa o il baricentro del componente si troverà all'origine dell'UCS, il che può essere utile per studiare il comportamento di un prodotto sotto la forza di gravità.

21.4.5 Componenti meccanici standard

21.4.5.1 Panoramica

Utilizzare il comando *COMPONENTSPANELOPEN*.

BricsCAD viene fornito con una libreria di oltre 1.000 parti meccaniche standard disponibili in più di 170.000 dimensioni diverse. Sono disponibili nel pannello Libreriae possono essere trascinate dalla libreria all'interno dell'assieme. Potete regolare i parametri nel pannello Proprietà, che elenca temporaneamente i parametri del componente inserito. Dopo l'inserimento, i parametri possono essere modificati nel Navigatore Meccanico e nel pannello Proprietà.

Nello spazio di lavoro meccanico, il riquadro Libreria contiene 3 categorie:

- Parti Standard
- Fori
- Lamiera

21.4.5.2 Parti standard (Hardware)

La categoria Parti standard contiene hardware standard parametrico raggruppato in 4 sottocategorie:

- Dispositivi di fissaggio - bulloni, dadi, rondelle, ecc.
- Parti meccaniche - cuscinetti, pignoni, e così via.
- Tubazioni - tubi, gomiti, flange, ecc.
- Forme strutturali - travi, canali, barre, ecc.

Il contenuto di ciascuna sottocategoria è ulteriormente strutturato per tipo, sottotipo e standard

Esempio: Parti standard/ Dispositivi di fissaggio/Bulloni/Bulloni a testa esagonale/ISO/ISO 4017.

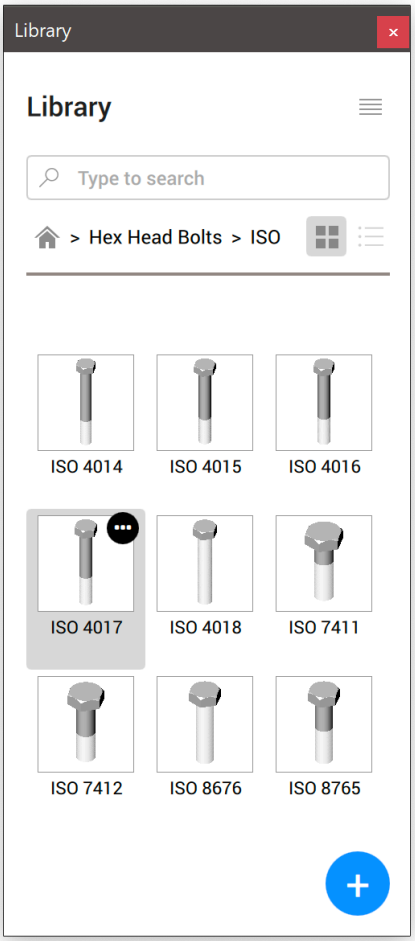


Figura 24: Libreria

Nota: Nella finestra di dialogo *Impostazioni* sono disponibili 2 impostazioni specifiche per le parti standard:

- **Rappresentazione del filetto (THREADDISPLAY)** viene utilizzato per rappresentare o meno una filettatura 3D per bulloni e dadi. Questa impostazione consente di ridurre il consumo di memoria e aumentare le prestazioni per gruppi di grandi dimensioni.
- **Il numero massimo di denti del pignone (GEARTEETHNUMBER)** viene utilizzata per limitare il numero di denti per le ruote pignone (disponibili in **Parti standard / SPROCKET)**. Questa impostazione consente di ridurre il consumo di memoria e aumentare le prestazioni.

Per impostazione predefinita, queste impostazioni sono impostate su *Off* per ottimizzare le prestazioni del modello. Quando vengono modificate, le impostazioni si applicano a tutti i nuovi documenti.

Rappresentazione standard della filettatura nelle visualizzazioni di disegno

Le viste di disegno degli assiemi generate con il comando *VISTABASE* contengono annotazioni di filettatura per le parti standard così come linee sottili per le viste laterali e un arco sottile di 270 gradi per le viste superiore/inferiore.

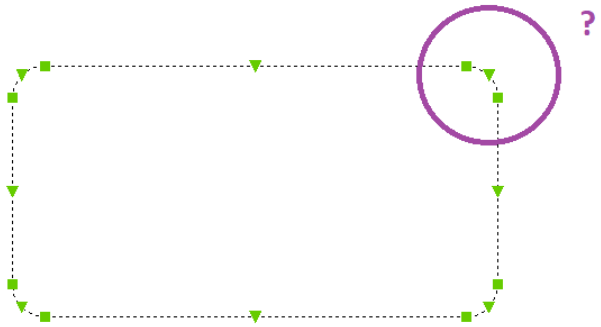


Figura 25: Esempi

21.4.5.3 Fori

La categoria Fori contiene una libreria parametrica di fori standard come scanalature, fresature, svasature ecc.

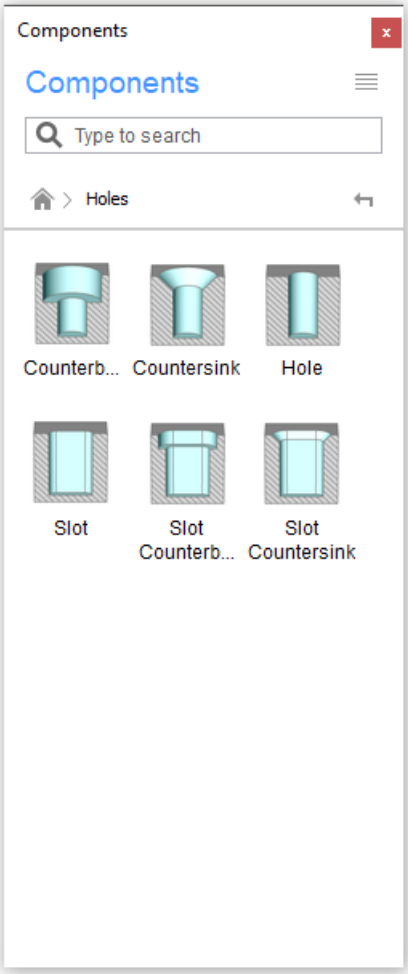


Figura 26: Fori

Esiste anche una sottocategoria di fori filettati, rappresentati correttamente nelle viste di disegno.

21.4.5.4 Caratteristiche di Forma della Lamiera

La categoria Lamiera contiene una libreria parametrica di caratteristiche di forma della lamiera come ponti, nervature.

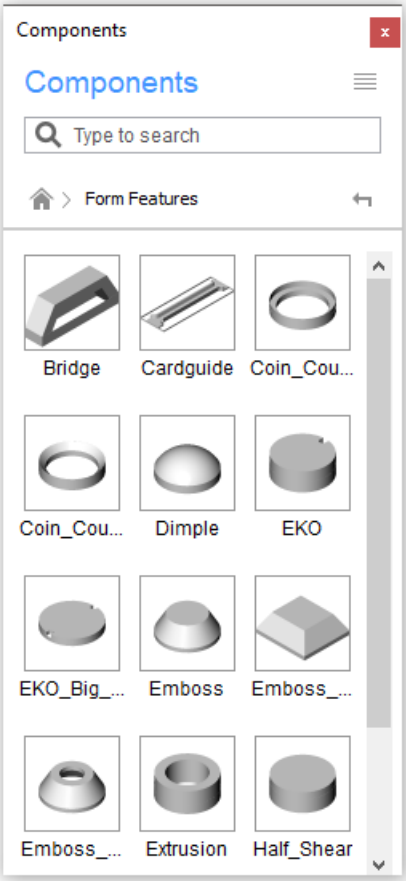


Figura 27: Caratteristica Forma

21.4.5.5 Posizionare un componente standard nel disegno corrente

1 Scegliere una categoria.

2 Scegliere uno standard.

3 Selezionare un tipo.

4 Fare clic sul componente standard desiderato.

5 Modificate i parametri nel pannello delle proprietà.

6 Fare clic nell'area di disegno per inserire il componente selezionato.

21.4.5.6 Creare il tuo componente standard

Utilizzare il comando *BMCREATECOMPONENT* per creare le proprie parti standard e aggiungerle al pannello Componenti.

21.4.5.7 Controllore la visualizzazione dei filetti

1 Aprire la finestra di dialogo *Impostazioni*.

2 Trovare la variabile *Rappresentazione del filetto* (**THREADDISPLAY**).

3 Attivare l'impostazione.

4 Se applicabile, aggiorna il documento per far sì che i filetti appaiano sulle parti esistenti:
- Imposta temporaneamente la **modalità di aggiornamento dei componenti di Assieme (BMUPDATEMODE)** su **[1] Aggiorna tutti i componenti**.
- Utilizzare il comando *BMUPDATE* per aggiornare il documento.
- Se applicabile, riportare la variabile **Modalità di aggiornamento dei componenti dell'assieme (BMUPDATEMODE)** al valore precedente.

21.4.5.8 Controllare il numero di denti del pignone

1 Aprire la finestra di dialogo *Impostazioni*.

2 Trovare la variabile *Numero massimo di denti del pignone* (**GEARTEETHNUMBER**).

3 Impostare il valore in modo che sia sufficiente per i pignoni utilizzati (si consiglia *500*).

Nota: Valori maggiori rallentano l'aggiornamento dei pignoni standard. Se il valore è inferiore al numero richiesto, viene visualizzato solo quel numero di denti.

4 Se applicabile, aggiornare il documento per far apparire i denti del pignone sulle parti esistenti:
- Imposta temporaneamente la **modalità di aggiornamento dei componenti di Assieme (BMUPDATEMODE)** su **[1] Aggiorna tutti i componenti**.
- Utilizzare il comando *BMUPDATE* per aggiornare il documento.
- Se applicabile, riportare la variabile **Modalità di aggiornamento dei componenti dell'assieme (BMUPDATEMODE)** al valore precedente.

21.4.6 Tubazioni nella Progettazione di Assiemi

21.4.6.1 Panoramica

Utilizzare i comandi: *BMINSERT*, *BMCONNECT* e *BMBOM*.

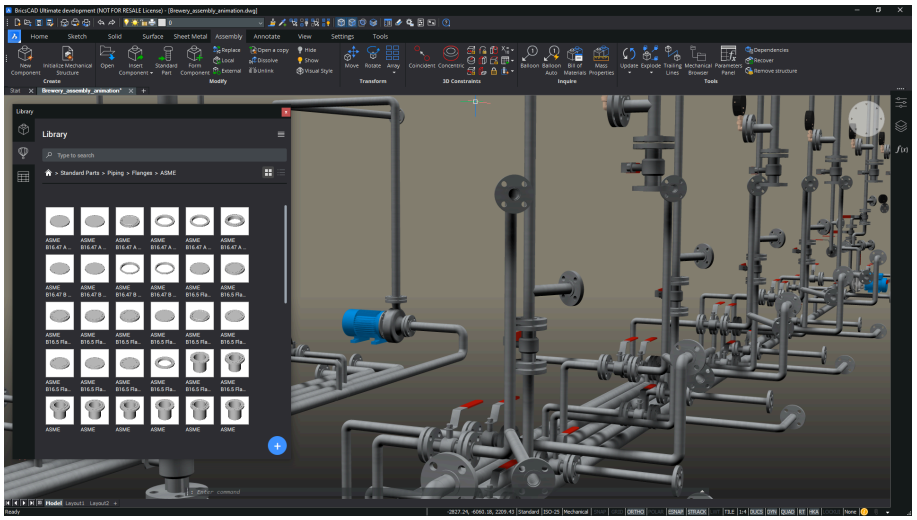


Figura 28: Tubazioni nella Progettazione di Assiemi

I tubi saldati e senza saldatura, i gomiti, i raccordi a T, raccordi a croce, i riduttori e le flange sono ampiamente utilizzati nell'industria meccanica, automobilistica e aeronautica. Si trovano anche nell'industria del petrolio e gas, nell'industria chimica e nell'approvvigionamento idrico. BricsCAD dispone di una libreria di parti di tubazioni parametriche di diversi standard, come ASME, DIN e ISO. Queste parti standard sono parametriche. È possibile regolare facilmente i parametri, collegarli a diversi parametri globali e utilizzare vincoli 3D per collegare correttamente le parti. BricsCAD consente di semplificare queste operazioni ripetitive utilizzando la **modalità Intelligente** del comando *BMINSERT* e un comando dedicato *BMCONNECT*.

21.4.6.2 Inserire parti di tubazione nel tuo assieme

Trascinare e rilasciare qualsiasi parte standard di tubazioni dal pannello **Libreria** all'interno dell'assieme.

Per impostazione predefinita, tutte le parti della categoria Tubazioni vengono inserite in modalità Intelligente, il che significa che se si passa con il mouse su una parte di tubazioni precedentemente inserita, BricsCAD:

- Copia le espressioni dei parametri di quella parte nei parametri corrispondenti della parte inserita.
- Aggiunge un gruppo di vincoli 3D per collegare le parti.

Nota: Premere il tasto **Ctrl** per passare dalla modalità Intelligente alla modalità Regulare, nella quale non vengono generati vincoli 3D con altre parti.

21.4.6.3 Creare manualmente vincoli 3D tra le parti standard di tubazioni

Le parti standard di tubazioni hanno un gruppo di entità wireframe (punti e linee) sul layer STD_PARTS_PIPING_CONNECTION, spento per impostazione predefinita.

Vincolare un raccordo

Per vincolare un raccordo (gomito, riduttore, flangia, ...) con un altro raccordo o tubo, creare 2 vincoli 3D coincidenti tra:

- 2 punti dei raccordi, che dovrebbero coincidere

- 2 linee di raccordi, che dovrebbero essere collineari.

Vincolareuna tubazione

Vincolare le tubazioni è più complesso perché un tubo può avere qualsiasi lunghezza. Se un tubo è già vincolato con un raccordo su un lato utilizzando 2 vincoli 3D coincidenti, il vincolo sull'altro lato deve essere diverso.

Devi creare:

- Un vincolo 3D coincidente tra la linea, che rappresenta l'asse del tubo e la linea corrispondente dell'altro raccordo
- Un vincolo di Distanza 3D, guidato dalla geometria, tra la faccia planare del tubo sul lato opposto (già vincolato) e il punto corrispondente del raccordo, che viene collegato.

È quindi necessario digitare il nome del vincolo di distanza nel campo espressione per il parametro Length (Lunghezza) del tubo. Poiché i vincoli basati sulla geometria consentono il controllo di una particolare dimensione, il tubo avrà sempre la lunghezza corretta e tale parametrizzazione non limiterà alcuna ulteriore modifica dell'assieme utilizzando la manipolazione diretta di vincoli dimensionali aggiuntivi non guidati dalla geometria.

21.4.6.4 Collegare automaticamente le parti di tubazioni

Il gruppo di vincoli viene creato automaticamente utilizzando la modalità Intelligente del comando *BMINSERT* o il comando dedicato *BMCONNECT* per collegare 2 parti standard.

21.4.6.5 Creare una Distinta dei Materiali per un assieme di tubazioni

In una distinta base è possibile che tutti i tubi dello stesso tipo e dimensione (PartNumber), ma con lunghezze diverse, vengano elencati come se fossero la stessa parte.

Utilizzare la funzione di Aggregazione del Gestiore BOM per combinare tutte queste parti in una singola riga con la loro lunghezza totale.

21.4.7 Serie di Componenti

21.4.7.1 Panoramica

Usare i comandi *SERIE*, *SERIERETTANG*, *SERIEPOLARE*, *SERIETRAIETT* e *BMINSERT*.

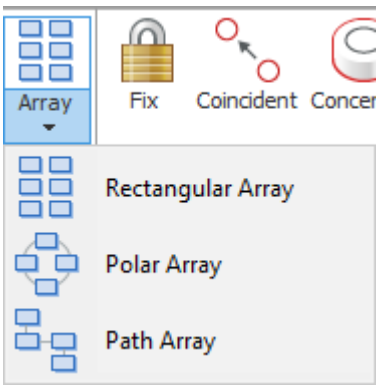
BricsCAD consente di inserire lo stesso componente più volte in modo regolare. Tutti gli inserti possono essere distribuiti uniformemente (alla stessa distanza) lungo una linea, un cerchio o una curva arbitraria. Sono supportati anche modelli di ordine superiore (ad esempio rettangolari o cilindrici). Ciò si ottiene utilizzando la tecnologia delle serie associative: rettangolari, polari e traiettoria.

Le serie associative sono entità persistenti, che possono essere modificate (manualmente o automaticamente) in qualsiasi momento dopo la creazione - per modificare il numero di elementi o la distanza tra loro. Tutti le serie del modello sono elencate nel Navigatore Meccanico, dove possono essere modificate.

Qualsiasi componente può essere incluso in una serie, comprese parti, sottoassiemi, parti standard e fori.

21.4.7.2 Crea una serie di componenti

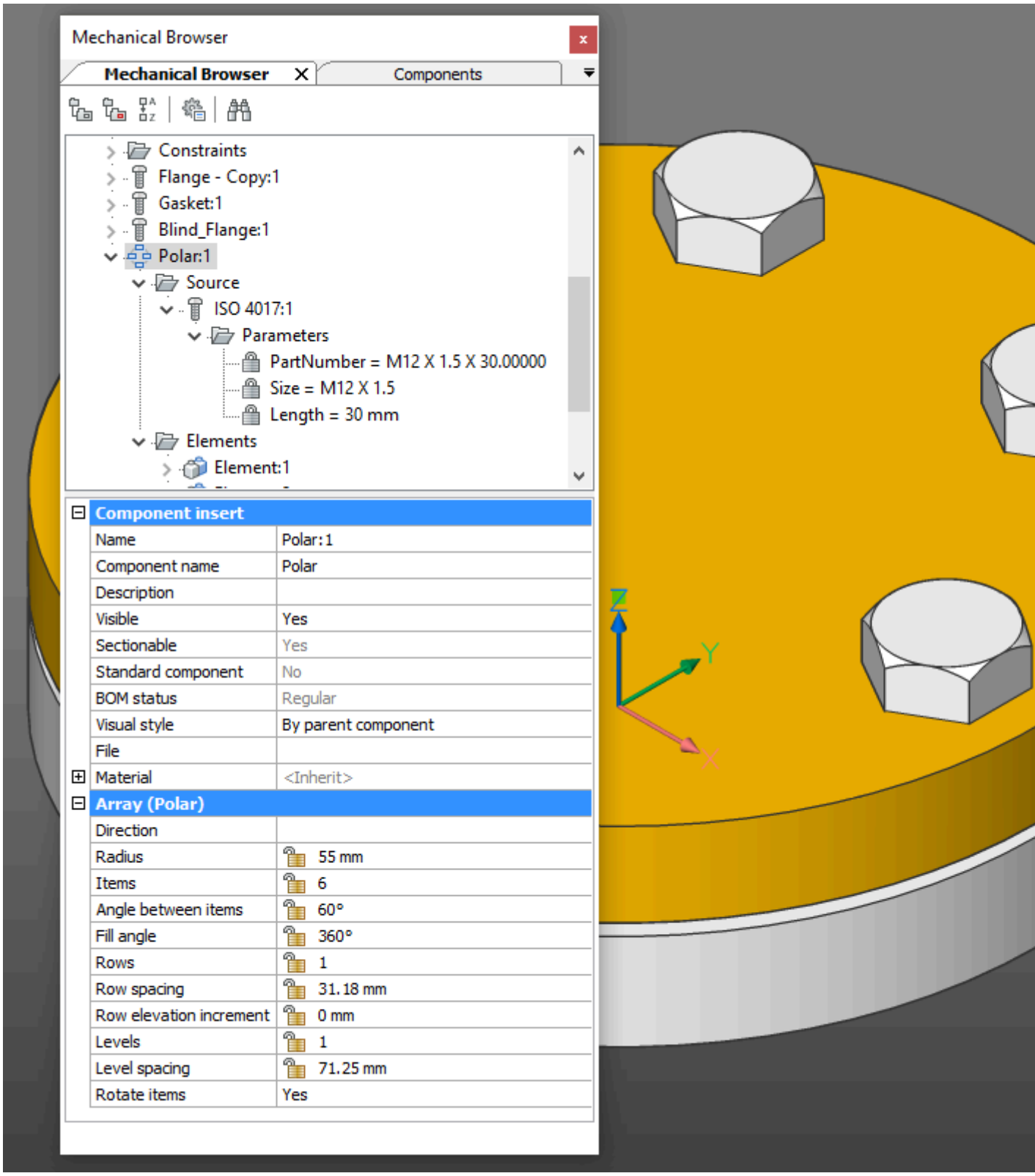
Utilizzare uno degli strumenti serie nel barra multifunzione **Assemblaggio** dell'area di lavoro Mechanical:



- **Serie Rettangolare**
- **Serie Polare**
- **Serie Traiettoria**.

Un altro modo per creare una serie di componenti è utilizzare l'opzione Multiplo/Serie dello strumento Inserisci Componente:

21.4.7.3 Serie di Componenti nel Navigatore Meccanico



Le serie di componenti sono rappresentate come nodi speciali nel Navigatore Meccanico. Essi prendono il nome dal tipo di serie (Rettangolare, Polare o Traiettoria) seguito dal punto e virgola e dall'indice della serie (ad esempio, Rettangolare:1, Polare:2, Traiettoria:3). Questi nodi hanno le stesse proprietà delle serie associative regolari. Le proprietà possono essere utilizzate per modificare il numero di elementi nella serie, la distanza/angolo tra loro e altri parametri della serie È anche possibile assegnare un'espressione persistente a qualsiasi proprietà. Verrà utilizzata per ricalcolare automaticamente il valore della proprietà dopo averne modificato i parametri che la controllano.

Ogni nodo della serie ha due nodi figlio; Sorgente ed Elementi:

- Nel nodo Sorgente è possibile trovare il componente inserito e i relativi parametri
- Il nodo Elementi può essere espanso in un elenco di elementi serie con la possibilità di nascondere o modificare lo stile di visualizzazione di un particolare elemento all'interno della serie.

21.4.7.4 Serie di Componenti e Distinta dei Materiali

Le serie di componenti sono trasparenti per la struttura di prodotto. i componenti all'interno delle serie sono considerati come inseriti direttamente nello stesso assieme. Pertanto, nella distinta base di livello superiore, i componenti inseriti nelle serie di primo livello sono elencati come componenti di primo livello.

La colonna **Quantità** nella tabella distinta dei materiali (BOM) viene riempita con il numero totale di elementi dello stesso componente inclusi i componenti all'interno delle serie.

21.4.7.5 Serie di componenti parametrici, parti standard e fori

Se un componente parametrico viene inserito come serie, i suoi parametri sono ancora disponibili per la modifica e per il collegamento con parametri globali. Tecnicamente i parametri sono associati con l'elemento sorgente di una serie. Quando questo elemento sorgente viene modificato parametricamente, tutti gli altri elementi della serie vengono modificati dopo di esso.

Le parti standard e i fori sono componenti parametrici ed è possibile modificarne i parametri all'interno delle serie. È possibile creare facilmente una parte con molti fori ed è possibile controllare il numero di foro, le posizioni e le dimensioni parametricamente in più inserire un bullone parametrico. Questa potente tecnologia consente di creare un assieme di flange parametrico con più configurazioni.

21.4.8 Proprietà Serie

21.4.8.1 Panoramica

La serie di componenti è una serie associativa costituita da un componente parametrico. Le serie di componenti hanno una rappresentazione speciale nel **Navigatore Meccanico**.

La serie di componenti ha 2 sottonodi principali:

- Fonte: nodo che contiene l'oggetto sorgente e i relativi parametri. L'oggetto di origine può essere presentato come componente o sottoinsieme con proprietà specifiche.
- Elementi: nodo che contiene tutte le istanze dell'oggetto sorgente

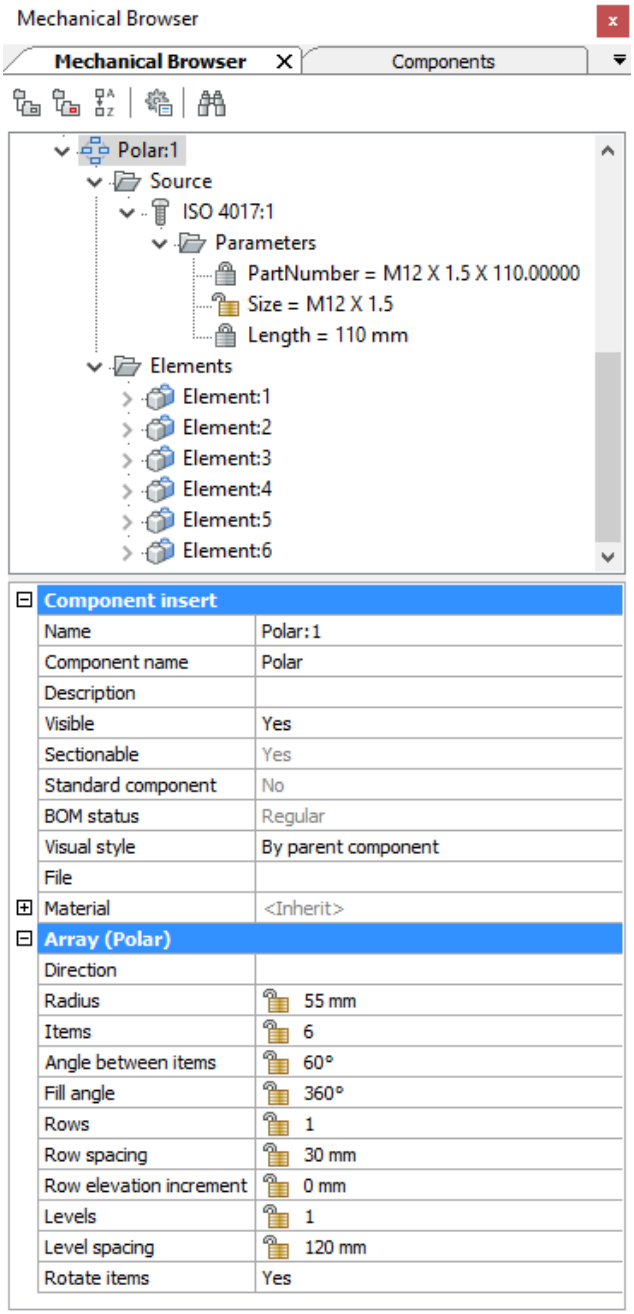
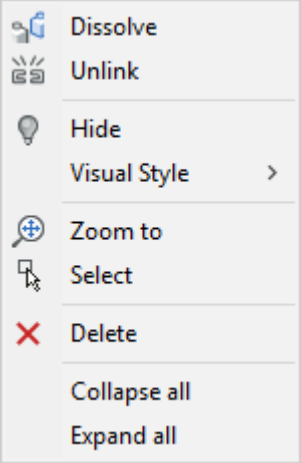
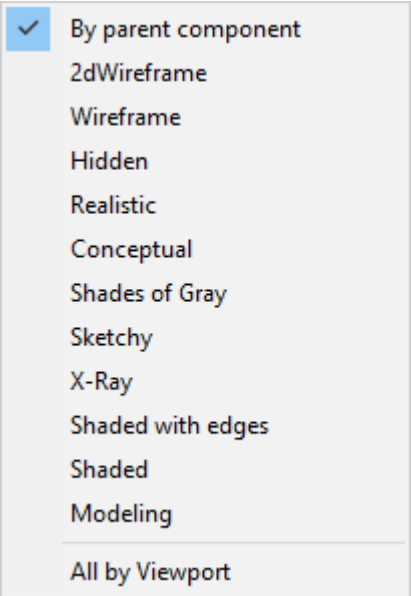


Figura 29: Navigatore Meccanico

Cliccando con il tasto destro del mouse nel nodo della serie di componenti viene visualizzato il menu contestuale mentre le proprietà della serie vengono visualizzate automaticamente quando si seleziona una serie nel Navigatore Meccanico.

Tabella 6: Menù contestuale

Menù contestuale	Descrizione
Menu contestuale delle serie di componenti	Fare clic con il tasto destro su una serie:  • **Dissolvi**: dissolve una serie creata nel disegno corrente (Vedere il comando *BMDISSOLVE*). • **Scollega**: scollega una serie da un solido 3D dissolvendo la caratteristica sottostante basata sul componente. • **Nascondi** / **Mostra** : Nasconde o mostra la serie selezionata. • **Escludi tutti gli elementi dalla sezione**: imposta la proprietà *Sezionabile* di tutti gli elementi simili su *NO*. Definisce se un elemento è influenzato dal comando *VISTASEZIONE*. • **Stile di Visualizzazione**: mostra il menu **Stile di Visualizzazione**. Sono disponibili gli Stili di Visualizzazione salvati nel disegno corrente.  • **Da Finestra**: Esegue il rendering della serie selezionata in base al modoombra della vista corrente (consultare il comando *SHADEMODE*). • **Tutto da Finestra**: esegue il rendering di tutte le serie in base allo shademode della vista corrente (consultare il comando *SHADEMODE*). • **Zoom a**: ingrandisce il gruppo di selezione • **Seleziona**: Seleziona la serie. • **Elimina**: rimuove la serie selezionata dall'assieme. • **Comprimi tutto**: Comprime il componente principale e tutti i componenti e sottocomponenti. • **Espandi tutto**: Espande il componente principale e tutti i componenti e sottocomponenti.

21.4.8.2 Comandi correlati

Tabella 7: Comandi correlati

Comando	Descrizione
SERIE3D	Crea serie 3D.
SERIE	Crea una serie polare o rettangolare di entità; visualizza le richieste nella barra dei comandi.
SERIECLASSICA	Crea un serie statica, polare o rettangolare, di entità tramite una finestra di dialogo.
MODIFSERIE	Modifica le entità serie associative e le relative entità di origine.
SERIETRAIETT	Distribuisce in modo associativo copie di entità in modo uniforme lungo un percorso in più righe e livelli.
SERIEPOLARE	Distribuisce associativamente copie di entità in modo uniforme in una serie circolare attorno a un punto centrale o un asse di rotazione, utilizzando più righe e livelli.
SERIERETTANG	Distribuisce associativamente copie di entità in un numero qualsiasi di righe, colonne e livelli.
INSERM	Inserisce serie di di blocchi.

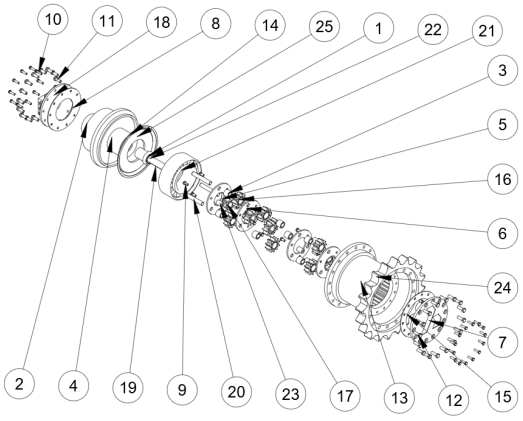
21.4.9 Rappresentazioni Esplose

21.4.9.1 Panoramica

Utilizzare i comandi *BMEXPLODE*, *BMEXPLODEMOVE*, *BMTRAILINGLINES* e *VISTAVASE*.

La funzionalità rappresentazione esplosa crea rappresentazioni esplose associative di assiemi e non modifica l'assieme di orgine. Le rappresentazioni esplose vengono memorizzate in blocchi dedicati, che possono essere modificati e inseriti in base alle proprie esigenze. È inoltre possibile generare viste di disegno dalle rappresentazioni esplose.

Le rappresentazioni esplose sono associative. Ogni parte in una rappresentazione esplosa è collegata alla parte corrispondente nell'assieme. Utilizzare il comando *BMBALLOON* per aggiornare le rappresentazioni esplose allo stato corrente dell'assieme e posizionare le annotazioni dei richiami sulle viste di disegno corrispondenti.



Bill of Materials Drive_Assembly		
No.	Component	Quantity
1	BBRG-03	2
2	BBRG-03	1
3	BBRG-60	9
4	Body	1
5	Carrier_1	2
6	Carrier_2	2
7	Cover	1
8	Cover_Interal_Seal	1
9	DIN 933	12
10	DIN 933	24
11	DIN 933	16
12	DIN 933	16
13	Drum	1
14	External_Medium_Seal	1
15	External_Seal	1
16	Gear_Planet	9
17	Gear_Sun	3
18	Internal_Cover	1
19	Key	1
20	Pin	3
21	RLR BRG-32	1
22	Shaft	1
23	Sleeve	6
24	Sprocket	1
25	Washer	1

Figura 30: Esempio di Rappresentazione Esplosa

21.4.9.2 Livelli della rappresentazione esplosa

Le rappresentazioni esplose contengono riferimenti ai componenti.

Sono supportati 2 livelli di rappresentazioni esplose: Alto e Basso.

- **Livello Alto**:

  la rappresentazione esplosa contiene solo riferimenti ai componenti di primo livello. I componenti di primo livello sono componenti inseriti direttamente nell'assieme principale. Un componente di primo livello in una rappresentazione esplosa non viene ulteriormente esploso: tutte le sue parti e i suoi sottoassiemi rimangono assemblati. Una rappresentazione esplosa di livello alto consente di vedere la composizione di base del tuo assieme.

- **Livello Basso**:

  la rappresentazione esplosa contiene solo riferimenti ai componenti di livello inferiore. I componenti di livello inferiore sono parti terminali, che non contengono altre parti o sottoassiemi. Le rappresentazioni esplose di livello basso consentono di visualizzare tutte le parti in isolamento.

Nota: Se si seleziona **livello Alto** e sono presenti alcuni componenti di livello medio contenenti non solo l'inserimento di componenti, ma anche altre entità, queste andranno perse nella rappresentazione esplosa perché non è possibile riferirle correttamente. Queste entità devono essere prima convertite in componenti di livello basso corrispondente.

21.4.9.3 Algoritmi

BricsCAD fornisce 3 algoritmi per creare automaticamente rappresentazioni esplose e un algoritmo manuale.

Tabella 8: Algoritmi

Algoritmo	Descrizione
Tabella per Tipo	Crea un'esploso simile a una tabella in cui i componenti dello stesso tipo sono raggruppati in righe.
Lineare	Individua la sequenza di smontaggio dei componenti in una determinata direzione e ordina i componenti. ***Nota:*** Considera possibili collisioni fisiche tra i componenti. I componenti possono essere spostati dall'algoritmo solo se non ci sono altri componenti (non ancora spostati), che li bloccano.
Automatico	Individua la sequenza di disassemblaggio dei componenti rispetto alla gerarchia di assemblaggio. ***Nota:*** Considera tutte le possibili collisioni fisiche. Per ogni parte o sottoassieme, la direzione di movimento viene identificata automaticamente.
Manuale	Crea una copia esatta dell'assieme pronta per la modifica personalizzata. In questa modalità è possibile creare una rappresentazione esplosa personalizzata senza modificare l'assemblaggio principale.

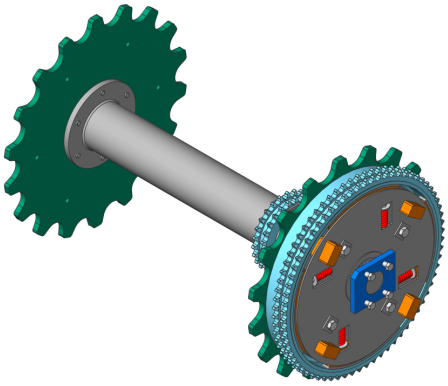
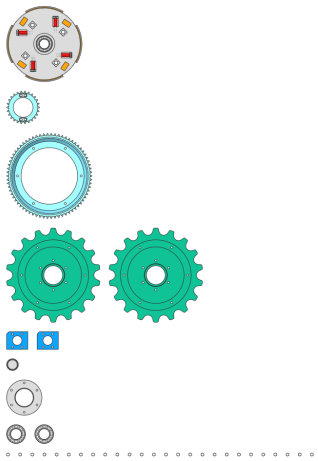
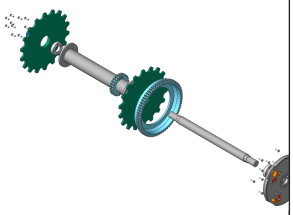
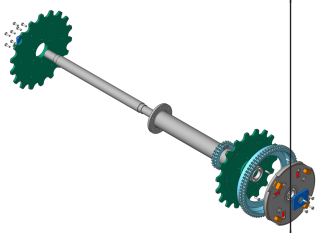
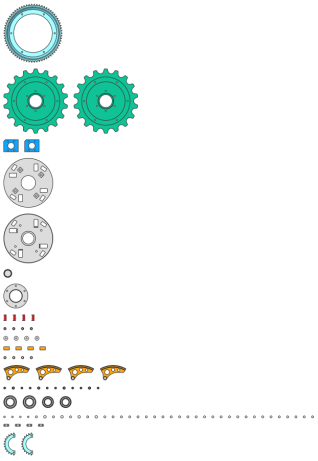
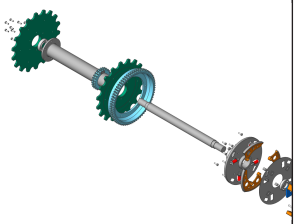
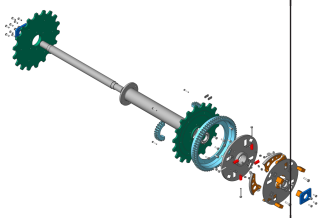


Figura 31: Esempio: il gruppo dell'albero di trasmissione principale

Tabella 9: Esempio: gruppo albero di trasmissione principale

	Tabella per Tipo	Lineare	Automatico
Livello Alto			
Livello Basso			

21.4.9.4 Step e animazioni

Le viste esplose possono contenere uno o più step, che descrivono una particolare sequenza di assemblaggio o smontaggio. Ciascun step corrisponde a un insieme di componenti che devono essere spostati in questa fase particolare per poter assemblare o smontare l'assieme principale. Tutti gli algoritmi di rappresentazione esplosa creano automaticamente tutti gli step necessari. È inoltre possibile eliminare, unire, dividere, spostare o aggiungere ulteriori step.

Ogni step ha un nome univoco che può essere utilizzato per descrivere lo step.

È possibile animare un particolare step, nonché l'intera sequenza di step, sia diretta che inversa (per ottenere animazioni di smontaggio e assemblaggio). Affinché l'animazione funzioni correttamente, una fase iniziale che definisce lo stato iniziale dell'assieme deve essere presente nella rappresentazione

esplosa. Utilizzare la proprietà **Nascondi automaticamente** della rappresentazione esplosa per nascondere automaticamente i componenti non importanti per un particolare step durante l'animazione.

21.4.9.5 Modifica delle rappresentazioni esplose

Ogni rappresentazione esplosa viene memorizzata come blocco. Ciò consente di modificare la rappresentazione esplosa con l'editor di blocchi.

Importante: Non utilizzare il comando *MODRIF* per modificare le rappresentazioni esplose. Questo comando è incompatibile con gli step delle rappresentazioni esplose.

Per aprire la rappresentazione esplosa per la modifica:

1 Cliccare il pulsante destro del mouse sulla rappresentazione esplosa richiesta nel Navigatore Meccanico.

2 Selezionare **Edita** dal menu contestuale.

L'editor blocchi si aprirà automaticamente.

Suggerimento: In alternativa, è possibile aprire la rappresentazione esplosa per la modifica alla fine del comando *BMEXPLODE*.

Quando la rappresentazione esplosa è aperta per la modifica, è possibile modificare lo stato dello step corrente. Per impostazione predefinita, questo è l'ultimo step realizzato dall'algoritmo. Tuttavia, è possibile modificare questa impostazione in due modi:

- Utilizzare la proprietà **Step corrente** della rappresentazione esplosa nel navigatore meccanico.
- Cliccare il pulsante destro del mouse sullo step desiderato nel Navigatore Meccanico e selezionare **Applica** dal menu contestuale.

Nota: Se si modifica lo step corrente, verranno eliminate tutte le modifiche non salvate e tutte le operazioni relative agli step. Assicurati che tutte le modifiche siano salvate prima di modificare lo step corrente o eseguire qualsiasi altra operazione.

Utilizzare i comandi *DMMOVE*, *SPOSTA*, *DMROTATE* e *BMEXPLODEMOVE* per spostare i componenti, all'interno delle rappresentazioni esplose, nelle posizioni desiderate. Il comando *BMEXPLODEMOVE* aggiunge automaticamente tutti i passaggi necessari dopo il passaggio corrente. Per altre operazioni, salvare lo step corrente al termine della modifica. Viene memorizzata solo la differenza tra lo step corrente e quello precedente.

È anche possibile modificare la rappresentazione esplosa eseguendo operazioni sugli step.

Esempio: se si rimuove uno step, tutti i componenti spostati in questo step verranno restituiti alle rispettive posizioni prima di tale step.

Salva lo step corrente una volta terminata la modifica. In caso contrario, le modifiche verranno scartate.

21.4.9.6 Linee guida

Le linee guida aiutano a comprendere il rapporto tra le parti. È una polilinea 3D che rappresenta una traiettoria e visualizza il movimento di una parte durante il processo di smontaggio.

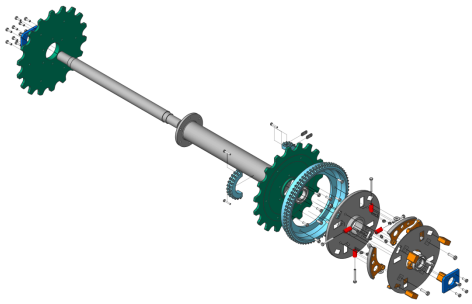


Figura 32: Esempio di linee guida

Utilizzare il comando *BMTRAILINGLINES* per aggiungere linee guida a una rappresentazione esplosa aperta per la modifica. Questo strumento crea tutte le linee di tracciamento. Per eseguire questa operazione, vengono utilizzate le informazioni degli step della vista esplosa.

È possibile generare linee guida per tutte le parti o per un sottoassieme di parti. Per un sottoassieme, verranno create linee guida solo per le parti di quel sottoassieme, anche se verranno aggiunte solo le linee guida necessarie. Alcune parti potrebbero non avere affatto linee guida. È inoltre possibile selezionare manualmente 2 parti per aggiungere una linea guida.

Quando viene calcolata una linea guida, l'algoritmo determina automaticamente la posizione della parte e la posizione da cui la parte è stata rimossa per calcolare la traiettoria. Considerano i movimenti di entrambe le parti.

Ogni linea guida viene visualizzata nel navigatore meccanico. Con il navigatore meccanico, è possibile visualizzare tutte le linee guida nel modello; evidenziare, selezionare e ingrandire le parti collegate alla linea selezionata; rinominare o rimuovere la linea guida.

È possibile modificare le linee guida con gli strumenti standard di modifica delle polilinee 3D .

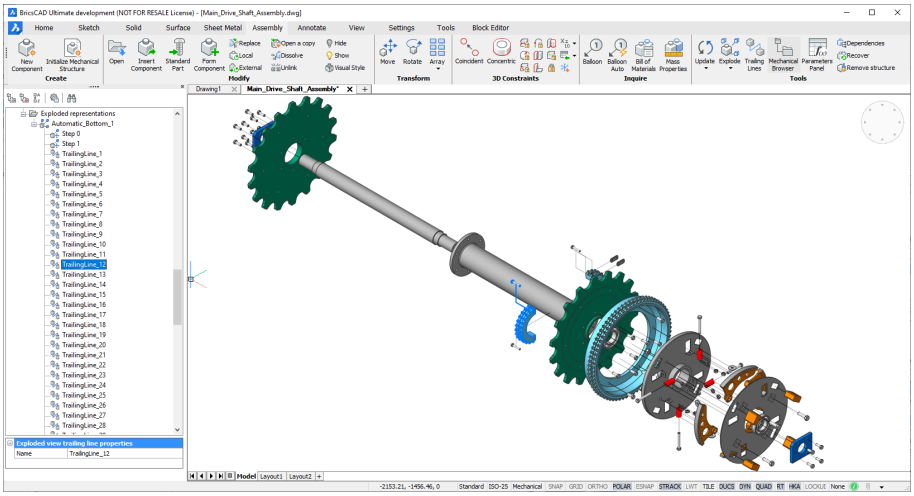


Figura 33: Proprietà della linea guida della vista esplosa

21.4.9.7 Crea una rappresentazione esplosa

Utilizzare il comando *BMEXPLODE* per creare una rappresentazione esplosa dell'assieme corrente.

Nota: È necessario disporre di almeno 1 elemento di una parte nell'assieme affinché il comando funzioni.

1 Scegliere una delle seguenti azioni per avviare il comando *BMEXPLODE*:
   - Cliccare sullo strumento **Esplodi** nel pannello **Strumenti** della scheda **Assemblaggio**.
   - Cliccare sullo strumento **Esplodi** sulla barra degli strumenti **Assemblaggio**.
   - Selezionare **Esplodi** nel menu **Assieme**.
   - Digitare **bmexplode** nella barra dei comandi.

   Viene richiesto: Selezionare algoritmo di esplosione [Tabella da tipi/Lineare/Automatico/Manuale/Impostazioni]:

2 Se applicabile, impostate il *Livello* e il *Nome* della rappresentazione esplosa utilizzando l'opzione **Impostazioni**.

3 Selezionare l'algoritmo.

   Viene richiesto: Selezionare il comportamento della vista esplosa [Edita/Generare viste di disegno/Termina]:

4 Eseguire una delle seguenti azioni:
   - Selezionare **Edita** per modificare la rappresentazione esplosa.
   - Selezionare **Genera viste di disegno** per generare viste di disegno della rappresentazione esplosa.
   - Selezionare **Termina** per terminare il comando.

21.4.9.8 Genera viste di disegno della rappresentazione esplosa

1 Avviare il comando *VISTABASE*.

   Viene richiesto: Selezionare gli oggetti o [modello Intero/Preimpostato/viste Speciali/opzioni di selezione (?)] <modello Intero>:

2 Selezionare l'opzione **Viste speciali**.

   Viene richiesto: Seleziona vista [vista Esplosa/vista Spiegata/Indietro] <Indietro>:

3 Selezionare l'opzione **Vista Esplosa**.

   Viene visualizzata una finestra di dialogo per selezionare la rappresentazione esplosa.

4 Selezionare la rappresentazione esplosa.

   Viene richiesto: Inserisci il nome del nuovo layout o di uno esistente da rendere corrente <Layout1>:

5 Se applicabile, immettere il *nome Layout* del layout per la rappresentazione esplosa.

6 Posizionare le viste di disegno generate della rappresentazione esplosa nel layout dello spazio carta.

Nota: Per utilizzare i richiami (vedere il comando *BMBALLOON*) con le viste di disegno delle rappresentazioni esplose, la variabile di sistema *GENERATEASSOCVIEWS* deve essere *ON*.

21.4.9.9 Aggiorna una rappresentazione esplosa

Avviare il comando *BMUPDATE* per sincronizzare le rappresentazioni esplose con lo stato corrente dell'assieme.

Se applicabile, modificare la rappresentazione esplosa. Tali modifiche potrebbero essere necessarie a causa dell'aggiunta e/o della cancellazione di alcune parti.

Nota: La posizione delle parti nelle rappresentazioni esplose create in modalità **Manuale** e **Lineare** non viene aggiornata automaticamente.

21.4.9.10 Gestisci rappresentazioni esplose

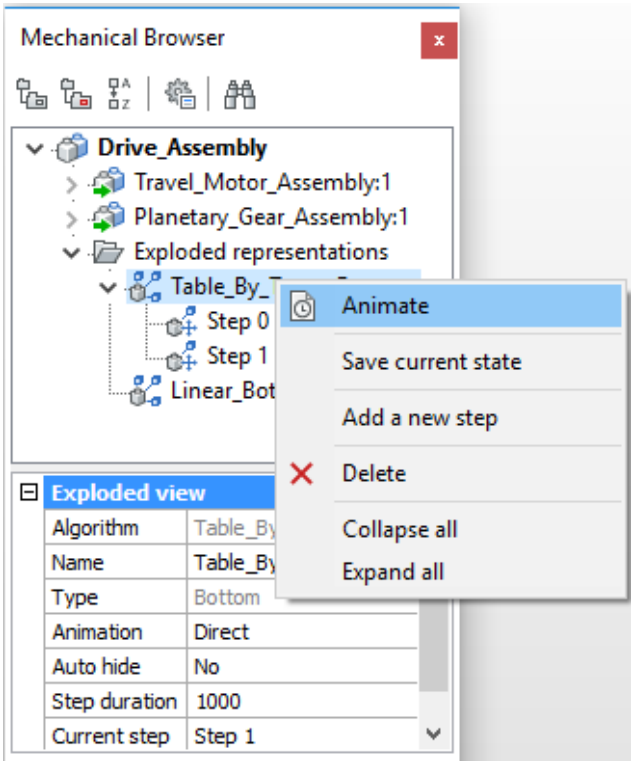


Figura 34: Gestisci rappresentazioni esplose

Aprire il Navigatore Meccanico per visualizzare tutte le rappresentazioni esplose nel documento. Tutte le rappresentazioni esplose saranno elencate nel gruppo **Rappresentazioni esplose**. Aprire il menu contestuale per le rappresentazioni esplose selezionate per visualizzare tutte le operazioni supportate in questo contesto.

Modifica delle rappresentazioni esplose

Ogni rappresentazione esplosa ha un insieme di proprietà. Puoi modificare alcune proprietà

Tabella 10: Modifica delle rappresentazioni esplose

Proprietà	Descrizione
Algoritmo	Il nome dell'algoritmo utilizzato per creare questa rappresentazione esplosa.
Nome	Nome della rappresentazione esplosa. Deve essere univoco per tutte le rappresentazioni esplose, così come per tutti i blocchi nel modello.

Tabella 10: Modifica delle rappresentazioni esplose

Proprietà	Descrizione
Tipo	Tipo di rappresentazione esplosa: *Alto* o *Basso*.
Animazione	La direzione dell'animazione. Scegli 1 di 2 valori: *Diretto* (dall'inizio alla fine) o *Inverso* (dalla fine all'inizio).
Nascondi automaticamente	Se impostato su *Sì*, tutte le parti non importanti per un particolare passaggio verranno nascoste durante l'animazione.
Durata dello step	Durata predefinita di ciascun step durante l'animazione (in millisecondi). Il valore predefinito è 1000, che è uguale alla durata di 1 secondo.
Step corrente	Lo step corrente che corrisponde allo stato della rappresentazione esplosa.

Operazioni su rappresentazioni esplose

Le operazioni supportate dipendono dal fatto che la rappresentazione esplosa sia aperta o meno per la modifica.

Se la rappresentazione esplosa **non è aperta per la modifica**, è possibile eseguire le seguenti operazioni:

Tabella 11: Operazioni su rappresentazioni esplose

Operazione	Descrizione
Edita	Apre la rappresentazione esplosa per la modifica. ***Suggerimento:*** È inoltre possibile modificare la rappresentazione esplosa con il comando *MODIFBLOC*.
Elimina	Rimuove la rappresentazione esplosa e il blocco associato dal documento.

Se la rappresentazione esplosa **è aperta per la modifica**, è possibile eseguire le seguenti operazioni:

Tabella 12: Operazioni su rappresentazioni esplose

Operazione	Descrizione
Anima	Analizza l'intera sequenza di step.
Salva lo stato corrente	Salva le posizioni correnti delle parti nello step corrente.

Tabella 12: Operazioni su rappresentazioni esplose

Operazione	Descrizione
Aggiungi un nuovo step	Aggiungi un nuovo step dopo l'ultimo. ***Nota:*** Il nuovo step verrà automaticamente impostato come *corrente*. Tutte le modifiche non salvate della rappresentazione esplosa andranno perse.
Elimina	Rimuove la rappresentazione esplosa e il blocco associato dal documento.

Inoltre, tutti gli step saranno elencati nel Navigatore Meccanico per questa rappresentazione esplosa.

21.4.9.11 Gestisci step della rappresentazione esplosa

Aprire il Navigatore Meccanico e aprire la rappresentazione esplosa necessaria per la modifica. Tutti gli step verranno elencati sotto il nodo corrispondente nella ramificazione.

Proprietà degli step

Tabella 13: Proprietà degli step

Proprietà	Descrizione
Nome	Il nome dello step. Deve essere unico per questa rappresentazione esplosa.
Durata, ms	La durata dello step. Per impostazione predefinita, è uguale alla durata dello step impostata per la rappresentazione esplosa.
Tipo durata	Specificare se la durata deve essere presa dalla rappresentazione esplosa o se è specifica per questo particolare step.
Valore durata (solo per il tipo di durata Assoluto)	La durata di questo step è in millisecondi.

Operazioni sugli step

Tabella 14: Operazioni sugli step

Operazione	Descrizione
Applica	Imposta lo step come corrente e aggiorna le parti nella rappresentazione esplosa in base a questo step.
Anima	Anima lo step selezionato.

Tabella 14: Operazioni sugli step

Operazione	Descrizione
Imposta come precedente	Cancella lo step, in modo che le posizioni di tutte le parti siano le stesse del passaggio precedente.
Fondi con precedente	Unisce questo e lo step precedente. Lo step precedente viene rimosso.
Dividi	Divide lo step in una sequenza di step, ciascuno di essi corrispondente esattamente a una parte.
Sposta sù	Sposta lo step di una posizione verso l'alto nella ramificazione.
Sposta giù	Sposta il passo di una posizione verso il basso nella ramificazione.
Elimina	Rimuove completamente lo step e i movimenti associati delle parti. Dopo di che, le posizioni delle parti vengono aggiornate nella rappresentazione esplosa.

21.4.9.12 Modifica delle rappresentazioni esplose

1 Selezionare il componente esploso nel Navigatore Meccanico.

2 Cliccare il pulsante destro del mouse sulla rappresentazione esplosa e selezionare **Edita** dal menu contestuale.

Nota: In alternativa, è possibile eseguire il comando *MODRIF* e selezionare il blocco con la rappresentazione esplosa.

3 Eseguire le operazioni necessarie per ottenere la rappresentazione esplosa desiderata.

4 Salva lo step corrente.

5 Salvare le modifiche usando il comando *BCLOSE* con l'opzione *Salva*.

Nota: Non utilizzare l'opzione *Esci* del comando *BCLOSE*, poiché può portare a uno stato incoerente del blocco e degli step della rappresentazione esplosa. Usa invece *Annulla*.

21.4.9.13 Aggiungi le linee guida

1 Aprire la rappresentazione esplosa per la modifica.

2 Eseguire il comando *BMTRAILINGLINES*.

3 Selezionare il sottoassieme di parti per cui creare linee guida o seleziona l'intero modello.

4 Selezionare i punti delle parti da utilizzare per creare la traiettoria: origine o centro.

5 Se applicabile, modificare le linee guida risultanti.

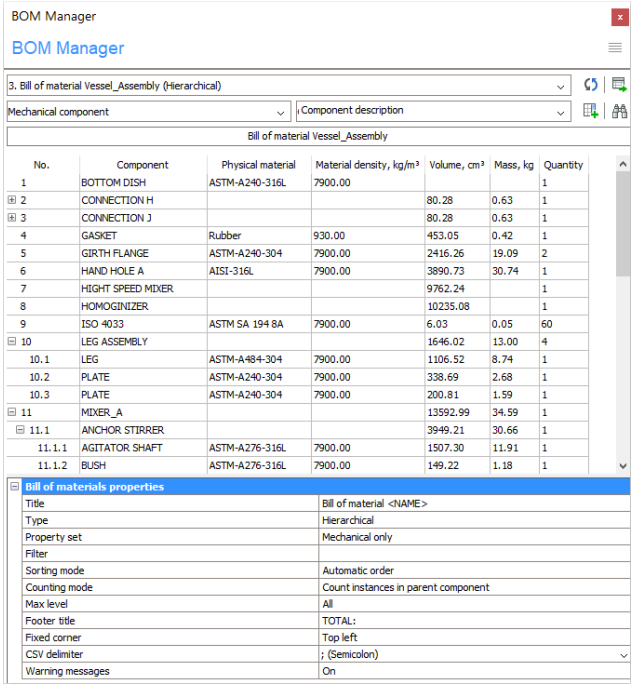
6 Salvare le modifiche.

21.4.10 Gestore Distinta dei Materiali

21.4.10.1 Panoramica

Utilizzare i comandi *BMBOMPANELOPEN*, *BMBOMPANELCLOSE* e *BMBOM*.

Il Gestore Distinte dei Materiali consente di sfogliare, creare e modificare le Distinte dei Materiali (BOM) nel disegno corrente. Utilizzare il gestore per configurare in modo interattivo le distinte dei materiali, sia nuove sia esistenti. La configurazione di una distinta dei materiali può essere salvata in un file modello e riutilizzata successivamente in altri disegni. È inoltre possibile aggiungere una distinta dei materiali non collegata a una tabella nel disegno.



No.	Component	Physical material	Material density, kg/m³	Volume, cm³	Mass, kg	Quantity
1	BOTTOM DISH	ASTM-A240-316L	7900.00			1
2	CONNECTION H			80.28	0.63	1
3	CONNECTION J			80.28	0.63	1
4	GASKET	Rubber	930.00	453.05	0.42	1
5	GIRTH FLANGE	ASTM-A240-304	7900.00	2416.26	19.09	2
6	HAND HOLE A	AISI-316L	7900.00	3890.73	30.74	1
7	HIGHT SPEED MIXER			9762.24		1
8	HOMOGINIZER			10235.08		1
9	ISO 4033	ASTM SA 194 8A	7900.00	6.03	0.05	60
10	LEG ASSEMBLY			1646.02	13.00	4
10.1	LEG	ASTM-A484-304	7900.00	1106.52	8.74	1
10.2	PLATE	ASTM-A240-304	7900.00	338.69	2.68	1
10.3	PLATE	ASTM-A240-304	7900.00	200.81	1.59	1
11	MIXER_A			13592.99	34.59	1
11.1	ANCHOR STIRRER			3949.21	30.66	1
11.1.1	AGITATOR SHAFT	ASTM-A276-316L	7900.00	1507.30	11.91	1
11.1.2	BUSH	ASTM-A276-316L	7900.00	149.22	1.18	1

Figura 35: Gestore BOM

Con il Gestore Distinte dei Materiali è possibile:

- Sfogliare le distinte materiali esistenti nel disegno.
- Creare una nuova distinta materiali utilizzando un tipo standard (come livello superiore, livello inferiore, gerarchico) o un modello BOM esistente.
- Modificare le distinte dei materiali:
  - Aggiungere o rimuovere colonne (incluse le colonne formula).
  - Spostare le colonne selezionate in una nuova posizione.
  - Rinominare le colonne.
  - Configurare la modalità di ordinamento della tabella BOM.
  - Configurare il valore del piè di pagina per ogni singola colonna.
  - Configurare le unità di ogni colonna e il modo in cui vengono visualizzate.
  - Configurare la larghezza di ogni colonna con le proprietà della colonna.

  - Configurare la visibilità di ogni singola colonna.
  - Filtrare i componenti con un'espressione filtro.
  - Modificare il titolo sia della riga del Titolo sia della riga del Piè di Pagina.
- Cercare i componenti nella tabella.
- Evidenziare e selezionare le istanze dei componenti.
- Salvare la configurazione BOM come file modello.
- Caricare una configurazione tabella BOM da un file modello.
- Posizionare la tabella BOM in un disegno.
- Eliminare tabelle BOM.

Per modificare o creare distinte base dei materiali è necessaria una licenza Mechanical o Ultimate. Tuttavia, il gestore è disponibile anche con una licenza Pro.

21.4.10.2 Aprire il Gestore BOM

Eseguire una delle seguenti operazioni:

- Posizionare il cursore su una barra degli strumenti, poi cliccare il tasto destro del mouse e scegliere **Pannelli > Gestore BOM** nel menu contestuale.
- Avviare il comando *BMBOMPANELOPEN*.

21.4.10.3 Strumenti

La maggior parte delle operazioni di configurazione del gestore sono disponibili tramite il menu principale. Un menu di scelta rapida è disponibile per le colonne o le righe selezionate. È inoltre possibile modificare le proprietà delle colonne selezionate e dell'intera BOM stessa tramite la sezione Proprietà della distinta dei materiali. Per lavorare con una particolare colonna, selezionarla sulla griglia BOM. È possibile selezionare più colonne contemporaneamente.

Menu principale

Cliccare su **Menu** (☰) nell'angolo in alto a destra del pannello Gestione BOM.

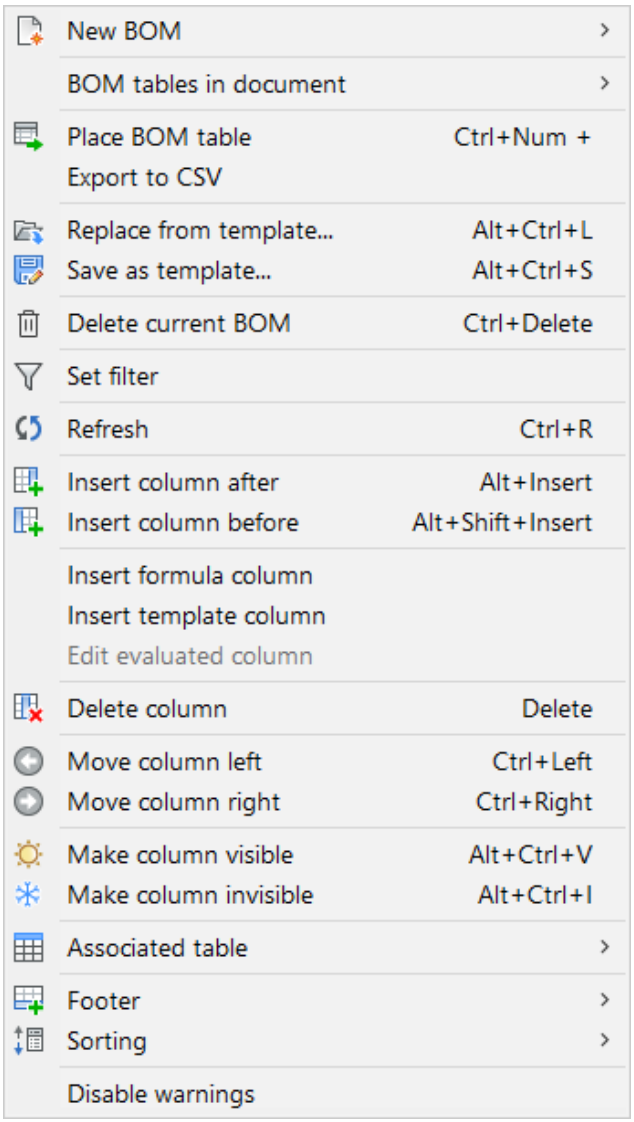


Figura 36: Menu principale

Tabella 15: Menu principale

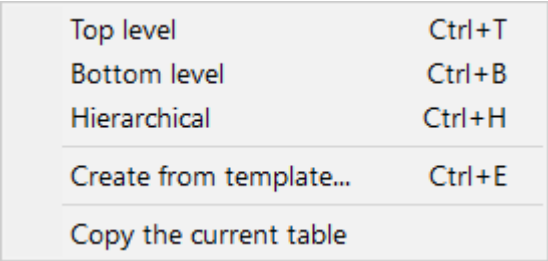
Icona	Strumento	Descrizione
	Nuova BOM	Apre un sottomenu:  **Figura 37:** Sottomenu • **Livello superiore**: crea una nuova BOM di livello superiore. • **Livello inferiore**: crea una nuova BOM di livello inferiore. • **Gerarchico**: crea una nuova BOM gerarchica. • **Crea da modello**: apre una finestra di dialogo per selezionare un file di modello di configurazione della distinta base. • **Copia la tabella corrente**: crea una nuova distinta base in base alla tabella corrente.
	Tabelle BOM nel documento	Apre un sottomenu: **Figura 38:** Sottomenu • **Vai alla prima tabella**: passa alla prima tabella BOM nel documento. • **Vai alla tabella precedente**: passa alla tabella BOM precedente nel documento. • **Vai alla tabella successiva**: passa alla tabella BOM successiva nel documento. • **Vai all'ultima tabella**: passa all'ultima tabella BOM nel documento. • **<Nome della BOM>**: passa alla tabella BOM corrispondente nel documento.

Tabella 15: Menu principale

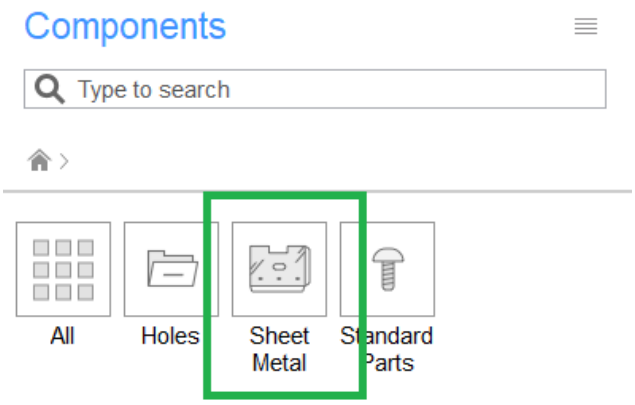
Icona	Strumento	Descrizione
	Posizionare/aggiornare una tabella BOM	Inizia il posizionamento interattivo della tabella nel disegno. Se la tabella è già posizionata nel disegno, aggiorna la tabella. ***Nota:*** Tutte le operazioni di modifica aggiornano automaticamente la tabella associata nel disegno.
	Esporta in CSV	Salva il contenuto della tabella corrente in un file CSV. Apre una finestra di dialogo per selezionare la posizione per il file CSV:  **Figura 39:** Selezionare il file per esportare la tabella BOM ***Nota:*** È possibile regolare il separatore per i valori CSV utilizzando le proprietà della tabella.
	Sostituisci da modello	Sostituisce la configurazione della tabella BOM corrente con il modello selezionato. Apre una finestra di dialogo per selezionare il file modello .
	Salva come template	Salva la configurazione della tabella BOM corrente come modello. Apre una finestra di dialogo per selezionare la posizione per il nuovo file modello .
	Elimina BOM corrente	Elimina la tabella BOM corrente dal documento.

Tabella 15: Menu principale

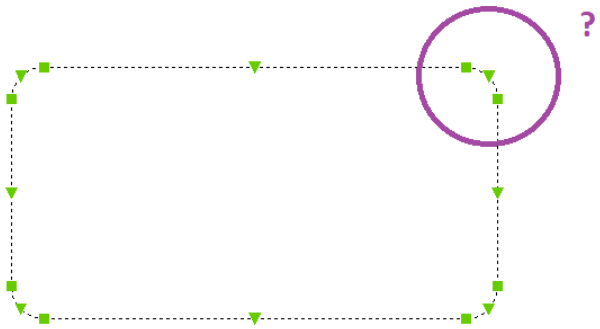
Icona	Strumento	Descrizione
	Imposta filtro	Consente di aprire una finestra di dialogo per configurare il filtro principale:  **Figura 40:** Configuratore filtro
	Aggiorna	Aggiorna la tabella BOM corrente ed elenca tutte le tabelle BOM nel disegno.
	Inserisci colonna dopo	Se viene selezionata una colonna: inserisce una colonna dal **Selettore colonne** dopo la colonna selezionata nella tabella. Se non è stata selezionata alcuna colonna: aggiunge una nuova colonna dopo l'ultima colonna nella tabella.
	Inserisci colonna prima	Se è selezionata una colonna: inserisce una colonna dal **Selettore colonne** prima della colonna selezionata nella tabella. Se non è stata selezionata alcuna colonna: aggiunge una nuova colonna come prima colonna della tabella.

Tabella 15: Menu principale

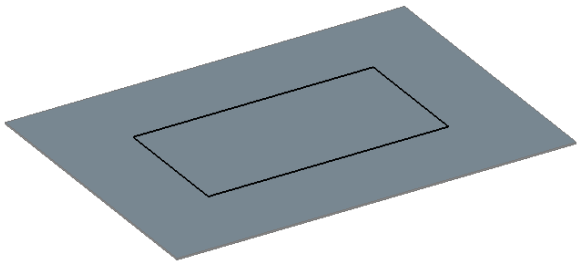
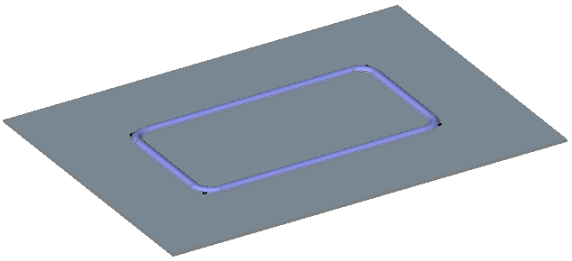
Icona	Strumento	Descrizione
	Aggiungi una colonna formula	Apre una finestra di dialogo per configurare una colonna formula:  **Figura 41:** Colonna Formula Fare clic su **OK** per aggiungere una nuova colonna formula dopo quella selezionata. Se non è stata selezionata alcuna colonna, la nuova colonna formula viene aggiunta dopo l'ultima colonna nella tabella.
	Inserisci colonna modello	Aprire una finestra di dialogo per configurare una colonna modello:  **Figura 42:** Colonna Modello Fare clic su **OK** per aggiungere una nuova colonna modello dopo quella selezionata. Se non è stata selezionata alcuna colonna, la nuova colonna modello viene aggiunta dopo l'ultima colonna nella tabella.
	Modifica colonna valutata	Apre una finestra di dialogo per modificare la formula o la colonna modello selezionata.

Tabella 15: Menu principale

Icona	Strumento	Descrizione
	Elimina colonna	Rimuovere la colonna selezionata dalla tabella. È inoltre possibile selezionare una colonna nella **griglia della BOM** e premere il tasto **Elimina** per rimuovere la colonna.
	Sposta colonna a sinistra	Sposta la(e) colonna(e) selezionate di una colonna a sinistra.
	Sposta colonna a destra	Sposta la(e) colonna(e) selezionate di una colonna a destra.
	Rendi visibile la colonna	Mostra la colonna selezionata nella tabella associata nel disegno.
	Rendi invisibile la colonna	Rimuove la colonna selezionata dalla tabella associata. Le colonne invisibili rimangono visibili nel gestore BOM, ma sono contrassegnate con uno schema di tratteggio diagonale sullo sfondo.

Tabella 15: Menu principale

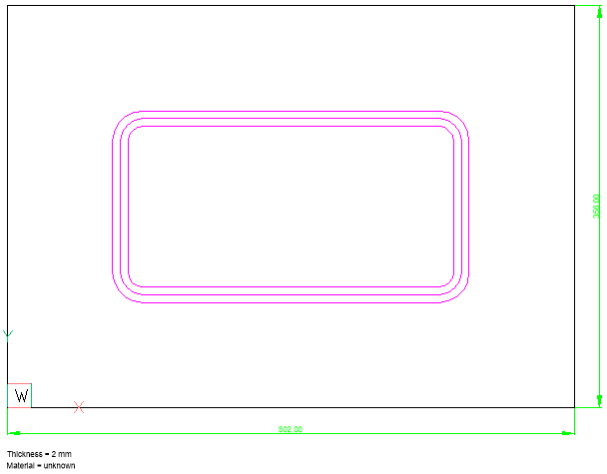
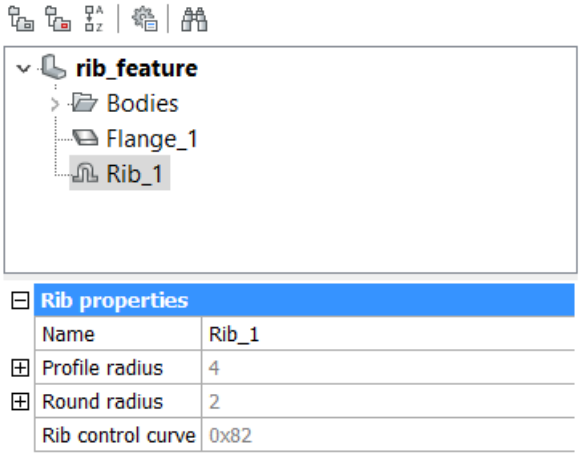
Icona	Strumento	Descrizione
	Tabella associata	Apre un sottomenu:  **Figura 43:** Sottomenu • **Zoom a**: zooma la tabella. Se necessario, impostare il layout della tabella come quello corrente. • **Seleziona**: seleziona la tabella. Se necessario, impostare il layout della tabella come quello corrente. • **Elimina**: elimina la tabella dal documento. • **Stacca**: stacca la tabella. La tabella nel documento non viene più aggiornata quando la tabella BOM viene modificata.
	Piè di pagina	Apre un sottomenu: **Figura 44:** Sottomenu Per ciascuna delle colonne selezionate: • **Nessuna funzione**: rimuove il piè di pagina. • **Somma**: visualizza la somma nel piè di pagina. • **Valore medio**: visualizza la media nel piè di pagina. • **Valore minimo**: visualizza il valore minimo nel piè di pagina. • **Valore massimo**: visualizza il valore massimo nel piè di pagina. Per ulteriori dettagli, vedere la sezione Piè di Pagina in Distinta dei Materiali alla pagina 1638.

Tabella 15: Menu principale

Icona	Strumento	Descrizione
	Riodina	Apre un sottomenu:  **Figura 45:** Sottomenu • **Ordine automatico**: ordina automaticamente la tabella. • **Ordine naturale**: ordina la tabella in base all'ordine naturale delle istanze dei componenti nel documento. • **Ordine crescente**: ordina le colonne selezionate in ordine crescente. • **Ordine decrescente**: ordina le colonne selezionate in ordine decrescente. Per maggiori dettagli, consulta la sezione Ordinamento in Distinta dei Materiali alla pagina 1638.
	Visualizza avvertimenti	Controlla la visualizzazione dei messaggi di avvertimento durante la creazione o l'aggiornamento delle tabelle BOM.

Barre degli Strumenti

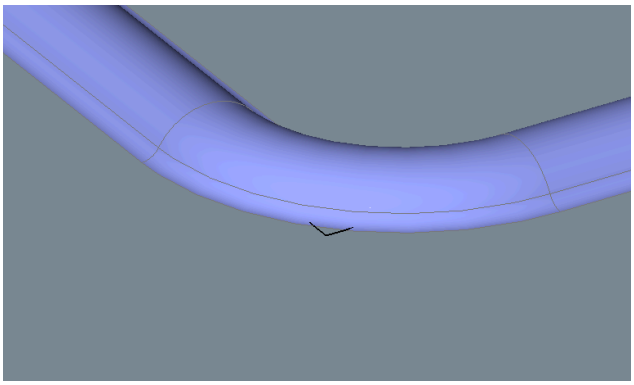


Figura 46: Gestore BOM

Tabella 16: Gestore BOM

Icona	Strumento	Descrizione
	Menu principale	Apre il menu principale.
	Selettore BOM	Seleziona la tabella BOM corrente. La selezione è memorizzata nel disegno. Quando il disegno viene chiuso e riaperto, la BOM selezionata viene ripristinata nel gestore. Se selezionata, la tabella viene aggiornata allo stato corrente del disegno.
	Aggiorna	Aggiorna sia la BOM corrente che l'elenco di tutte le tabelle BOM nel disegno.
	Inserisci/ Aggiorna tabella	Inizia il posizionamento interattivo della tabella nel disegno. Aggiorna la tabella se esiste già. ***Nota:*** Nota: tutte le operazioni di modifica aggiornano automaticamente le tabelle associate nel disegno.
	Selettore Categoria	Seleziona una categoria di proprietà.
	Selettore colonne	Seleziona la colonna da aggiungere alla BOM corrente. È anche possibile utilizzare questa opzione per immettere una formula per un campo formula. Per aggiungere una formula, digitare un'espressione che inizia con "=", ad esempio "=<Massa>*<Quantità>". La formula può fare riferimento ad altre colonne. Usa i nomi visualizzati racchiusi in <>.
	Inserisci colonna dopo	Se è selezionata una colonna: inserisce la colonna indicata nel **Selettore colonna** dopo la colonna selezionata nella tabella. Se non sono selezionate colonne: aggiunge la nuova colonna dopo l'ultima colonna nella tabella.

Tabella 16: Gestore BOM

Icona	Strumento	Descrizione
	Trova	Attiva e disattiva il campo di ricerca. Puoi anche usare **Ctrl+F**.
	Campo cerca	Immettere una stringa di ricerca; sono supportati i caratteri jolly.
	Trova precedente	Seleziona la corrispondenza precedente nella tabella BOM. Puoi anche usare **Ctrl+F3**.
	Trova successivo	Seleziona la corrispondenza successiva nella tabella BOM. Puoi anche usare **F3**.
	Seleziona tutte le corrispondenze trovate	Seleziona tutte le corrispondenze nella tabella BOM.

Menu di scelta rapida della colonna

Cliccare il pulsante destro del mouse sull'etichetta di una colonna per aprire il menu contestuale. La colonna viene aggiunta alla selezione corrente.

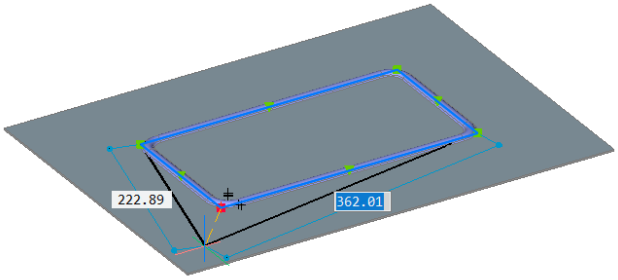


Figura 47: Menu di scelta rapida della colonna

Tabella 17: Menu di scelta rapida della colonna

Icona	Strumento	Descrizione
	Aggiungi colonna modello da selezione	Apre una finestra di dialogo per configurare una nuova colonna del modello. Le colonne selezionate verranno aggiunte automaticamente all'espressione del modello.

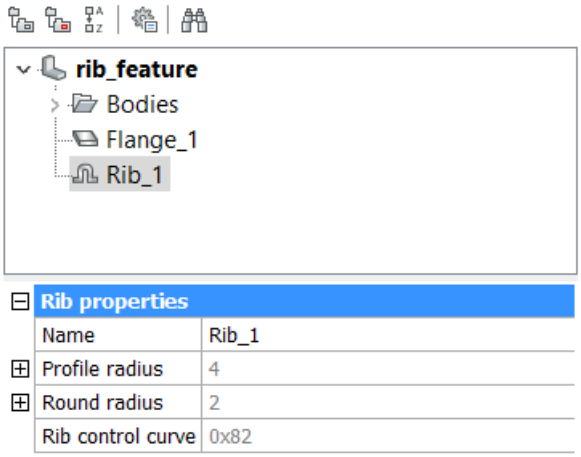
Tabella 17: Menu di scelta rapida della colonna

Icona	Strumento	Descrizione
	Elimina colonna	Rimuove le colonne selezionate dalla tabella. Puoi anche selezionare una colonna nella **griglia BOM** e premere il tasto **Elimina**.
	Piè di pagina	Apri un sottomenu per le colonne selezionate: • **Nessuna funzione**: rimuove il piè di pagina. • **Somma**: visualizza la somma nel piè di pagina. • **Valore medio**: visualizza il valore medio nel piè di pagina. • **Valore minimo**: visualizza il valore minimo nel piè di pagina. • **Valore massimo**: visualizza il valore massimo nel piè di pagina.
	Riodina	Apre un sottomenu: • **Ordine automatico**: ordina automaticamente la tabella**.** • **Ordine naturale**: ordina la tabella in base all'ordine naturale delle istanze dei componenti nel documento. • **Ordine crescente**: ordina le colonne selezionate in ordine crescente. • **Ordine decrescente**: ordina le colonne selezionate in ordine decrescente.

Tabella 17: Menu di scelta rapida della colonna

Icona	Strumento	Descrizione
	Funzione di aggregazione	Apri un sottomenu per le colonne selezionate: • **Nessuna funzione**: rimuovere la funzione di aggregazione. • **Somma**: visualizza la somma di tutti i valori del gruppo. • **Valore medio**: visualizza la media di tutti i valori del gruppo. • **Valore minimo**: visualizza il minimo di tutti i valori del gruppo. • **Valore massimo**: visualizza il massimo di tutti i valori del gruppo. • **Concatenazione**: mostra una concatenazione di tutti i valori nel gruppo in ordine crescente separata da un delimitatore specificato (';' per impostazione predefinita). • **Concatenazione con conteggio**: visualizza una concatenazione di tutti i valori del gruppo con il numero corrispondente di occorrenze in ordine crescente. Le coppie di valori e il conteggio sono separati da un delimitatore specificato (';' per impostazione predefinita).
	Ruolo colonna	Apre un sottomenu per la colonna selezionata: • **Numero**: la colonna viene utilizzata come sorgente di numeri dal comando *BMBALLOON*. • **Nome**: la colonna viene utilizzata come sorgente di nomi dal comando *BMBALLOON*. • **Quantità**: la colonna viene utilizzata come sorgente di quantità dal comando *BMBALLOON*. • **Regolare**: La colonna non ha un ruolo speciale.
%..	**Formato**	Imposta il formato specificato per le colonne selezionate. Apre una finestra di dialogo: **Figura 48:** Formato

Tabella 17: Menu di scelta rapida della colonna

Icona	Strumento	Descrizione
	Imposta filtro colonna	Apre una finestra di dialogo per configurare il filtro colonna:  **Figura 49:** Filtro colonna
	Rendi visibile la colonna	Mostra le colonne selezionate nella tabella associata nel disegno.
	Rendi invisibile la colonna	Rimuove la colonna selezionata dalla tabella associata. Le colonne invisibili rimangono visibili nel gestore BOM, ma sono contrassegnate con uno schema di tratteggio diagonale sullo sfondo.
	Adatta colonne	Adatta la larghezza delle colonne selezionate al loro contenuto. La larghezza della colonna si adatta automaticamente al contenuto a meno che la larghezza non sia stata modificata manualmente.
	Adatta tutte le colonne	Adatta la larghezza di tutte le colonne al contenuto. La larghezza della colonna si adatta automaticamente al contenuto a meno che la larghezza non sia stata modificata manualmente.

Menu contestuale della riga

Cliccare il pulsante destro del mouse su qualsiasi cella per aprire il menu contestuale.

Figura 50: Menù contestuale

Tabella 18: Menu contestuale della riga

Icona	Strumento	Descrizione
	Zoom a	Zooma sulla parte o sulle parti selezionate nel modello.
	Selezione	Seleziona le parti. Sostituisce una selezione esistente.
	Aggiungi alla Selezione	Aggiunge parti alla selezione.
	Rimuovi dalla Selezione	Rimuove le parti dalla selezione.
	Rimuovi evidenziazione	Rimuovi evidenziazione.
	Pulisci selezione	Pulisci selezione.
	Comprimi	Comprime il gruppo di sottoassiemi nella griglia BOM.
	Espandi	Espande il gruppo di sottoassiemi nella griglia BOM.
	Espandi questo livello	Espande tutti i gruppi di sottoassiemi allo stesso livello dell'albero dell'assieme di quello corrente.
	Comprimi questo livello	Comprime tutti i gruppi di sottoassiemi allo stesso livello dell'albero dell'assieme di quello corrente.
	Espandi tutto	Espande tutti i gruppi di sottoassiemi nella BOM corrente.
	Comprimi tutto	Comprime tutti i gruppi di sottoassiemi nella BOM corrente.
	Limita per questo livello	Imposta il livello corrente come livello massimo per le tabelle BOM gerarchiche.
	Mostra tutti i livelli	Rimuove il limite di livello massimo per le tabelle BOM gerarchiche.

Titolo BOM

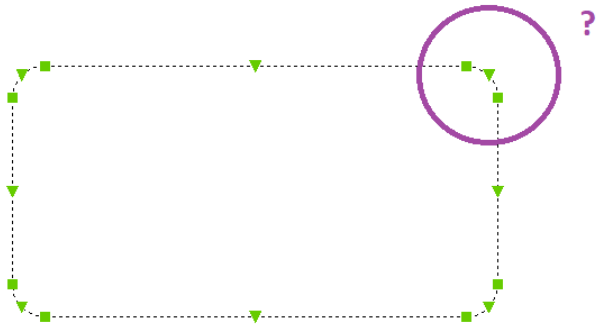


Figura 51: Titolo BOM

Visualizzare il titolo della BOM corrente, come mostrato nel disegno. Cliccare il tasto sinistro su questo controllo per modificare il disegno. In questa modalità, il controllo **Titolo BOM** mostra invece il **modello del titolo**. Il **modello del titolo** è una stringa che può contenere un segnaposto *<NOME>*, che verrà sostituito dal nome dell'assieme.

Griglia

No.	Component ▲2	Physical material	Material density	Is standard	Quantity ▼1
1	ISO 4033			Yes	60
2	MM-THD			Yes	30
3	LEG			No	4
4	FLANGE			No	2
5	CONNECTION H	ASTM-A276-316L	7900.00	No	1
6	CONNECTION J	ASTM-A276-316L	7900.00	No	1
7	GASKET			No	1
8	HAND HOLE			No	1
9	HIGHT SPEED MIXER			No	1
10	HOMOGINIZER			No	1
11	MIXER_A			No	1
12	NOZZLE B			No	1
13	NOZZLE D			No	1
14	NOZZLE F	AISI-316L	7900.00	No	1
15	NOZZLE L			No	1

Figura 52: Griglia

La griglia visualizza il contenuto della tabella BOM selezionata. Ogni riga è collegata con le istanze corrispondenti. Quando viene selezionata la riga, le parti corrispondenti vengono evidenziate nel disegno. Utilizzare il menu di scelta rapida per selezionare le parti corrispondenti, aggiungerle o rimuoverle dalla selezione corrente e zoomarle. Doppio clic su qualsiasi cella della riga per zoomare le parti corrispondenti.

Spostamento di una colonna

Per spostare una colonna nella tabella BOM, trascinarla e rilasciarla nella nuova posizione.

Colonne visibili e invisibili

Figura 53: Colonne visibili e invisibili

È possibile rendere invisibile qualsiasi colonna nella tabella BOM, in modo che non venga visualizzata nella tabella risultante. Le colonne invisibili rimangono visibili nella griglia della BOM con uno sfondo tratteggiato. È possibile modificare la visibilità delle colonne nel menu principale, nel menu contestuale delle colonne o modificando la proprietà **Visibile** di una colonna nella **griglia Proprietà**.

Regolare la larghezza

È possibile regolare la larghezza di qualsiasi colonna nella griglia BOM, la quale è indipendente dalla larghezza nella tabella associata. Se la larghezza della colonna è troppo stretta per visualizzare il testo della cella, l'altezza della riga corrispondente aumenta per adattarsi al testo.

Quando la larghezza della colonna viene regolata manualmente, la larghezza rimane la stessa indipendentemente dal contenuto della tabella. La larghezza è anche memorizzata nella configurazione BOM. Tutte le larghezze delle colonne regolate rimangono invariate anche quando il disegno viene chiuso e riaperto.

Per adattare automaticamente la larghezza delle colonne al contenuto, effettuate una delle seguenti operazioni:

- Selezionare le colonne, cliccare il pulsante destro del mouse e sul menu contestuale selezionare **Adatta colonne**.
- Doppio clic sul separatore nell'intestazione della colonna.

Ordinare una BOM

Utilizzare la griglia per controllare come ordinare il contenuto. Fare doppio clic sull'etichetta di una colonna per cambiare l'ordine tra crescente, decrescente e non ordinato. Se una colonna viene utilizzata per ordinare la tabella, alla relativa etichetta viene aggiunto un simbolo (▲ per crescente ? per decrescente). Doppio clic sull'etichetta per ripristinare l'ordinamento.

È possibile utilizzare più colonne per ordinare. I simboli di ordinamento di queste colonne hanno un pedice per indicare il criterio di ordinamento. Ad esempio, se si ordina in base alla quantità in ordine decrescente e quindi in base al nome del componente in ordine crescente, la colonna Quantità ha un simbolo ? e la colonna Componente ha un simbolo ▲?.

Espandi/comprimi gruppi

Figura 54: Espandi/comprimi gruppi

Nelle BOM gerarchiche, ogni sottoinsieme forma un gruppo che può essere compresso o espanso nella griglia BOM.

Per comprimere o espandere un gruppo, effettuare una delle seguenti operazioni:

- Cliccare sull'icona del nodo nella prima colonna della griglia BOM.
- Cliccare il pulsante destro del mouse sulla riga, quindi selezionare **Comprimi** o **Espandi** nel menu contestuale.

È inoltre possibile comprimere o espandere tutti i gruppi o tutti i gruppi corrispondenti allo stesso livello di assieme usando il menu di scelta rapida.

Nota: Lo stato del gruppo di sottoassiemi (compresso o espanso) non è memorizzato nel documento. Quando la tabella BOM viene chiusa e riaperta, tutti i gruppi di sottoassiemi vengono espansi per impostazione predefinita.

Proprietà

Figura 55: Proprietà della distinta dei materiali

La griglia delle proprietà si trova nella parte inferiore del pannello del gestore. Per regolarne le dimensioni, utilizzare il controllo splitter (cursore a doppia freccia), situato tra la griglia e la griglia delle proprietà.

La griglia delle proprietà contiene le proprietà della tabella corrente; titolo, tipo, filtro, modalità di ordinamento e titolo piè di pagina. Se vengono selezionate delle colonne, la griglia visualizzerà anche le loro proprietà comuni alle colonne selezionate. E' possibile modificare contemporaneamente qualsiasi proprietà comune per tutte le colonne selezionate. Le proprietà della colonna modificano il titolo, la visibilità e la larghezza della colonna.

Per i valori numerici, è possibile impostare il tipo di piè di pagina, il posizionamento delle unità e la modalità delle unità (se la colonna possiede unità che possono essere modificate).

Il **posizionamento unità** definisce dove vengono visualizzate le unità: nel titolo della colonna, nella stessa cella del valore oppure in una colonna aggiuntiva dedicata.

La **modalità unità** definisce come ottenere l'unità visualizzata; l'unità migliore per tutti i valori, l'unità migliore per visualizzare un determinato valore nella cella o un'unità specificata. È possibile scegliere la **modalità Unità** per le proprietà di distanza, area, volume o massa.

21.4.10.4 Filtro

È possibile aggiungere un filtro personalizzato alla BOM corrente. Per applicare un filtro, effettuare una delle seguenti operazioni:

- Utilizzare la proprietà **Filtro** nella sezione **Proprietà della distinta dei materiali** della **Griglia delle proprietà**.
- Utilizzare il comando **Imposta filtro** dal **menu Principale**.

In entrambi i casi, viene visualizzata la finestra di dialogo *Configuratore filtro*:

Figura 56: Configuratore filtro

Ogni riga di questa finestra di dialogo rappresenta una condizione elementare. È anche possibile aggiungere gruppi per combinare diverse condizioni elementari e/o gruppi. I gruppi consentono la creazione di filtri complessi.

Aggiungere una nuova condizione elementare

Eseguire le seguenti operazioni:

1 Se applicabile, scegliere la categoria richiesta per limitare l'elenco di proprietà a quella particolare categoria.

2 Selezionare la proprietà desiderata dall'elenco delle proprietà. Se selezionata, viene aggiunta automaticamente una condizione per la proprietà.

3 Se applicabile, il tipo di condizione richiesto:
   - Non uguale a
   - Minore di
   - Maggiore di
   - Non minore di
   - Non maggiore di

4 Impostare il valore di destinazione. Per i valori di testo, è possibile selezionare il valore di destinazione dall'elenco a discesa di tutti i valori possibili. Per valori numerici, è possibile specificare l'unità del valore di destinazione. Il valore verrà convertito automaticamente nell'unità del documento.

Rimuovere una o più condizioni

Eseguire le seguenti operazioni:

1 Selezionare le condizioni da rimuovere. È possibile utilizzare il tasto Ctrl per selezionare diverse condizioni e il tasto Maiusc per selezionare una gamma di condizioni.

2 Cliccare sullo strumento **Elimina** (✕),

   o cliccare il pulsante destro del mouse sulla riga che si desidera eliminare e selezionare **Rimuovi condizione** nel menu contestuale.

Aggiungere un nuovo gruppo di condizioni

Eseguire le seguenti operazioni:

1 Cliccare sullo strumento **Aggiungi** (+).

2 Selezionare **Aggiungi "Tutt i" gruppi** per aggiungere un nuovo gruppo "Tutti i",
o selezionare **Aggiungi "Qualsiasi tra" i gruppi** per aggiungere un nuovo gruppo "Qualsiasi tra".

3 Aggiungere le condizioni necessarie.

Raggruppamento delle condizioni selezionate

Eseguire le seguenti operazioni:

1 Seleziona le condizioni da rimuovere. È possibile utilizzare il tasto Ctrl per selezionare diverse condizioni e il tasto Maiusc per selezionare una gamma di condizioni.

2 Cliccare il pulsante destro del mouse sulla riga che si desidera raggruppare.

3 Selezionare **Forma gruppo "Tutti i" da selezionati** per formare un nuovo gruppo "Tutti i",
oppure selezionare **Forma gruppo "Qualsiasi tra" da selezionati** per creare un nuovo gruppo "Qualsiasi tra".

Per applicare il filtro, fare clic su **OK**.

21.4.10.5 Filtri colonna

I filtri colonne consentono di impostare le condizioni su cui includere le righe nella tabella. Ciascun filtro colonna imposta una condizione che i valori di colonna devono soddisfare per essere inclusi. Ad esempio, è possibile impostare un filtro colonna in modo da includere solo parti con quantità superiori a un certo numero o parti il cui nome contiene **Ugello**. Una colonna non può avere più di un filtro. Tuttavia, è possibile impostare contemporaneamente i filtri per tutte le colonne necessarie. Le colonne con filtri attivi sono contrassegnate da un'icona di filtro () nel titolo.

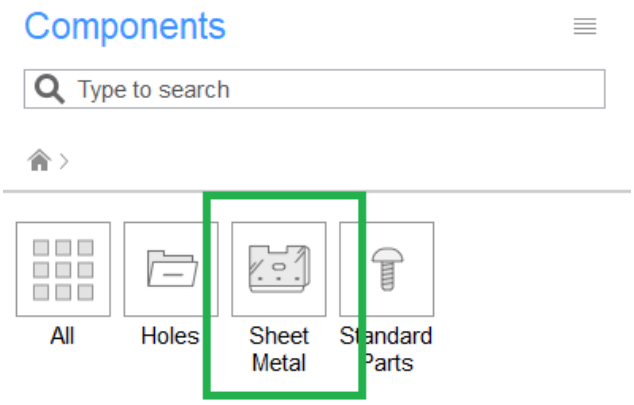


Figura 57: Esempio di colonne con filtri attivi

Per aggiungere un filtro colonna, cliccare il pulsante destro del mouse sul titolo della colonna e selezionare **Imposta filtro colonna** nel menu contestuale.

Viene visualizzata una finestra di dialogo *Filtro colonna*.

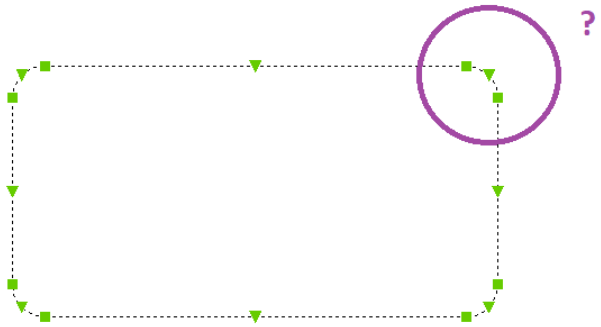


Figura 58: Filtro colonna

Selezionare il tipo di filtro richiesto. Successivamente, è necessario impostare i valori di destinazione per il filtro, a seconda del tipo di filtro e del contenuto della colonna.

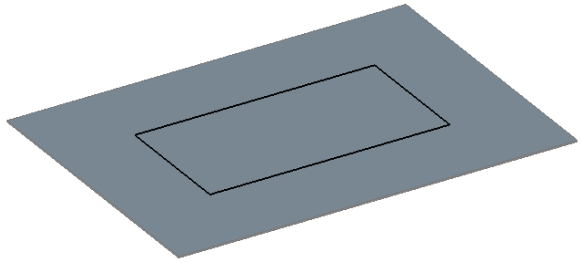


Figura 59: Valori di destinazione per il filtro

Per le colonne con contenuto numerico, è possibile impostare l'unità per il valore di destinazione. Il valore di destinazione verrà convertito automaticamente nell'unità interna del documento.

Per i filtri di testo, è possibile scegliere tra le opzioni **ignora maiuscole e minuscole** e **corrispondenza maiuscole e minuscole**. I filtri **incluso in** e **non incluso in** mostrano l'elenco di tutti i valori possibili per questa colonna (rispetto ai filtri già applicati). Spuntare i valori che si desidera includere o escludere di conseguenza. È anche possibile spuntare tutti i valori, deselezionare tutti i valori o invertire la selezione corrente utilizzando i pulsanti corrispondenti sulle barre degli strumenti o i comandi del menu contestuale.

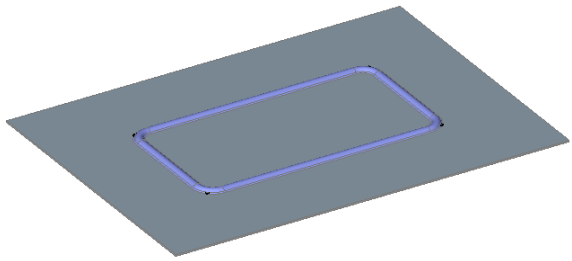


Figura 60: Filtro colonna

Per applicare il filtro colonna, cliccare su **OK**.

Per rimuovere un filtro colonna, cliccare il pulsante destro del mouse sul titolo della colonna e selezionare **Rimuovi filtro colonna** dal menu contestuale.

Per maggiori dettagli, consultare la sezione Filtro in Distinta dei Materiali.

21.4.10.6 Ordina

Le tabelle BOM supportano diverse modalità di ordinamento (per ulteriori dettagli, vedere la sezione Ordinare in Distinta dei Materiali).

Per passare da una modalità all'altra, utilizzare la proprietà **Modalità di ordinamento** nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**. Utilizzare il menu a discesa **Modalità ordinamento** o fare doppio clic sulle etichette di colonna nella **Tabella** per passare dall'ordine crescente a quello decrescente o naturale per la rispettiva colonna.

21.4.10.7 Unità Colonne

È possibile modificare il **posizionamento Unità** e la **modalità Unità** per la maggior parte delle proprietà numeriche con unità. Selezionare le colonne richieste nella **Griglia** e utilizzare la sezione **Proprietà Colonna** della **griglia delle Proprietà** per modificare la **posizione Unità** e la **modalità Unità**.

Per maggiori dettagli, vedere la sezione Unità Colonne in Distinta dei Materiali.

Vedi anche: Modificare le unità e la modalità di posizionamento delle unità per la colonna.

21.4.10.8 Colonne valutate

Una BOM supporta le colonne valutate. Esistono 2 tipi di colonne valutate: formula e colonne modello. Per ulteriori dettagli sui campi formula, ad esempio la sintassi delle espressioni, vedere la sezione Colonne Calutate in Distinta dei Materiali.

Vedi anche: Aggiungere una colonna valutata alla pagina 1632.

Colonna Formula

Per aggiungere una colonna formula, eseguire le seguenti operazioni:

1 Selezionare la categoria **Colonne valutate** utilizzando il **Selettore categoria**.

2 Selezionare **Colonna Formula** nel **selettore Colonna**.

3 Utilizzare **Inserisci colonna dopo** o **Inserisci colonna prima** degli strumenti.

In alternativa, è possibile utilizzare l'opzione **Inserisci colonna formula** dal **menu Principale**.

In entrambi i casi, viene visualizzata una finestra di dialogo *Configuratore espressione*:

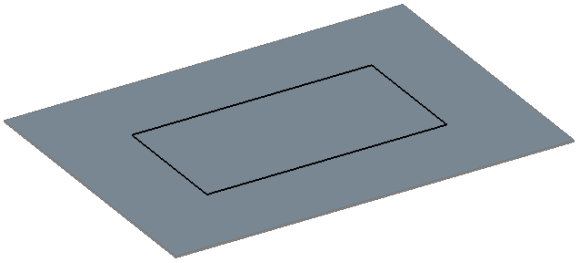


Figura 61: Configuratore Espressione

Questa finestra di dialogo consente di configurare il titolo, l'unità e l'espressione della colonna formula. Visualizza anche un'anteprima dei risultati della formula. Il campo espressione supporta il completamento automatico.

Per aggiungere un riferimento a una colonna o una proprietà all'espressione nella posizione del cursore, selezionare la proprietà richiesta dall'elenco delle proprietà. È inoltre possibile cliccare sullo strumento **Aggiungi riferimento colonna** (ABC) e selezionare la proprietà dal menu oppure cliccare sullo strumento **Aggiungi funzione** (fx) per selezionare la funzione desiderata.

Per aggiungere una colonna formula alla tabella BOM, fare clic su **OK**.

Colonna Modello

Per aggiungere una colonna modello, eseguire le seguenti operazioni:

1 Selezionare la categoria *<Colonne valutate>* utilizzando il **Selettore categoria**.

2 Selezionare la *<Colonna Modello>* nel **selettore Colonna**.

3 Utilizzare **Inserisci colonna dopo** o **Inserisci colonna prima** degli strumenti.

In alternativa, è possibile utilizzare l'opzione **Inserisci colonna modello** dal **menu Principale**.

In entrambi i casi, viene visualizzata una finestra di dialogo *Configuratore modello*.

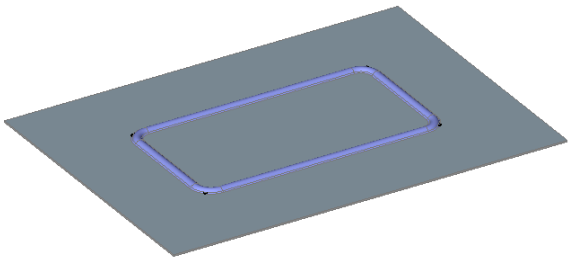


Figura 62: Configuratore di modelli

Questa finestra di dialogo consente di configurare il titolo, l'espressione e, se necessario, l'unità per la colonna modello. Inoltre, visualizza un'anteprima dei risultati. Il campo espressione supporta il completamento automatico.

Le colonne modello possono contenere più di una sottoespressione modello. In tal caso, vengono valutate dalla prima all'ultima e viene utilizzato il rendimento della prima sottoespressione in risultati non vuoti.
Il **configuratore modelli** consente di aggiungere, eliminare e passare da una sottoespressione modello all'altra utilizzando la barra degli strumenti. Consente inoltre di modificare l'ordine delle sottoespressioni modello.

Nota: Nel campo espressione, tutte le sottoespressioni vengono visualizzate contemporaneamente, dove ciascuna sottoespressione è racchiusa tra parentesi graffe {}.

Per aggiungere una sottoespressione, cliccare su .

Per eliminare la sottoespressione corrente, cliccare su .

Nota: Quando l'ultima sottoespressione viene rimossa, verrà sostituita con una vuota.

Per modificare la sottoespressione corrente, selezionare il numero della sottoespressione richiesta dall'elenco a discesa.

Usare e per modificare l'ordine delle sottoespressioni.

La sottoespressione attualmente selezionata può essere configurata utilizzando la sezione **Modello**. Ogni colonna in questa sezione rappresenta un elemento dell'espressione modello che è possibile modificare.

I tipi di elementi del modello sono:

- **Riferimento colonne**: rappresenta un riferimento a una colonna BOM o a una proprietà. Per aggiungere un riferimento colonna, selezionare la colonna desiderata dall'elenco delle proprietà. Verrà aggiunto automaticamente dopo l'ultima colonna selezionata o, se non è selezionata alcuna colonna, dopo l'ultima colonna. Se necessario, verrà aggiunto automaticamente un **Delimitatore** davanti al **riferimento colonna**.
- **Delimitatore**: rappresenta un separatore tra 2 elementi. Il testo delimitatore apparirà nel risultato finale solo se il testo su entrambi i lati non è vuoto. Ad esempio, se si dispone di riferimenti colonna PartNumber e Length con il delimitatore "X" tra di essi, la "X" verrà aggiunta al testo finale solo se partNumber e Length non sono vuoti. Nella maggior parte dei casi, i delimitatori vengono aggiunti automaticamente. Tuttavia, è possibile aggiungere delimitatori utilizzando il pulsante e selezionando **Aggiungi delimitatore** dall'elenco a discesa. In alternativa, è possibile aggiungere un delimitatore dal menu contestuale.
- **Testo**: rappresenta un testo personalizzato. Puoi aggiungere un testo personalizzato cliccando su e selezionando **Aggiungi testo** dall'elenco a discesa. In alternativa, è possibile aggiungere un testo personalizzato dal menu contestuale.
- **Testo valutato**: rappresenta un elemento che deve essere aggiunto all'espressione finale così com'è. Il testo valutato può contenere funzioni o altri simboli speciali che modificheranno il risultato. È possibile aggiungere un testo valutato utilizzando l'icona e selezionando **Aggiungi espressione** dall'elenco a discesa. In alternativa, è possibile aggiungere un testo valutato dal menu contestuale.

In qualsiasi momento, è possibile modificare il contenuto di qualsiasi elemento nel modello. Puoi anche eliminare gli elementi selezionandoli e utilizzando il pulsante oppure scegliendo **Elimina colonna** nel menu contestuale. Per spostare un elemento, selezionare una colonna nel modello, quindi trascinarla nella posizione desiderata. Qualsiasi modifica del modello verrà riflessa nel campo dell'espressione. Analogamente, qualsiasi modifica nel campo dell'espressione verrà riflessa nell'area del modello.

Per aggiungere una colonna modello alla tabella BOM, cliccare su **OK**.

21.4.10.9 Riga piè di pagina

Una tabella BOM può includere una riga di piè di pagina. Per aggiungere una riga di piè di pagina, specificare il tipo di piè di pagina per una o più colonne. Per impostazione predefinita, il titolo della riga del piè di pagina è *TOTALE*:. Modificare il titolo del piè di pagina con la proprietà **Titolo piè di pagina** nella sezione **Proprietà della distinta dei materiali** nella **griglia delle Proprietà**.

Per ulteriori informazioni su una riga di piè di pagina in una BOM, vedere la sezione Piè di Pagina in Distinta dei Materiali.

Vedi anche: Aggiungere un piè di pagina per la colonna.

21.4.10.1 Colonne aggregate 0

Nelle tabelle BOM di livello superiore e inferiore, è possibile assegnare quasi tutte le colonne con una funzione di aggregazione. Le funzioni di aggregazione raggruppano i valori di più righe nella BOM (che differiscono solo in tali valori) in un unico valore. Ad esempio, le funzioni di aggregazione possono essere utilizzate per calcolare la lunghezza totale dei tubi rispetto alle loro dimensioni e spessore o per concatenare tutte le dimensioni di bulloni e dadi utilizzati nell'assieme in un'unica cella.

Per ulteriori dettagli, vedere la sezione Colonne in Distinta dei Materiali.

Per impostare una funzione di aggregazione, effettuare una delle seguenti operazioni:

- Cliccare il pulsante destro del mouse sul titolo della colonna e selezionare la funzione dal sottomenu **Funzione di aggregazione**.
- Selezionare la colonna e modificare la funzione di aggregazione nella proprietà della colonna corrispondente.

Le colonne aggregate sono contrassegnate con il simbolo ∑ nel titolo.

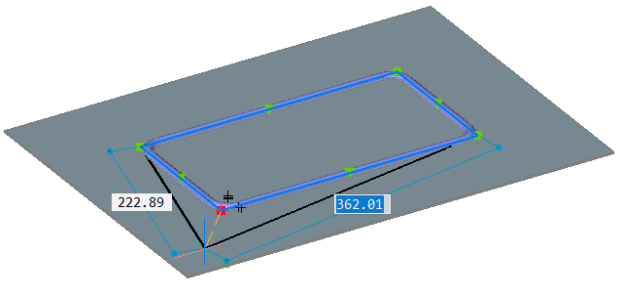


Figura 63: Esempio di colonne aggregate

21.4.10.1 Modelli
1

È possibile salvare le configurazioni BOM come modello. Utilizzare il gestore BOM per creare una nuova tabella da un modello e sostituire la configurazione della tabella corrente con una proveniente da un modello. È possibile salvare la configurazione come modello in qualsiasi momento. È possibile modificare una configurazione modello caricata con i normali strumenti e salvarla come un nuovo modello.

Per ulteriori informazioni sui modelli BOM, vedere la sezione Modelli in Distinta dei Materiali.

21.4.10.1 Ricerca
2

Figura 64: Cerca

Il gestore BOM dispone di un campo di ricerca che consente di cercare un particolare valore nella tabella. È inoltre possibile utilizzare i caratteri jolly nel campo di ricerca. Per impostazione predefinita, BricsCAD esegue la ricerca nell'intera tabella. Per limitare l'area di ricerca, selezionare le colonne richieste. Se il valore viene trovato, il cursore si sposta su quella cella. Se non vengono selezionate colonne, viene selezionata la riga. La **Griglia** scorre automaticamente per rendere visibile la cella. Per trovare un altro valore, utilizzare i pulsanti o . È inoltre possibile utilizzare **F3** o **Ctrl+F3**. Per selezionare tutti i risultati trovati, utilizzare il pulsante .

Per impostazione predefinita, viene cercato il testo nel campo di ricerca. Tuttavia, puoi anche utilizzare le **caratteri jolly**.

Sono supportati i seguenti simboli speciali:

Tabella 19: Caratteri Jolly

Nome	Caratteri	Descrizione
Citazione indietro	`	Esclude il carattere successivo (lo considera letteralmente).
Virgola	,	Separa i modelli.

Tabella 19: Caratteri Jolly

Nome	Caratteri	Descrizione
Asterisco	*	Corrisponde a qualsiasi sequenza di caratteri, inclusa una sequenza nulla
Punto interrogativo	?	Corrisponde a un singolo carattere.
Periodo	.	Corrisponde a un singolo carattere non alfanumerico.
Chiocciola	@	Corrisponde a un singolo carattere alfabetico.
Cancelletto	#	Corrisponde a un singolo carattere numerico.
Tilde	~	Se il primo carattere in un modello corrisponde a tutto tranne che al modello.
Parentesi	[...]	Corrisponde a un singolo carattere racchiuso.
Tilde tra parantesi	[~...]	Corrisponde a un singolo carattere non racchiuso.
Trattino	-	Tra parentesi, corrisponde a un singolo carattere nell'intervallo specificato.

Per aprire il campo di ricerca, utilizzare lo strumento **Cerca** sulla barra degli strumenti o utilizzare **Ctrl + F**.

21.4.10.1 Tabella associata 3

Ogni tabella BOM nel gestore può avere una tabella associata nel documento, la quale si aggiorna se la configurazione della tabella BOM viene modificata. Una tabella BOM può avere solo una tabella associata. Tuttavia, è possibile creare tabelle BOM che non hanno una tabella associata nel documento.

Per creare una tabella associata, utilizzare **Inserisci/Aggiorna tabella**. Se la tabella BOM è stata creata con *BMBOM*, la tabella associata viene creata automaticamente.

È possibile individuare la tabella associata nel documento con **Tabella associata - Zoom A** o **Tabella associata - Seleziona**. Se necessario, il layout attivo passa al layout della tabella associata.

È possibile eliminare la tabella associata con **Tabella associata - Elimina**. In alternativa, è possibile scollegare la tabella associata dalla tabella BOM con **Tabella associata - Stacca**. Dopo che è stata scollegata, la tabella associata perde il collegamento alla tabella BOM e non si aggiornerà con BOM.

21.4.10.1 Aggiornare la BOM
4

Da BricsCAD V20.2, il gestore BOM aggiorna automaticamente il contenuto della tabella nel pannello, quando l'assieme viene modificato o aggiornato. Per gli assiemi di grandi dimensioni, questo potrebbe essere fastidioso. Il gestore BOM aggiorna automaticamente la tabella quando è attivo.

Per evitare l'aggiornamento automatico, chiudere il gestore BOM per rendere inattivo il pannello Gestire BOM.

Usare **Aggiorna** per aggiornare manualmente il contenuto del gestore BOM.

Nota: La tabella associata non viene aggiornata automaticamente quando l'assieme viene modificato.

Utilizzare **Inserisci/Aggiorna tabella** per aggiornare la tabella associata.

21.4.10.1 Aggiungere una nuova distinta dei materiali
5

1 Usare il sottomenu **Nuova BOM** del **Menu principale**.

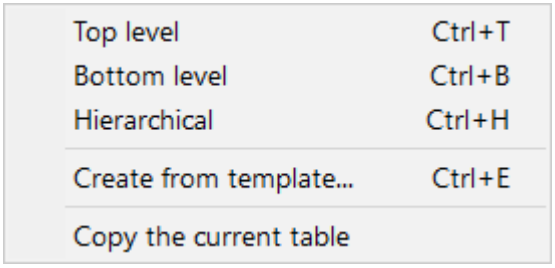


Figura 65: Nuovo sottomenu BOM

2 Selezionare il tipo di BOM o selezionare **Crea da modello** per scegliere il file modello per una BOM.

21.4.10.1 Selezionare la BOM corrente
6

Utilizzare il **selettore BOM** per selezionare una distinta dei materiali esistente.

Nota: La tabella si aggiornerà automaticamente allo stato corrente dell'assieme e della tabella associata nel disegno (se esiste).

21.4.10.1 Modificare il tipo di BOM corrente
7

Modificare la proprietà **Tipe** nella sezione **Proprietà della distinta dei materiali** della **griglia Proprietà**.

Bill of materials properties	
Title	Bill of material <NAME>
Type	Hierarchical
Property set	Top level
Filter	Bottom level
Sorting mode	Hierarchical
Counting mode	Automatic order
Max level	Count instances in parent component
Footer title	All
Fixed corner	TOTAL:
CSV delimiter	Top left
Warning messages	; (Semicolon)
	On

Figura 66: Proprietà della distinta dei materiali

21.4.10.1 Modificare il gruppo di proprietà della BOM corrente

8

Modificare la proprietà **Gruppo di proprietà** nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**.

Bill of materials properties	
Title	Bill of material <NAME>
Type	Hierarchical
Property set	Mechanical only
Filter	Mechanical only
Sorting mode	All except coordinates
Counting mode	All properties
Max level	Count instances in parent component
Footer title	All
Fixed corner	TOTAL:
CSV delimiter	Top left
Warning messages	; (Semicolon)
	On

Figura 67: Tutte le proprietà

21.4.10.1 Modificare la modalità di conteggio di una BOM gerarchica

9

Modificare la proprietà **Modalità di conteggio** nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**.

Bill of materials properties	
Title	Bill of material <NAME>
Type	Hierarchical
Property set	Mechanical only
Filter	
Sorting mode	Automatic order
Counting mode	Count instances in parent component
Max level	Count all instances
Footer title	Count instances in parent component
Fixed corner	TOTAL:
CSV delimiter	Top left
Warning messages	; (Semicolon)
	On

Figura 68: Proprietà della distinta dei materiali

21.4.10.2 Modificare la modalità di raggruppamento della BOM di livello superiore o inferiore 0

Modificare la proprietà Modalità raggruppamento nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**.

Figura 69: Proprietà della distinta dei materiali

21.4.10.2 Impostare un filtro personalizzato per la BOM corrente 1

1 Selezionare l'opzione **Imposta filtro** dal **menu principale**.

Viene visualizzata una finestra di dialogo *Configuratore filtro*.

2 Configurare il filtro utilizzando lo strumento di dialogo.

Vedere Filtro alla pagina 1619 per i dettagli.

3 Cliccare su **OK** per applicare il filtro.

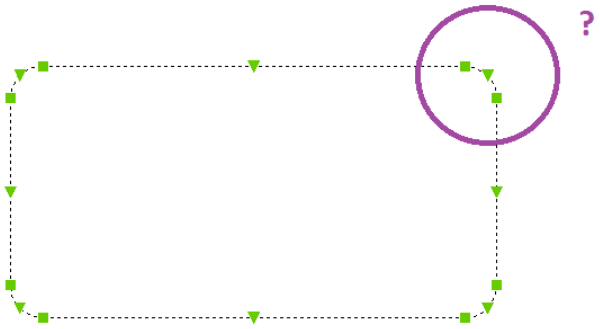


Figura 70: Configuratore filtro

21.4.10.2 Aggiungere nuove colonne alla BOM corrente 2

1 Se applicabile, utilizzare il selettore **Categoria** per selezionare la categoria delle proprietà da aggiungere.

Figura 71: Tutte le colonne

2 Selezionare la proprietà da aggiungere dall'elenco delle colonne disponibili. Questo elenco contiene tutte le colonne della categoria selezionata che non sono ancora state aggiunte alla BOM. Iniziando a digitare il nome del campo, BricsCAD suggerisce automaticamente i nomi. Cliccare per selezionarla dall'elenco.

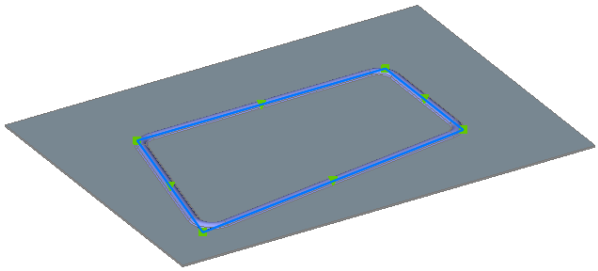


Figura 72: Descrizione componente

3 Se applicabile, selezionare la colonna nella BOM accanto alla quale si desidera posizionare la nuova colonna.

4 Cliccare su **Inserisci colonna dopo** ( ) per posizionare la nuova colonna dopo quella selezionata o **Inserisci colonna prima** ( ) per posizionare la nuova colonna prima di quella selezionata.

Se non è stata selezionata alcuna colonna, la nuova colonna viene posizionata rispettivamente in ultima o prima posizione.

21.4.10.2 Aggiungere una colonna valutata

3

1 Selezionare la categoria **Colonne valutate** utilizzando il **Selettore categoria**.

2 Selezionare **Colonna Formula** per creare una colonna formula oppure selezionare **Colonna Modello** per creare una colonna modello.

3 Cliccare su **Inserisci colonna dopo** ( ) per posizionare la nuova colonna dopo quella selezionata o **Inserisci colonna prima** ( ) per posizionare la nuova colonna prima di quella selezionata.

Se non è stata selezionata alcuna colonna, la nuova colonna viene posizionata rispettivamente in ultima o prima posizione.

Viene visualizzata una finestra di dialogo per configurare la colonna valutata.

4 Cambiare il **Titolo** della colonna.

5 Se applicabile, cambiare l'**Unità**.

6 Configurare l'espressione utilizzando gli strumenti di dialogo.

7 Cliccare su **OK** per aggiungere la colonna alla tabella BOM.

21.4.10.2 Aggiungere una colonna modello da colonne selezionate

4

1 Selezionare le colonne da combinare in una colonna modello. Utilizzare il tasto Ctrl per selezionare più colonne o il tasto Maiusc per selezionare un intervallo di colonne.

2 Cliccare il pulsante destro del mouse sul titolo di una delle colonne selezionate.

3 Selezionare **Aggiungi colonna modello da selezione** dal menu contestuale.

Viene visualizzata una finestra di dialogo per configurare la colonna valutata.

4 Cambiare il **Titolo** della colonna.

5 Se applicabile, cambiare l'**Unità**.

6 Configurare l'espressione utilizzando gli strumenti di dialogo.

7 Cliccare su **OK** per aggiungere la colonna alla tabella BOM.

8 Se applicabile, rimuovere o nascondere le colonne che non sono più necessarie.

21.4.10.2 Modificare una colonna valutata

5

1 Cliccare il pulsante destro del mouse sul titolo della colonna nella **griglia BOM**.

2 Selezionare **Modifica espressione colonna** dal menu contestuale.

Viene visualizzata una finestra di dialogo per configurare la colonna valutata.

3 Se applicabile, modificare il titolo, l'unità e l'espressione della colonna.

4 Cliccare su **OK** per applicare le modifiche.

21.4.10.2 Rimuovere una colonna dalla BOM corrente

6

1 Selezionare la colonna nella griglia.

2 Cliccare sullo strumento **Elimina la colonna** ( ),

o premere **Elimina**.

21.4.10.2 Modificar il titolo della BOM

7

Utilizzare **Titolo BOM** per modificare il titolo,

o utilizzare la proprietà **Titolo** nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**.

Puoi utilizzare il segnaposto *<NOME>* per fare riferimento al nome dell'assieme.

21.4.10.2 Modificare il titolo della riga di piè di pagina

8

Modificare la proprietà **Tipo di piè di pagina** nella sezione **Proprietà della distinta dei materiali** della **griglia delle Proprietà**.

21.4.10.2 Aggiungere un prefisso o un suffisso ai numeri

9

1 Selezionare la colonna **Numero**.

2 Selezionare la proprietà **Prefisso numerico** nella sezione **Proprietà colonna**.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

Figura 73: Proprietà colonna

3 Se necessario, immettere la nuova stringa di prefisso.

4 Selezionare la proprietà **Suffisso numerico** nella sezione **Proprietà colonna**.

5 Se necessario, immettere la nuova stringa di suffisso.

21.4.10.3 Modificare il delimitatore tra numeri in una BOM gerarchica 0

1 Selezionare la colonna **Numero**.

2 Selezionare la proprietà **Delimitatore** nella sezione **Proprietà colonna**.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	.
Format	
Visible	Yes
Width	Auto
Filter	No filter

Figura 74: Proprietà colonna

3 Immettere la nuova stringa delimitatore.

21.4.10.3 Modificare il tipo di numero nella BOM gerarchica 1

1 Selezionare la colonna **Numero**.

2 Selezionare la proprietà **Tipo numero** nella sezione **Proprietà colonna**.

Column properties	
Title	No.
Category	BOM Standard
Role	Number
Number prefix	A
Number suffix	
Number type	Full
Delimiter	Full
Format	Short Continuous
Visible	Yes
Width	Auto
Filter	No filter

Figura 75: Proprietà colonna

3 Selezionare il tipo di numero richiesto.

21.4.10.3 Modificare il titolo della colonna

2

1 Selezionare la colonna nella **griglia BOM**.

2 Selezionare la proprietà **Titolo** nella sezione **Proprietà colonna**.

Figura 76: Proprietà colonna

3 Modificare il valore della proprietà **Titolo** con il nuovo titolo della colonna.

21.4.10.3 Modificare il formato della colonna

3

1 Selezionare le colonne.

2 Cliccare il pulsante destro del mouse sul titolo di una colonna selezionata.

3 Seleziona **Formato** nel menu contestuale.

Viene visualizzata la finestra di dialogo *Formato*.

Figura 77: Formato

4 Definire il formato desiderato.

5 Cliccare su **OK** per applicare le modifiche.

21.4.10.3 Modificare le unità e le modalità di posizionamento delle unità per una colonna

4

1 Selezionare le colonne.

2 Se applicabile, modificare la proprietà **Posizionamento unità** nella sezione **Proprietà colonna**.

3 Se applicabile, modificare la proprietà **Modalità unità** nella sezione **Proprietà colonna**.

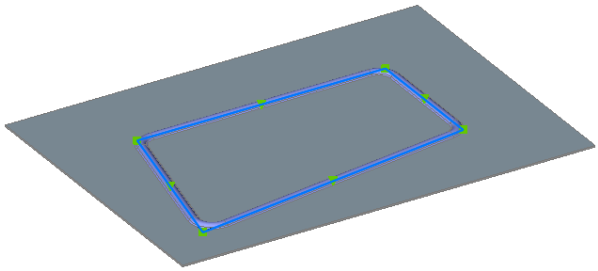


Figura 78: Proprietà colonna

4 Se applicabile, modificare il **formato titolo**.
   - Usare *<Titolo>* per aggiungere il titolo della colonna.
   - Utilizzare *<Unità>* per aggiungere l'unità della colonna.
   - Utilizzare *\n* per aggiungere un'interruzione di riga.

21.4.10.3 Aggiungere un piè di pagina a una colonna

5

1 Selezionare le colonne.

2 Utilizzare il sottomenu **Piè di pagina** del **menu Principale** per selezionare il tipo di valore del piè di pagina,

oppure cliccare il pulsante destro del mouse sull'etichetta della colonna e quindi selezionare il sottomenu **Piè di pagina**.
Sono

disponibili i seguenti tipi:
   - **Nessuna funzione** (la colonna verrà rimossa dal piè di pagina. Se non vi è alcuna colonna con un altro tipo di piè di pagina, la riga del piè di pagina verrà rimossa completamente)
   - **Somma**
   - **Valore medio**
   - **Valore minimo**
   - **Valore massimo**

Come alternativa:

1 Selezionare le colonne.

2 Nella **griglia delle Proprietà**, selezionare la proprietà **Tipo di piè di pagina** e scegliere il tipo di piè di pagina.

Figura 79: Proprietà colonna

Nota: Per passare da **Somma** a **Nessuna funzione**, cliccare sul pulsante **Piè di pagina** ( ).

21.4.10.3 Salvare la configurazione BOM come file modello 6

1 Scegliere **Salva come modello** () nel menu principale ().

Viene visualizzata la finestra di dialogo per selezionare il nome e la posizione per un file di modello.

Figura 80: Seleziona il file per salvare il template della tabella BOM

2 Specificare il nome e la posizione del file modello.

3 Cliccare su **Salva**.

21.4.10.3 Esportare la tabella corrente in CSV 7

1 Se applicabile, regolare la proprietà **Delimitatore CSV** della tabella.

Figura 81: Delimitatore CSV

2 Scegliere **Esporta in CSV** nel menu principale ().

Viene visualizzata la finestra di dialogo per selezionare il nome e la posizione per un file CSV.

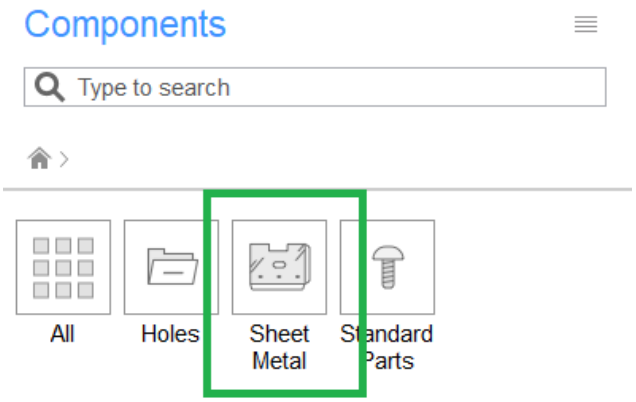


Figura 82: Selezionare il file per esportare la tabella BOM

3 Specificare il nome e la posizione del file CSV.

4 Cliccare su **Salva**.

21.4.11 Distinta dei Materiali

21.4.11.1 Panoramica

È possibile utilizzare i comandi *BMBOM*, *BMBOMPANELOPEN* e *BMBMPANELCLOSE*.

Una Distinte dei Materiali (BOM) è un elenco di componenti meccanici unici utilizzati in un assieme. In BricsCAD, l'elenco è rappresentato come entità tabella (vedere Tabelle) e può essere collocato in qualsiasi punto del disegno. Ogni entità ha il proprio numero di seriale (che inizia con 1), il nome e il numero di occorrenze.

È possibile visualizzare e configurare le Distinte dei Materiali nel Gestore BOM.

21.4.11.2 Tipi di Distinta dei Materiali

Ci sono 3 tipi di BOM in BricsCAD:

- **Livello superiore**

  Contiene solo componenti del livello superiore. I componenti di livello superiore sono componenti inseriti direttamente nell'assieme principale. Questo tipo di BOM elenca solo i componenti principali, senza dettagli sui sottocomponenti.
- **Livello inferiore**

  Contiene solo componenti (parti) di livello inferiore. I componenti di livello inferiore sono parti che non contengono altre parti o sottoassiemi. Questo tipo di BOM elenca tutte le parti nell'assieme principale, indipendentemente dal componente a cui appartengono le parti.
- **Gerarchico**

  Contiene tutti i componenti con i relativi sottocomponenti elencati direttamente di seguito. I numeri di serie dei sottocomponenti sono una sequenza di numeri separati da punti (ad esempio,"1.2.3"). Questo tipo di BOM mostra la struttura gerarchica dell'assieme.

Nota: La proprietà **stato BOM** controlla quali parti vengono incluse in una Distinta dei Materiali. Il livello di un componente in una tabella BOM viene impostato rispetto allo **stato BOM** di tutti i componenti correlati. Ad esempio, un componente con stato BOM Terminale viene sempre considerato di livello inferiore, anche se contiene altre parti o sottoassiemi.

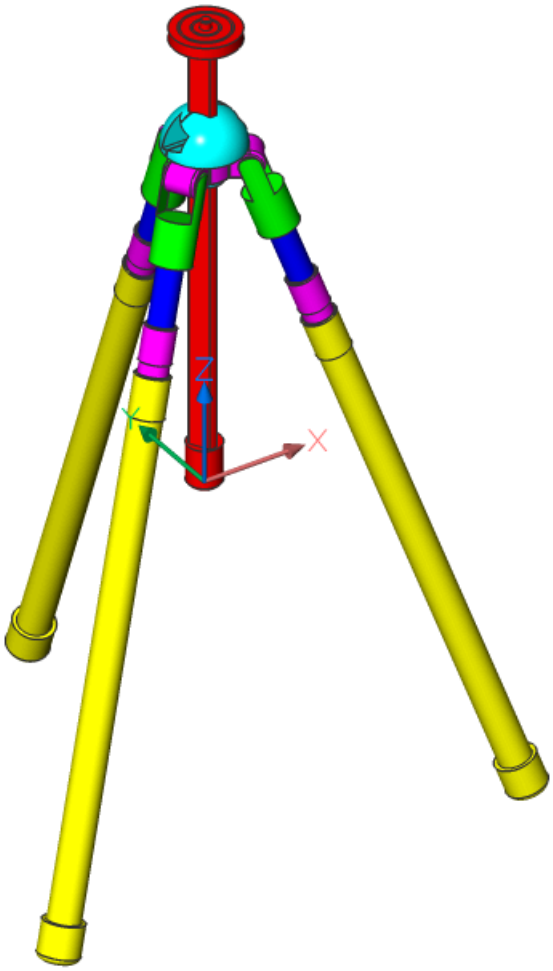


Figura 83: Esempio: Assieme di un treppiede

Tabella 20: Tipi di BOM

Tipo	Distinta dei Materiali
Livello superiore	Bill of Materials tripod_main_assy No. | Component | Quantity 1 | tripod_axis_assy | 1 2 | tripod_foot_assy | 3 **Figura 84:** Livello superiore

Tabella 20: Tipi di BOM

<table>
<tr><th>Tipo</th><th>Distinta dei Materiali</th></tr>
<tr><td>Livello inferiore</td><td>
<table>
<tr><th colspan="3">Bill of Materials tripod_main_assy</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>lower_tube_prt</td><td>3</td></tr>
<tr><td>2</td><td>middle_tube_prt</td><td>3</td></tr>
<tr><td>3</td><td>tripod_axis_prt</td><td>1</td></tr>
<tr><td>4</td><td>tripod_bond_prt</td><td>3</td></tr>
<tr><td>5</td><td>tripod_crosspiece_prt</td><td>1</td></tr>
<tr><td>6</td><td>tripod_sphere_prt</td><td>1</td></tr>
<tr><td>7</td><td>upper_tube_prt</td><td>3</td></tr>
</table>
Figura 85: Livello inferiore
</td></tr>
<tr><td>Gerarchia</td><td>
<table>
<tr><th colspan="3">Bill of Materials tripod_main_assy</th></tr>
<tr><th>No.</th><th>Component</th><th>Quantity</th></tr>
<tr><td>1</td><td>tripod_axis_assy</td><td>1</td></tr>
<tr><td>1.1</td><td>tripod_axis_prt</td><td>1</td></tr>
<tr><td>1.2</td><td>tripod_crosspiece_prt</td><td>1</td></tr>
<tr><td>1.3</td><td>tripod_sphere_prt</td><td>1</td></tr>
<tr><td>2</td><td>tripod_foot_assy</td><td>3</td></tr>
<tr><td>2.1</td><td>bond_assy</td><td>3</td></tr>
<tr><td>2.1.1</td><td>tripod_bond_prt</td><td>3</td></tr>
<tr><td>2.2</td><td>tubes_assy</td><td>3</td></tr>
<tr><td>2.2.1</td><td>lower_tube_prt</td><td>3</td></tr>
<tr><td>2.2.2</td><td>middle_tube_prt</td><td>3</td></tr>
<tr><td>2.2.3</td><td>upper_tube_prt</td><td>3</td></tr>
</table>
Figura 86: Gerarchia
</td></tr>
</table>

21.4.11.3 Modalità di conteggio e livello massimo

Per le tabelle delle BOMi gerarchiche, sono supportate 2 modalità di conteggio:

- **Conta tutte le istanze**

  Il numero totale di sottoassiemi e parti nell'intero assieme.

- **Conta le istanze nel componente principale**

  Il numero totale di sottoinsiemi e parti nel componente principale. Il componente principale è considerato come il componente radice.

Ad esempio, l'assieme principale del Treppiede contiene 3 sottoassiemi per i Piedistalli del treppiede. Ogni istanza di un assieme dei Piedistalli contiene 1 assieme Nodo e 1 assieme Tubo.

Se la **modalità di conteggio** è impostata su **Conta tutte le istanze**, il risultato è:

Tabella 21: BOM dell'Assieme del Treppiede (conteggio di tutte le istanze)

#	Componente	Quantità
1	Assieme del treppiede	1
1.1	Assieme del piedistallo del treppiede	3
1.1.1	Assieme	3
1.1.2	Assiemi tubi	3

Se la **modalità di conteggio** è impostata su **Conta istanze nel componente principale**, il risultato è:

Tabella 22: BOM dell'Assieme del Treppiede (conteggio istanze per componente)

#	Componente	Quantità
1	Assieme del treppiede	1
1.1	Assieme del piedistallo del treppiede	3
1.1.1	Assieme	1
1.1.2	Assiemi tubi	1

Il valore predefinito per la **modalità di conteggio** è **Conta le istanze nel componente principale**. È possibile modificare la modalità di conteggio in Gestore BOM o con il comando *BMBOM*.

Nota: Prima di BricsCAD V20.2, tutte le tabelle BOM gerarchiche utilizzavano la modalità **Conta tutte le istanze** per impostazione predefinita. Quando tali tabelle vengono aggiornate in BricsCAD V20.2 o versione successiva, la modalità predefinita viene impostata su **Conta le istanze nel componente principale**. Ciò può comportare quantità diverse per alcune parti. Se necessario, impostare manualmente la modalità di conteggio nel Gestore BOM.

Per impostazione predefinita, le distinte dei materiali gerarchiche mostrano tutti i livelli nel modello. È possibile limitare una BOM gerarchica a un certo livello utilizzando la proprietà **Livello massimo**. Quando il livello massimo è impostato, verranno incluse solo le parti fino a quel livello. Ad esempio, se **Livello massimo** = 2, verranno elencati solo i componenti di livello superiore e i sottocomponenti inclusi direttamente nei componenti di livello superiore.

21.4.11.4 Colonne della BOM

Per impostazione predefinita, la tabella BOM contiene le colonne Numero, Componente e Quantità.

Utilizzando l'opzione Configura del comando *BMBOM* è possibile aggiungere altre colonne, ad esempio Descrizione, Densità, Volume, Massa, Materiale, Spessore (per parti in lamiera) e Parametri (per componenti parametrici).

È possibile aggiungere qualsiasi proprietà incorporata dei componenti, istanze o riferimenti a blocchi alla tabella BOM. Sono supportate anche colonne valutate personalizzate, utilizzando la tecnologia dell'Estrazione Dati.

Bill of Materials tripod_main_assy					
No.	Component	Quantity	Density	Volume	Mass
1	tripod_axis_assy	1	8000.00 kg/m³	330.77 cm³	2.65 kg
1.1	tripod_axis_prt	1	8000.00 kg/m³	194.42 cm³	1.56 kg
1.2	tripod_crosspiece_prt	1	8000.00 kg/m³	60.06 cm³	0.48 kg
1.3	tripod_sphere_prt	1	8000.00 kg/m³	76.29 cm³	0.61 kg
2	tripod_foot_assy	3	8000.00 kg/m³	166.24 cm³	1.33 kg
2.1	bond_assy	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.1.1	tripod_bond_prt	3	8000.00 kg/m³	21.32 cm³	0.17 kg
2.2	tubes_assy	3	8000.00 kg/m³	144.92 cm³	1.16 kg
2.2.1	lower_tube_prt	3	8000.00 kg/m³	62.85 cm³	0.50 kg
2.2.2	middle_tube_prt	3	8000.00 kg/m³	34.83 cm³	0.28 kg
2.2.3	upper_tube_prt	3	8000.00 kg/m³	47.24 cm³	0.38 kg

Figura 87: BOM con densità, volume e massa

Modalità di raggruppamento

Le tabelle BOM elencano solo componenti univoci; più elementi degli stessi componenti vengono raggruppati, in base alla modalità di Raggruppamento.

A partire da BricsCAD V21, sono supportate le seguenti modalità di raggruppamento:

- **Auto** (modalità predefinita)

  Più elementi degli stessi componenti vengono raggruppati in base ai parametri e alle proprietà aggiunte. Ciò significa che più elementi dello stesso componente parametrico sono sempre raggruppati in righe separate nella BOM, rispetto ai relativi parametri, indipendentemente dal fatto che tali parametri vengano aggiunti come colonne alla BOM.

Bill of Materials Bookshelf					
No.	Component	Quantity	L	T	W
1	Frame Board	2	360.0	20.0	200.0
2	Frame Board	2	700.0	20.0	200.0
3	Shelf Board	2	266.7	2.5	200.0
4	Shelf Board	2	466.7	2.5	200.0

  Figura 88: Auto

- **Per componenti e colonne**

  In questa modalità, vengono raggruppati più elementi degli stessi componenti solo rispetto alle proprietà aggiunte. Ciò significa che più elementi dello stesso componente parametrico possono essere raggruppati nella stessa riga anche se i loro parametri sono diversi, a seconda delle colonne aggiunte alla BOM. Tuttavia, le istanze di componenti diversi sono sempre raggruppate in righe diverse. Questa modalità è utile se alcuni parametri corrispondono a stati diversi dei componenti ma non risultano parti diverse.

- **Solo per colonne**

In questa modalità, gli elementi di componenti vengono raggruppati in base alle proprietà aggiunte alla BOM, indipendentemente dal fatto che si tratti di elementi dello stesso componente. Questa modalità è utile per ottenere una tabella di riepilogo, come un elenco di tutti i materiali fisici o un elenco di tubi di tutte le dimensioni.

Colonne visibili e invisibili

Puoi rendere invisibile qualsiasi colonna nella tabella BOM. Le colonne invisibili non vengono visualizzate nella tabella nel documento. Tuttavia, è ancora possibile utilizzare queste colonne nelle formule. Le colonne invisibili vengono visualizzate anche nel Gestore BOM.

Colonne aggregate

A partire da BricsCAD V21, è possibile impostare una funzione di aggregazione su quasi tutte le colonne nelle tabelle BOM di livello superiore o inferiore. Le funzioni aggregate raggruppano i valori di più righe nella BOM (che differiscono solo in tali valori) in un unico valore. Sono supportate le seguenti funzioni di aggregazione:

- **Somma**: Somma i valori raggruppati.
- **Media**: Trova il valore medio dei valori raggruppati.
- **Minimo**: Trova il valore minimo dei valori raggruppati.
- **Massimo**: Trova il valore massimo dei valori raggruppati.
- **Concatena** Tutti i valori univoci del gruppo in ordine crescente.
- **Concatena con conteggio**: Concatena tutti i valori univoci con il numero di conteggi nel gruppo.

Ad esempio, prendere la seguente tabella BOM:

Tabella 23: Tabella BOM

Componente	Larghezza	Lunghezza	Altezza	Quantità
Trave	16	250	10	1
Trave	16	200	10	2
Trave	16	220	20	1
Trave	20	215	20	3
Trave	20	225	20	1

Applica una funzione di aggregazione alla colonna **Length**. I risultati vengono raggruppati nella seguente tabella:

Tabella 24: Tabella BOM

Componente	Larghezza	Altezza	Lunghezza (somma)	Lunghezza (media)	Lunghezza (minima)	Lunghezza (massima)	Lunghezza (concatena)	Lunghezza (concatena con conteggio)
Trave	16	10	650	216.6667	200	250	200; 250	2x200; 250
Trave	16	20	220	220	220	220	220	220
Trave	20	20	870	217,5	215	225	215; 225	3x215; 225

Per le colonne **Concatenato con conteggio**, è possibile:

- Regola la posizione del numero di conteggi (davanti o dietro il valore).
- Aggiungere il delimitatore tra i valori.
- Aggiungere il delimitatore tra i valori e il loro numero di conteggi.
- Definire il suffisso e/o il prefisso per il numero di conteggi.

Se in una colonna valutata viene utilizzata una colonna aggregata, i valori finali vengono utilizzati per impostazione predefinita. Tuttavia, è possibile applicare una funzione di aggregazione a una colonna valutata. In questo caso, i risultati di una colonna valutata vengono calcolati per la prima volta per tutti i valori originali (prima dell'aggregazione), quindi i valori risultanti vengono aggregati.

Colonna numerica e ruoli delle colonne

Una colonna numerica è una colonna speciale per numerare le righe. Assegna automaticamente i numeri a tutte le righe nella tabella BOM, senza vuoti e indipendentemente dalle altre tabelle. Le righe vengono numerate a partire da 1.

Le seguenti impostazioni possono essere modificate per la colonna numerica:

- **Prefisso**: una stringa che viene aggiunta davanti a qualsiasi numero. Ad esempio, si desidera che tutti i numeri inizino con "A".
- **Suffisso**: una stringa aggiunta dopo il numero. Ad esempio, si desidera aggiungere "-1" a tutti i numeri.

Per una BOM gerarchica, sono disponibili le seguenti impostazioni:

- **Tipo di numero**: specifica il formato del numero:
  - **Completo**: i numeri vengono formattati come" 1.2.3".
  - **Breve**: vengono visualizzati solo i numeri per il livello attuale. Viene utilizzato insieme alla colonna Livello, che mostra il livello della parte corrente nell'assieme.
  - **Continuo**: tutti i componenti vengono numerati continuamente a partire da 1, indipendentemente dal loro livello.

- **Delimitatore**: specifica il simbolo per separare i numeri per diversi livelli quando il tipo di numero è impostato su *Completo*.

In generale, la colonna numerica viene utilizzata come origine per annotare il disegno in base al comando *BMBALLOON*. Tuttavia, qualsiasi colonna della tabella può essere utilizzata come origine per le bolle di annotazione, modificandone il ruolo. Sono disponibili i seguenti ruoli di colonna:

- **Numero**: La colonna contiene numeri per le annotazioni.
- **Nome**: La colonna contiene i nomi delle parti per le annotazioni.
- **Quantità**: La colonna contiene le quantità di parti per le annotazioni.
- **Regolare**: La colonna non ha un ruolo speciale.

Solo una colonna può avere un ruolo **Numero**, **Nome** o **Quantità**. Se nessuna colonna ha uno di questi ruoli assegnati, le annotazioni automatiche non sono possibili.

Larghezza colonna

Figura 89: Proprietà colonna

Per impostazione predefinita, la larghezza di ciascuna colonna viene impostata automaticamente, in modo che il contenuto si adatti senza interruzioni di riga aggiuntive. Tuttavia, è possibile impostare una larghezza di colonna fissa.

Per impostare la larghezza della colonna, eseguire una delle operazioni seguenti:

- Selezionare la colonna desiderata nel Gestore BOM, quindi digitare un valore per la proprietà **Larghezza** nelle unità di layout.
- Modificare la larghezza della colonna richiesta nella tabella associata. A partire da BricsCAD V20.2, le tabelle BOM mantengono automaticamente la larghezza delle colonne modificate manualmente.

Per ripristinare la modalità di larghezza automatica, selezionare la colonna nel Gestore BOM e selezionare **Auto** nell'elenco a discesa della proprietà **Larghezza**.

Formato colonna

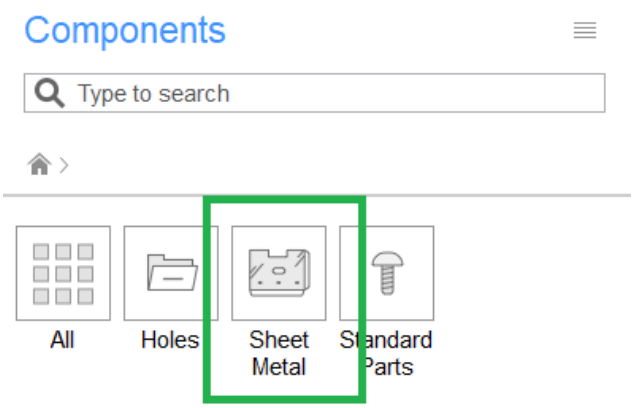


Figura 90: Formato

È possibile formattare i valori di colonna:

1 Selezionare la colonna richiesta nel gestore BOM.

2 Scegliere l'opzione **Formato** nel menu contestuale.

3 Definire il formato desiderato nella finestra di dialogo **Formato**.

Nota: Le impostazioni disponibili dipendono dal tipo di dati della colonna selezionata. È possibile selezionare più colonne dello stesso tipo per formattarle allo stesso modo.

Gruppo di Proprietà

Poiché le tabelle BOM utilizzano la tecnologia dell'Estrazione Dati, è possibile elencare le proprietà dei componenti meccanici e delle entità, ad esempio i riferimenti di blocco nelle tabelle BOM.

Nota: Queste proprietà non sono sempre necessarie. Negli assiemi di grandi dimensioni, queste proprietà potrebbero influire negativamente sulle prestazioni. Ciò è dovuto alle proprietà delle cache di BricsCAD durante la creazione delle tabelle BOM.

I gruppi di proprietà sono gruppi integrati di proprietà. È possibile scegliere il gruppo di proprietà più adatto alle proprie esigenze. Sono supportati i seguenti gruppi di proprietà:

- **Solo Meccanico** (insieme di proprietà predefinito): sono elencate solo le proprietà dei componenti meccanici.
- **Tutte tranne le coordinate**: elenca tutte le proprietà dei componenti, tranne quelle delle coordinate. Le proprietà delle coordinate vengono in genere utilizzate solo per le istanze dei componenti.
- **Tutte le proprietà**: elenca tutte le possibili proprietà dei componenti e delle istanze dei componenti.

È possibile modificare il gruppo di proprietà di una tabella BOM nel Gestore BOM o utilizzare *BMBOM* quando si aggiunge una nuova tabella.

21.4.11.5 Filtrare una tabella BOM

È possibile filtrare le parti che formano la Distinta dei Materiali. A partire da BricsCAD V21, ci sono 2 tipi di filtro: **principale** e **colonna**.

Filtro principale

Un filtro principale funziona allo stesso modo del comando *ESTRDATI*. Innanzitutto, tutte le parti dell'assieme, al livello richiesto (superiore, inferiore o gerarchico), vengono confrontate con l'espressione del filtro. Quindi, solo le parti che superano il filtro vengono incluse nella BOM.

L'espressione del filtro potrebbe non fare riferimento alle colonne BOM perché il filtraggio viene eseguito prima della generazione della BOM. Ad esempio, è possibile creare un filtro per tutte le parti standard perché **E' standard** è una proprietà normale del componente. Tuttavia, non è possibile impostare il filtro principale per filtrare tutte le parti con una quantità superiore a 1. Per questo è necessario utilizzare i filtri Colonna.

Configuratore filtro

Il modo più semplice per configurare il filtro principale è utilizzare il **Configuratore filtro** nel Gestore BOM.

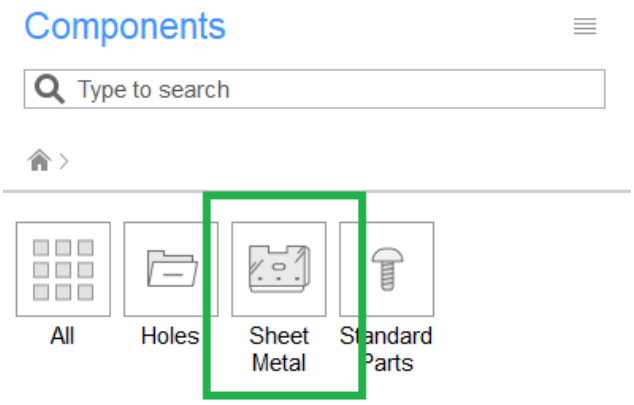


Figura 91: Configuratore filtro

Il Configuratore filtri consente di definire un insieme di condizioni. Ogni condizione definisce una relazione tra una proprietà selezionata e un valore di destinazione.

Le possibili relazioni sono:

- Uguale a
- Non uguale a
- Minore di
- Maggiore di
- Minore o uguale a
- Maggiore o uguale a

Le condizioni possono essere aggiunte a un gruppo, che definisce il modo in cui vengono trattate tali condizioni. Esistono 2 tipi di gruppi:

- **Tutti** i gruppi: è "vero" se tutte le condizioni di quel gruppo sono "vere". Esattamente come l'operatore "and".

- **Qualsiasi** gruppo: è "vero" se almeno una condizione in quel gruppo risulta " vera". Esattamente come l'operatore "or".

Un gruppo può contenere un numero arbitrario di condizioni. Può anche contenere gruppi nidificati, rendendo così possibili filtri complessi.

Per aggiungere una condizione:

- Selezionare una proprietà dall'elenco delle proprietà.
- Selezionare obbligatoriamente un operatore di relazione e specificare il valore di destinazione.

  Per i valori di testo, è possibile selezionare un valore di destinazione anche dall'elenco di tutti i valori possibili. Per i valori numerici, è possibile specificare anche l'unità di destinazione per convertire automaticamente il valore rispetto all'unità predefinita del documento.

Per i valori di testo, le relazioni "Uguale a" e "Non uguale a" supportano i caratteri jolly.

Il Configuratore filtri converte automaticamente l'insieme di condizioni in un'espressione di filtro, che può essere resa visibile facendo clic su :

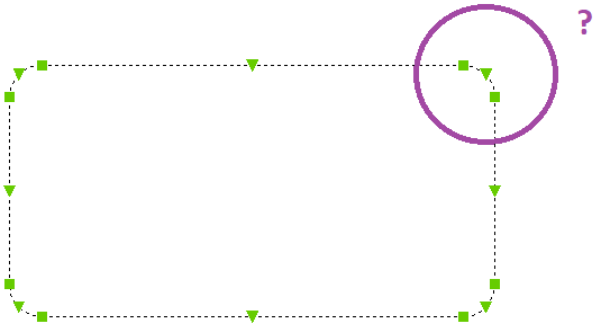


Figura 92: Espressione

Questa espressione è completamente modificabile e associativa a due vie con l'insieme di condizioni.

Descrizione della sintassi:

Tabella 25: Operatori relazionali:

Operatore	Significato
==	uguale a
!=	non uguale a
<	minore di
>	maggiore di
<=	minore o uguale a

Tabella 25: Operatori relazionali:

Operatore	Significato
>=	maggiore o uguale a

- I nomi delle proprietà devono essere racchiusi tra virgolette.
- Gli operatori relazionali possono essere combinati utilizzando operatori "and" e "or". L'operatore "and" si traduce in "vero" se entrambi i lati dell'operatore sono veri. Operatore" or" risulta "falso" se il lato sinistro o destro dell'operatore è vero.
- Gli operatori logici "and" e "or" fanno distinzione tra maiuscole.
- Utilizzare "(" and ")" per combinare due operatori in uno.
- I valori di stringa sono racchiusi tra virgolette doppie "".
- Gli operatori relazionali per le stringhe sono sensibili alle maiuscole e minuscole.
- Gli operatori "==" e "!=" supportano i caratteri jolly.

Ad esempio, per ottenere una BOM di tutte le parti standard, utilizzare la seguente espressione:

"È standard"=="Sì"

Per ottenere una BOM di tutte le parti standard ISO, utilizzare la seguente espressione:

"È standard"==" Sì" and "Parte standard"=="ISO"

Per ottenere una BOM di tutte le parti standard ISO o DIN, utilizzare la seguente espressione:

"È standard"=="Sì" and ("Parte standard"=="ISO" or "Parte standard"=="DIN")

Caratteri Jolly

I caratteri jolly consentono di abbinare i valori delle proprietà a un modello specifico. Ad esempio, se si desidera trovare tutti i componenti contenenti" ugelli" nei loro nomi, è possibile utilizzare la seguente espressione: "Nome componente"=="*ugelli*".

I caratteri jolly possono essere utilizzati anche come valori di destinazione nel Configuratore filtri.

Solo gli operatori relazionali "Uguale a" e "Non uguale a" supportano i caratteri jolly.

Nota: I caratteri jolly nel filtro principale sono sensibili alle maiuscole e minuscole. Per i filtri colonna, è possibile specificare se è necessaria una corrispondenza con distinzione tra maiuscole e minuscole o meno.

Sono supportati i seguenti simboli speciali:

Tabella 26: Simboli speciali

Nome	Carattere	Descrizione
Citazione indietro	`	Esclude il carattere successivo (lo considera letteralmente).

Tabella 26: Simboli speciali

Nome	Carattere	Descrizione
Virgola	,	Separa i modelli.
Asterisco	*	Corrisponde a qualsiasi sequenza di caratteri, inclusa una sequenza nulla
Punto interrogativo	?	Corrispondenza a un singolo carattere.
Punto	.	Corrisponde a un singolo carattere non alfanumerico.
Chiocciola	@	Corrispondenza a un singolo carattere alfabetico.
Cancelletto	#	Corrispondenza a un singolo carattere numerico.
Tilde	~	Se il primo carattere in un modello corrisponde a tutto tranne che al modello.
Parentesi	[...]	Corrispondenza a un singolo carattere racchiuso.
Tilde tra parantesi	[~...]	Corrisponde a un singolo carattere non racchiuso.
Trattino	-	Tra parentesi, corrisponde a un singolo carattere nell'intervallo specificato.

Filtro colonna

I filtri delle colonne sono stati introdotti in BricsCAD V21. Ciò consente di impostare rapidamente le condizioni di visualizzazione delle righe, a seconda del contenuto della colonna (in alcuni processori, in qualche modo simile ai filtri dei fogli di calcolo). A differenza del filtro principale, vengono applicati dopo la generazione della tabella BOM. Pertanto, è possibile utilizzare filtri di colonna per colonne con contenuto aggregato, ad esempio le quantità.

Ogni colonna può avere un solo filtro. Tuttavia, è possibile aggiungere filtri colonna a tutte le colonne di cui si ha bisogno. Nella BOM finale vengono visualizzate solo le righe che superano tutti i filtri di colonna.

È possibile configurare i filtri colonna utilizzando la finestra di dialogo **Filtro colonna** nel Gestore BOM:

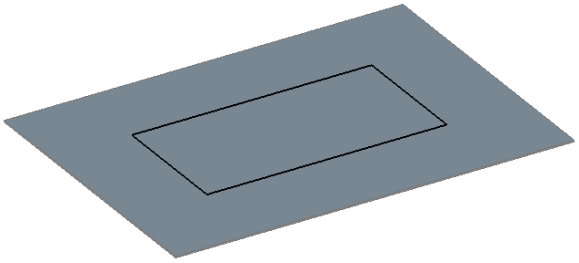


Figura 93: Filtro Rapido

In questa finestra di dialogo è possibile selezionare il tipo di filtro dall'elenco e configurarlo di conseguenza. L'elenco dei possibili filtri dipende dal tipo di contenuto della colonna, come mostrato nella tabella:

Tabella 27: Filtri

Tipo di filtro	Tipo Colonna	Descrizione
sempre accettato	qualsiasi	sono accettati tutti i valori di colonna
uguale a	numerico	il valore deve essere uguale a quello di destinazione
non uguale a	numerico	il valore non deve essere uguale a quello di destinazione
minore di	numerico	il valore deve essere inferiore a quello di destinazione
maggiore di	numerico	il valore deve essere maggiore a quello di destinazione
minore o uguale a	numerico	il valore deve essere minore o uguale a quello di destinazione
maggiore o uguale a	numerico	il valore deve essere maggiore o uguale a quello di destinazione
tra	numerico	il valore deve essere compreso nell'intervallo dato, incluse entrambe le estremità
fuori intervallo	numerico	il valore non deve essere nell'intervallo specificato, entrambe le estremità escluse

Tabella 27: Filtri

Tipo di filtro	Tipo Colonna	Descrizione
corrisponde esattamente	testo	il valore deve corrispondere esattamente a quello di destinazione
non corrisponde	testo	il valore deve essere diverso da quello di destinazione
inizia con	testo	il valore dovrebbe iniziare con il testo specificato
non inizia con	testo	il valore non deve iniziare con il testo specificato
termina con	testo	il valore deve terminare con il testo specificato
non termina con	testo	il valore non deve terminare con il testo specificato
contiene	testo	il valore deve contenere il testo specificato
non contiene	testo	il valore non deve contenere il testo specificato
corrisponde al modello	testo	il valore deve corrispondere al modello di caratteri jolly specificato
non corrisponde al modello	testo	il valore non deve corrispondere al modello di caratteri jolly specificato
in	testo e numerico	il valore dovrebbe essere presente nel gruppo di valori specificato
non in	testo e numerico	il valore non deve essere presente nel gruppo di valori specificato

Per tutti i filtri di testo, è anche possibile specificare se devono distinguere tra maiuscole e minuscole o meno.

21.4.11.6 Ordina una tabella BOM

Esistono 3 modalità di ordinamento per le tabelle BOM:

- **Ordine automatico**

Il contenuto della tabella viene ordinato come segue: nome del componente, nome del materiale fisico, spessore (se applicabile), parametri (in ordine alfabetico) e quantità.

- **Ordine naturale**

  Il contenuto della tabella non è ordinato. Al contrario, tutti i componenti sono nell'ordine di aspetto nel documento. Nella maggior parte dei casi, le parti aggiunte in precedenza precederanno le parti aggiunte successivamente.

- **Ordine personalizzato**

  Il contenuto della tabella viene ordinato in base alle colonne selezionate. Se non viene specificata alcuna colonna, viene utilizzato l'ordine naturale.

A partire da BricsCAD V21, le tabelle BOM utilizzano il cosiddetto ordinamento "naturale" del testo. Significa che se il testo contiene una combinazione di lettere e numeri, verrà ordinato in base a tali numeri. Ad esempio, le stringhe "N1", "M2", "N11", "N2", "AA1" verranno ordinate come "M2", "N1", "N2", "N11", "AA1." Pertanto, è possibile assegnare numeri personalizzati con vari prefissi ai componenti utilizzando parametri personalizzati e quindi ordinare le tabelle BOM in base a tali numeri nell'ordine corretto.

21.4.11.7 Colonne valutate

Le colonne valutate sono colonne speciali, i cui risultati dipendono da altre colonne che utilizzano espressioni definite dall'utente. A partire da BricsCAD V21, sono supportati 2 tipi di colonne valutate: colonne **formula** e **modello**.

Colonna Formula

Una Distinta dei Materiali supporta i campi formula. La stessa sintassi viene utilizzata nel comando *ESTRDATI*. **FormulaExpression** è l'espressione nel formato dei campi formula **\AcExpr**. Può contenere costanti come PI o funzioni come SIN (vedere Usare le Espressioni). Un operando in un'espressione può essere un **ColumnTitle** racchiuso tra minore/maggiore (<>).

Ad esempio, formula *=<Massa>/<Volume>* calcola la densità media del componente.

Se una riga non contiene un valore, verrà utilizzato in alternativa un valore predefinito. È possibile specificare il valore predefinito per un operando utilizzando la seguente sintassi: *<"ColumnTitle",DefaultValue>* dove **ColumnTitle** è il titolo della colonna obbligatoria e **DefaultValue** è il valore predefinito. Il **ColumnTitle** deve essere compreso tra doppie virgolette ("). Se non viene specificato alcun valore predefinito, viene utilizzato un valore vuoto, che nella maggior parte dei casi si traduce in un output vuoto.

Una colonna formula può contenere più espressioni. Ogni espressione deve essere racchiusa tra parentesi graffe ({}). Queste espressioni vengono valutate dalla prima all'ultima e viene utilizzata la prima che produce un output non vuoto.

A partire da BricsCAD V21, una formula può fare riferimento ad altre colonne formula, a condizione che non formino un ciclo. In una BOM gerarchica, una formula può anche fare riferimento ai valori nella riga principale aggiungendo *Parent:* prima del nome della colonna. Può anche fare riferimento a proprietà che non vengono aggiunte esplicitamente alla tabella, ma tutte le altre colonne formula a cui si fa riferimento

nella formula devono esistere nella tabella. È ancora possibile utilizzare tali colonne visualizzate come invisibili nella tabella BOM nelle espressioni formula.

Per le proprietà di distanza, area, volume e massa, i valori verranno convertiti nell'unità **definita dall'utente** prima di utilizzarli nell'espressione formula. Se l'unità **definita dall'utente** non è impostata, viene utilizzata l'unità del documento. Per la proprietà di massa, il chilogrammo viene utilizzato in documenti metrici e le libbre nei documenti imperiali.

Colonna Modello

Le colonne modello consentono di combinare diverse proprietà, oltre a un testo personalizzato, in un'unica colonna. Una sintassi speciale, basata sulle espressioni Diesel, viene utilizzata per definire le colonne modello. Le espressioni per le colonne modello devono iniziare con il segno **$**. Un operando in un'espressione può essere un **ColumnTitle** racchiuso tra minore/maggiore (<>). È anche possibile utilizzare funzioni speciali:

Tabella 28: Funzioni speciali

Funzione	Descrizione
$(+, arg1, arg2[, ... , arg9])	Aggiunge gli argomenti forniti al primo.
$(-, arg1, arg2[, ... , arg9])	Sottrae gli argomenti forniti dal primo.
$(*, arg1 , arg2[, ... , arg9])	Moltiplica il primo argomento per gli argomenti forniti.
$(/, arg1, arg2[, ..., arg9])	Divide il primo argomento per gli argomenti forniti.
$(=, arg1, arg2)	Restituisce 1 se arg1 è uguale a arg2; in caso contrario, restituisce 0. Sia arg1 che arg2 devono essere numeri.
$(<, arg1, arg2)	Restituire 1 se arg1 è inferiore a arg2; in caso contrario, restituire 0. Sia arg1 che arg2 devono essere numeri.
$(>, arg1, arg2)	Restituisce 1 se arg1 è maggiore di arg2; in caso contrario, restituisce 0. Sia arg1 che arg2 devono essere numeri.
$(!=, arg1, arg2)	Restituire 1 se arg1 non è uguale a arg2; in caso contrario, restituire 0. Sia arg1 che arg2 devono essere numeri.
$(<=, arg1, arg2)	Restituire 1 se arg1 è inferiore o uguale arg2; in caso contrario, restituire 0. Sia arg1 che arg2 devono essere numeri.
$(>=, arg1, arg2)	Restituisce 1 se arg1 è maggiore o uguale a arg2; in caso contrario, restituisce 0. Sia arg1 che arg2 devono essere numeri.

Tabella 28: Funzioni speciali

Funzione	Descrizione
$(and, arg1, arg2[, ... , arg9])	Applicare l'operazione AND, bit per bit agli argomenti.
$(angtos, angle[, mode, precision])	Converte l'angolo in stringa rispettando la modalità fornita (0 - gradi, 1 - d/m/s, 2 - gradianti, 3 - radianti) e precisione.
$(delim, delimiter_string)	Inserisce un delimitatore intelligente. La stringa delimitatore viene aggiunta all'output solo se l'output prima e dopo questo delimitatore non è vuoto.
$(edtime, time, format)	Formatta la data in base alla stringa di formato fornita
$(eq, arg1, arg2)	Restituire 1 se arg1 è uguale arg2; in caso contrario, restituire 0. Sia arg1 che arg2 devono essere testo.
$(eval, expression)	Valuta l'espressione fornita come espressione Diesel.
$(fix, value)	Tronca il valore reale a quello intero.
$(getvar, variable)	Restituisce il valore della variabile di sistema fornita.
$(if, arg, then[, else])	Restituisce 'then' se 'arg' non è 0; in caso contrario, restituisce 'else' o stringa vuota se 'else' non viene fornito.
$(index, position, comma_separated_string)	Restituisce la sottostringa nella posizione specificata nella stringa separata da virgole specificata.
$(nth, position, arg1[, arg2, ... , arg9])	Restituisce l'argomento nella posizione specificata.
$(or, arg1, arg2[, ..., arg9])	Applica l'operazione OR, bit per bit agli argomenti.
$(rtos, value[, mode, precision])	Formatta il valore specificato rispettando la modalità e la precisione richiesta.
$(strlen, string)	Restituisce la lunghezza della stringa specificata.
$(substr, string, start[, length])	Restituisce la sottostringa della 'stringa' dall"inizio' della 'lunghezza' specificata. La numerazione inizia da 1.
$(upper, string)	Converte 'stringa' in maiuscolo.

Tabella 28: Funzioni speciali

Funzione	Descrizione
$(xor, arg1, arg2[, ... , arg9])	Applica XOR, bit per bit a tutti gli argomenti.

Una colonna modello può contenere più espressioni. Ogni espressione deve essere racchiusa tra parentesi graffe ({}). Queste espressioni vengono valutate dalla prima all'ultima e viene utilizzata la prima che produce un output non vuoto.

A partire da BricsCAD V21, una colonna modello può fare riferimento ad altre colonne valutate, a condizione che non formino un ciclo. In una BOM gerarchica, una colonna modello può anche fare riferimento ai valori nella riga principale aggiungendo *Parent:* prima del nome della colonna. Può anche fare riferimento a proprietà che non vengono aggiunte esplicitamente alla tabella, ma tutte le altre colonne valutate a cui si fa riferimento in una qualsiasi delle espressioni devono essere presenti nella tabella. È comunque possibile utilizzare le colonne visualizzate come invisibili nella tabella BOM nelle espressioni modello.

È possibile utilizzare la stringa "\n" per aggiungere un'interruzione di riga.

Ad esempio, la seguente espressione combina le proprietà PartNumber, Width, Height e Length:

```
${<PartNumber>$(delim," X ")<Width>$(delim," X ")<Height>$(delim," X ")<Lenght>}
```

Configuratore Espressione

È possibile utilizzare il *Configuratore Espressione* per configurare una colonna valutata. Questa finestra di dialogo è disponibile Gestore BOM.

Figura 94: Editor modelli

Questa finestra di dialogo consente di impostare il titolo e l'unità per la colonna, modificare l'espressione utilizzando un editor con evidenziazione della sintassi e completamento automatico e visualizzare un'anteprima della tabella BOM con la colonna valutata. Per una colonna modello, la sezione **Modello** consente di inserire, spostare ed eliminare facilmente proprietà, nonché delimitatori e testi personalizzati.

21.4.11.8 Unità Colonne

È possibile regolare la modalità **posizionamento unità** per le colonne con le unità associate. Sono disponibili le seguenti opzioni:

- **Non mostrare le unità**

  La tabella BOM non visualizza le unità per questa colonna.
- **Nel titolo della colonna**

  se tutti i valori della tabella utilizzano la stessa unità o possono essere convertiti in un'unità comune, tutti i valori vengono convertiti in tale unità comune. Il nome dell'unità viene aggiunto al titolo della colonna. È possibile configurare la posizione del titolo e dell'unità di misura utilizzando la proprietà **Formato titolo**. Usare *<Titolo>* per indicare il titolo e *<Unità>* per indicare l'unità. Ad esempio, il formato predefinito del titolo è "*<Titolo>*, **<Unità>**". È possibile utilizzare"\n" per aggiungere un'interruzione di riga.

  L'unità comune viene prelevata nel seguente ordine:

  a Un'unità impostata dall'utente.

  b Se la proprietà è una distanza, un'area, un volume o una massa e la modalità unità **Ideale per tutti i valori**, viene utilizzata la rappresentazione migliore per la maggior parte dei valori.

  c Se tutti i valori hanno la stessa unità, verrà utilizzata quest'ultima.
- **Nella cella**

  Il nome dell'unità viene aggiunto al valore. Il valore e l'unità vengono visualizzati nella stessa cella.
- **Nella colonna aggiuntiva**

  Il nome dell'unità viene inserito in una colonna separata.

È possibile impostare la **Modalità Unità** per i valori di distanza, area, volume e massa per definire le unità utilizzate per visualizzare un determinato valore. Ad esempio, se ci sono masse che vanno da 1 grammo a diverse tonnellate, scegliere **Ideale per ogni valore** per visualizzare ciascuna massa convertita nell'unità di massa più adatta. Per visualizzare tutte le masse in chilogrammi, impostare chilogrammi come unità **definito da utente** per la colonna. L'impostazione predefinita è **Ideale per tutti i valori**.

Opzioni **Modalità unità**:

- **Non mostrare le unità**

  Unità non visualizzate.
- **Ideale per tutti i valori**

  Viene visualizzata l'unità con la migliore rappresentazione per la maggior parte dei valori. Sono supportate solo le proprietà dimensioni, area, volume e massa.
- **Definito da utente**

  È possibile scegliere l'unità da visualizzare. Oltre alle dimensioni, all'area, al volume e alle proprietà di massa, è anche possibile selezionare un'unità richiesta per le proprietà del materiale, come densità, conduttività termica e calore specifico.
- **Ideale per ogni valore**

Viene utilizzata l'unità più adatta a ciascun valore. La modalità **Nel titolo della colonna** del **Posizionamento unità** passa automaticamente alla modalità **Nella colonna aggiuntiva**. Sono supportate solo le dimensioni, l'area, il volume e le proprietà di massa.

Nota: Se un'unità definita dall'utente è impostata per una colonna e questa colonna viene utilizzata in un campo formula, i valori convertiti in tale unità verranno sostituiti nella formula. In caso contrario, verrà utilizzata l'unità documento.

21.4.11.9 Piè di pagina

La tabella Distinta dei Materiali supporta un piè di pagina. Per definire quali colonne formeranno il piè di pagina, impostare il **tipo di piè di pagina** per le colonne richieste:

- **Nessuna funzione**

  L'ID colonna non verrà incluso nella riga del piè di pagina.
- **Somma**

  La somma di tutti i valori per ogni singola istanza di componente viene visualizzata nel piè di pagina.

  Nota: Non è l'equivalente della somma di tutti i valori visualizzati nella tabella, poiché viene utilizzato il numero di ogni componente. Per i campi formula, viene utilizzata la somma dei valori per Quantità=1.
- **Medip**

  Il valore medio, di tutti i valori, per ogni singolo componente, viene visualizzato nel piè di pagina. Per i campi formula, viene utilizzata la somma dei valori per Quantità=1.
- **Minimo**

  Il valore più piccolo per ogni singolo componente viene visualizzato nel piè di pagina. Per i campi formula, viene utilizzata la somma dei valori per Quantità=1.
- **Massimo**

  Il valore più grande per ogni singolo componente viene visualizzato nel piè di pagina. Per i campi formula, viene utilizzata la somma dei valori per Quantità=1.

Se tutte le colonne hanno il tipo di piè di pagina **Nessuna funzione**, la riga del piè di pagina non viene visualizzata.

21.4.11.1 Modelli

0

È possibile salvare una configurazione BOM come modello e utilizzarla per creare nuove tabelle BOM. Il file modello contiene tutti i dati necessari. Include il tipo di tabella, il titolo, il filtro e il gruppo di colonne con tutte le proprietà associate. È possibile utilizzare <NOME> nel titolo come segnaposto per il nome effettivo dell'assieme. Ad esempio, se il nome dell'assieme è 'Il mio Assieme' e il titolo della BOM è specificato come Distinta dei Materiali per <NOME>, il titolo della BOM sarà 'Distinta dei Materiali per Il mio assieme', poiché 'Il mio Assieme' sostituisce <NOME>.

Per impostazione predefinita, i files dei modelli BOM hanno un'estensione .BOM.

Un file di modello BOM è un'estensione del file di definizione dell'estrazione dati (.DXD). Tuttavia, non è possibile utilizzare un file con estensione DXD come modello BOM, poiché mancano alcuni dati specifici della BOM. Non è possibile utilizzare un file .BOM per definire un modello di estrazione dati.

È possibile modificare i file modello in qualsiasi editor di testo. Tuttavia, Bricsys consiglia di utilizzare il comando *BMBOM* o il Gestore BOM per creare e modificare i file modello.

21.4.11.1 Stato BOM

1

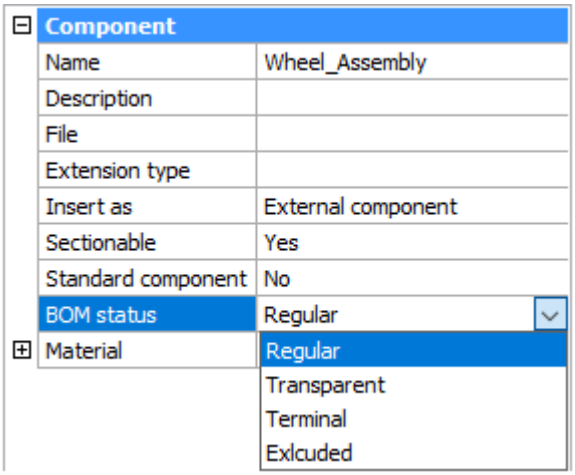


Figura 95: Componente

Ogni componente ha una proprietà di **Stato BOM**. È possibile utilizzarla per controllare il comportamento del componente nelle tabelle delle **Distinta dei Materiali**. Questa proprietà può avere uno dei seguenti valori:

- **Regolare**

  Il componente viene visualizzato in una Distinta dei Materiali nella stessa posizione della struttura dell'assieme e il valore predefinito della modalità selezionata per una Distinta dei Materiali.
- **Trasparente**

  Il componente funge da contenitore trasparente per i suoi componenti secondari. Il componente non viene visualizzato nella Distinta dei Materiali, ma i componenti secondari possono comparire, a seconda dello Stato BOM, della posizione nella struttura dell'assieme e della modalità della tabella BOM.
- **Terminale**

  Il componente viene trattato come un componente figlio (non ha sottocomponenti). Questo è utile per parti complesse o componenti standard.
- **Escluso**

  Il componente e i relativi sottocomponenti non vengono visualizzati nella Distinta dei Materiali.

Impostare lo Stato BOM di un componente radice o esterno

1 Aprire il file contenente il componente.

2 Selezionare il componente radice nel Navigatore Meccanico.

3 Impostare la proprietà **Stato BOM**.

Impostare lo Stato BOM di un componente locale

1 Selezionare il componente nel Navigatore Meccanico

2 Cliccare il pulsante destro del mouse sul nodo selezionato per aprire il menu di scelta rapida

3 Selezionare lo Stato BOM dal sottomenu **Stato BOM**.

Figura 96: Menu contestuale stato BOM

Nota: Non è possibile impostare lo stato BOM tramite il menu contestuale per i componenti locali inseriti in un componente esterno. A tale scopo, è necessario aprire il componente che contiene il componente locale.

21.4.11.1 Angolo fisso

2

A partire da BricsCAD V21, è possibile impostare qualsiasi angolo della tabella come fisso. L'angolo fisso manterrà sempre la posizione iniziale durante l'aggiornamento della BOM. Quando la tabella viene spostata manualmente, verrà mantenuta la nuova posizione di questo angolo. Gli angoli fissi consentono di allineare le tabelle BOM con altri oggetti.

È possibile scegliere quale angolo mantenere fisso durante il posizionamento della tabella tramite il comando *BMBOM* o il Gestore BOM; utilizzare il tasto Tab per scorrere gli angoli. È possibile modificare l'angolo fisso di una tabella BOM nel Gestore BOM.

21.4.12 Animazioni 3D multiparametro

21.4.12.1 Panoramica

Usare i comandi: *ANIMATIONEDITOROPEN* e *ANIMATIONEDITORCLOSE*.

È possibile animare diversi parametri e programmare i movimenti della fotocamera con il pannello Editor Animazione. Una volta programmata l'animazione, è possibile riprodurla in BricsCAD o renderizzarla su un file video 2D.

L'animazione di ogni parametro ha una propria "timeline". È possibile aggiungere tutte le sequenze di animazione ad un parametro desiderato, purché queste sequenze non si sovrappongano.

C'è anche una speciale timeline per i movimenti della fotocamera. Puoi aggiungere più sequenze della fotocamera. È possibile impostare sia il punto di messa a fuoco sia la posizione della fotocamera con una polilinea o un punto. Questo comportamento è simile a quello del comando *ANIPATH*.

Figura 97: Timeline

21.4.12.2 Scene, Timeline e Sequenze di Animazione

Scene

Una scena è costituita da una o più **timelines** e ognuna di queste timeline possiede una o più **sequenze di animazione**. Quando l'editor viene avviato per la prima volta, crea automaticamente una scena. Una scena imposta la durata dell'animazione. Insieme, definiscono il **programma di animazione** utilizzato per la riproduzione e il rendering. Attualmente, con l'Editor Animazione, è possibile eseguire solo il rendering della scena corrente. Tuttavia, puoi avere più scene nello stesso documento e passare da una all'altra.

Sequenze di animazione

Una Sequenza di Animazione è la base di un'animazione. Rappresenta l'animazione singola e continua di un oggetto. Una sequenza di animazione ha associato un orario di inizio e una durata. Possiede anche valori specifici per l'oggetto animato, come i valori inizio e fine per i parametri o i percorsi della fotocamera e del punto di vista per le sequenze della fotocamera:

- **Parametro**

  Una sequenza di animazione rappresenta la modifica continua di un parametro da un valore iniziale a un valore finale. Ad esempio, un vincolo di distanza tra due facce: l'animazione sposterà una faccia a un valore di distanza massima e minima.

- **Fotocamera**

  Una sequenza di animazione corrisponde a un movimento continuo della fotocamera su un percorso. È possibile regolare vari movimenti della fotocamera con la timeline della fotocamera.

Timeline

Una timeline è una rappresentazione visiva di una sequenza di animazione per un oggetto. La timeline mostrerà come e quando animare i parametri. Ogni parametro ha la sua timeline in una scena. Le sequenze di animazione non possono sovrapporsi. Una timeline può essere temporaneamente disabilitata per escluderla da un'animazione. Tutte le timelines sono indipendenti. Ciò significa che puoi animare molti parametri contemporaneamente e vedere i loro effetti combinati.

21.4.12.3 Aprire l'editor di animazione

Effettuare una delle seguenti operazioni:

- Posizionare il cursore su una barra degli strumenti, cliccare il pulsante destro del mouse e scegliere **Pannelli > Editor Animazione** dal menu di scelta rapida.

- Cliccare sullo strumento **Editor di animazione** ( ) nella sezione **Vista** della barra multifunzione.
- Scegliere **Editor Animazione** dal menu **Visualizza**.
- Digitare *ANIMATIONEDITOROPEN* nella barra dei comandi.

21.4.12.4 L'interfaccia dell'Editor di Animazione

L'Editor Animazione è un pannello. Possiede 3 aree principali:

- **Barra degli Strumenti**: Contiene strumenti per selezionare una scena, regolare un'animazione e riprodurre un video.
- **Area Timelines**: per aggiungere e modificare le timelines e le sequenze di animazione.
- **Configuratore**: per configurare i parametri della scena corrente e la sequenza di animazione selezionata.

Barra strumenti

Figura 98: Barra strumenti

La barra degli strumenti contiene strumenti per:

- Aggiungere e rimuovere scene.
- Controllare la riproduzione dell'animazione.
- Impostare la durata della scena corrente.
- Aggiungere una sequenza temporale della fotocamera.
- Avviare un render.

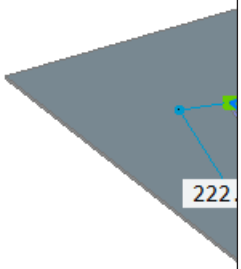
Tabella 29: Barra strumenti

Strumento	Nome	Descrizione
	Aggiungi Scena	Aggiunge una scena vuota con una durata predefinita.

Tabella 29: Barra strumenti

Strumento	Nome	Descrizione
	Selettore Scena	Seleziona la scena corrente.
	Rimuovi scena	Rimuove la scena corrente dal documento.
⏮	**Alla sequenza precedente**	Salta all'inizio della sequenza animata precedente.
▶	**Play**	Avvia l'animazione.
⏸	**Pausa**	Mette in pausa l'animazione.
■	**Stop**	Arresta l'animazione e riporta il cursore nella posizione iniziale.
⏭	**Alla sequenza successiva**	Salta all'inizio della sequenza animata successiva.

Tabella 29: Barra strumenti

Strumento	Nome	Descrizione
	**Rendering video**	Aprie una finestra di dialogo per avviare il rendering di un video 2D.
	Attiva/Disattiva configuratore	Attiva o disattiva il pannello di configurazione.

Area della Timeline

Utilizzare l'**Area Timelines** per aggiungere e rimuovere le timelines, per aggiungere e rimuovere sequenze di animazione e per visualizzare il programma di animazione. Cliccare su una sequenza di animazione per selezionarla o trascinare l'inizio o la fine della barra per regolare la durata. La sequenza di animazione selezionata è visualizzata in blu, le altre sequenze sono visualizzate in grigio. Ogni timeline nell'area delle timelines può essere temporaneamente disabilitata per escluderla dall'animazione.

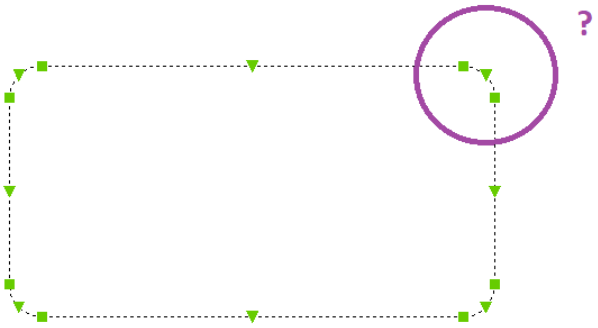


Figura 99: Area della Timeline

L'**Area Timelines** ha un cursore che rappresenta la posizione corrente nell'animazione. Quando viene riprodotta l'animazione, il cursore rappresenta la posizione di riproduzione corrente. Puoi anche spostare manualmente il cursore in una nuova posizione in qualsiasi momento. Quando si sposta il cursore, tutti i

parametri cambiano per corrispondere al fotogramma corrente. Viene aggiornata anche la posizione della fotocamera.

Per assicurarsi che lo stato del modello corrisponda alla posizione corrente dell'animazione, cliccare su ↻.

Nota: È possibile regolarlo manualmente in qualsiasi momento, ma ciò non influirà sull'animazione.

Scala Temporale

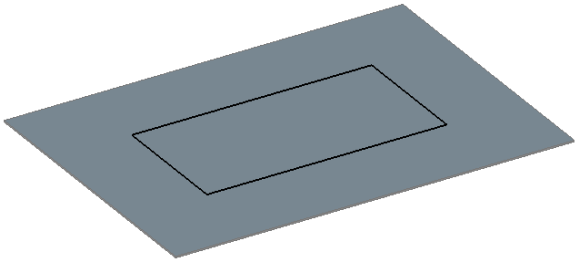


Figura 100: Scala Temporale

Tabella 30: Scala Temporale

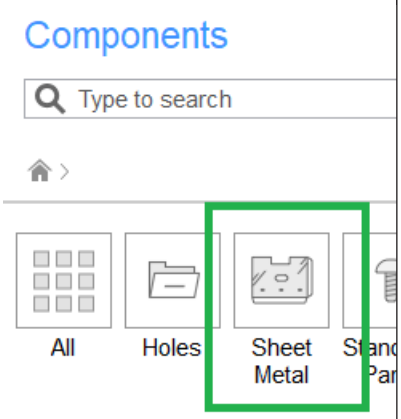
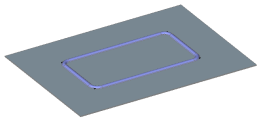
Strumento	Nome	Descrizione
⊕	**Aggiungi Timeline**	Aggiunge il parametro selezionato alla timeline.
	**Selettore Oggetto**	Seleziona l'oggetto da aggiungere alla timeline.
↻	**Aggiorna modello**	Aggiorna lo stato del modello rispetto alla posizione corrente del cursore.

Tabella 30: Scala Temporale

Strumento	Nome	Descrizione
	**Scala Temporale**	Visualizza i marcatori del tempo lungo la timeline
	Cursore	Visualizza la posizione corrente della riproduzione.

Timeline

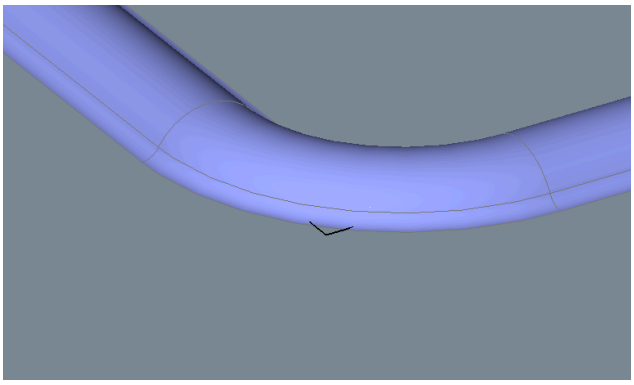
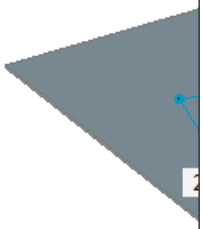


Figura 101: Timeline

Tabella 31: Timeline

Strumento	Nome	Descrizione
	Timeline Abilitata	Indica che la sequenza temporale è abilitata. Cliccare per disattivare la timeline.
	Timeline Disabilitata	Indica che la timeline è disabilitata. Cliccare per attivare la timeline.

Tabella 31: Timeline

Strumento	Nome	Descrizione
	**Oggetto della Timeline**	Visualizza il nome dell'oggetto associato.
⊕	**Aggiungi Sequenza**	Aggiunge una nuova sequenza di animazione, nella posizione del cursore.
	Rimuovi Timeline	Rimuove una timeline dall'animazione.
	Sequenza Animazione	Presenta la durata e la posizione di una sequenza di animazione. Cliccare il pulsante sinistro del mouse per selezionare la sequenza.
	Sequenza Animazione Selezionata	Rappresenta la durata e la posizione della sequenza dell'animazione selezionata. È possibile visualizzare e modificare le proprietà nel **Configurator**. Spostare il mouse alla fine della sequenza, fino a quando il cursore si trasforma in una doppia freccia. Quindi fare clic con il tasto sinistro e trascinare per modificare l'ora di inizio, l'ora di fine e la durata. Cliccare il pulsante sinistro del mouse al centro della sequenza e trascina in una nuova posizione nella timeline, senza modificare la durata.

Configuratore

Accedi a tutte le proprietà della scena dell'animazione e della sequenza selezionata tramite il **Configuratore**. Tutte le proprietà sono raggruppate per oggetto.

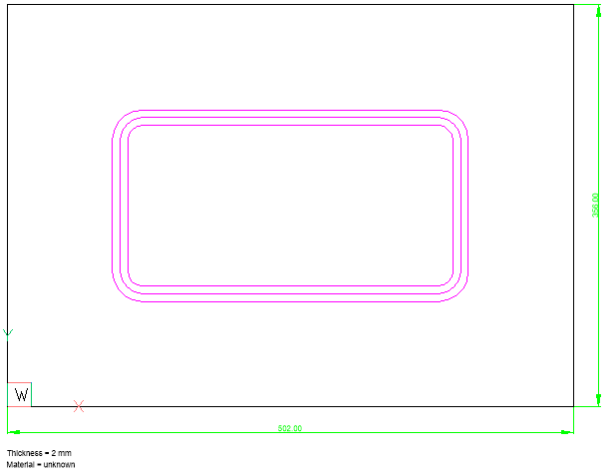


Figura 102: Configuratore

- **Scena**: Proprietà della scena corrente.
  - **Nome**: Nome della scena.
  - **Durata**: Durata della scena, in secondi.
- **Sequenza**: Proprietà della sequenza selezionata.
  - **Tipo di sequenza** (solo lettura): il tipo di sequenza.
  - **Nome oggetto** (sola lettura): il nome dell'oggetto controllato dalla sequenza.
  - **Ora di inizio**: ora di inizio della sequenza, in secondi. La durata verrà regolata, l'ora di fine rimarrà invariata.
  - **Durata**: durata della sequenza, in secondi. Quando viene modificata, l'ora di fine verrà regolata in base all'ora di inizio e alla durata.
  - **Ora di fine**: ora di fine della sequenza, in secondi. Quando viene modificata, la durata verrà regolata, l'ora di inizio rimarrà invariata
- **Parametro** (per sequenze parametriche)
  - **Valore iniziale**: Valore iniziale del parametro.
  - **Valore finale**: Valore finale del parametro.
- **Fotocamera** (per sequenza della fotocamera)
  - **Tipo percorso fotocamera**: tipo di entità percorso della fotocamera. Può assumere uno dei seguenti valori:
    - **Punto**: la fotocamera è fissa nel punto dato.
    - **Percorso**: la fotocamera si sposta lungo il percorso specificato.
  - **Punto fotocamera** (quando il **tipo percorso fotocamera** è impostato su **Punto**). Vengono visualizzati i seguenti controlli:

Tabella 32: Punto fotocamera

	Elenco dei punti utilizzati in precedenza. Selezionare il punto dall'elenco a discesa per utilizzarne uno.

Tabella 32: Punto fotocamera

	Cliccare per selezionare il punto nel modello. Dopo aver selezionato il punto, viene visualizzata una finestra di dialogo: **Figura 103:** Nome Percorso • **Nome**: Immettere il nome del punto. • **OK**: Cliccare per accettare la selezione del punto. • **Annulla**: Cliccare per annullare la selezione del punto.
	Fare clic per rimuovere il punto dall'elenco.

- **Percorso fotocamera** (quando il **tipo percorso fotocamera** è impostato su **Percorso**). Vengono visualizzati i seguenti controlli:

Tabella 33: Percorso fotocamera

	Elenco dei percorsi utilizzati in precedenza. Selezionare il punto dall'elenco a discesa per utilizzarne uno.
	Cliccare per selezionare il percorso nel modello. Dopo aver selezionato il percorso, viene visualizzata una finestra di dialogo: **Figura 104:** Nome Percorso • **Nome**: immettere il nome del percorso. • **OK**: Cliccare per accettare la selezione del punto. • **Annulla**: Cliccare per annullare la selezione del punto.
	Cliccare per rimuovere il percorso dall'elenco.

- **Tipo percorso destinazione**: tipo di percorso di destinazione. Destinazione definisce il punto in cui la telecamera guarderà in un determinato momento. Può assumere uno dei seguenti valori:
- **Punto**: la destinazione è fissa a un determinato punto.
- **Percorso**: la destinazione si muove lungo un determinato percorso.
- **Punto destinazione** (quando il **Tipo percorso destinazione** è impostato su **Punto**). Vengono visualizzati i seguenti controlli:

Tabella 34: Punto destinazione

	Elenco dei punti utilizzati in precedenza. Selezionare il punto dall'elenco a discesa per utilizzarne uno.
	Cliccare per selezionare il punto nel modello. Dopo aver selezionato il punto, viene visualizzata una finestra di dialogo: **Figura 105:** Nome Percorso • **Nome**: Immettere il nome del punto. • **OK**: Cliccare per accettare la selezione del punto. • **Annulla**: Cliccare per annullare la selezione del punto.
	Fare clic per rimuovere il punto dall'elenco.

- **Percorso destinazione** (quando il **Tipo percorso destinazione** è impostato su **Percorso**).

  Vengono visualizzati i seguenti controlli:

Tabella 35: Percorso destinazione

	Elenco dei percorsi utilizzati in precedenza. Selezionare il punto dall'elenco a discesa per utilizzarne uno.
	Cliccare per selezionare il percorso nel modello. Dopo aver selezionato il percorso, viene visualizzata una finestra di dialogo: **Figura 106:** Nome Percorso • **Nome**: immettere il nome del percorso. • **OK**: Cliccare per accettare la selezione del punto. • **Annulla**: Cliccare per annullare la selezione del punto.
	Cliccare per rimuovere il percorso dall'elenco.

Nota: Analogamente a *ANIPATH*, tutti i punti e i percorsi selezionati vengono memorizzati nel disegno.

21.4.12.5 Riproduzione Animazione

Usa i comandi di riproduzione dell'animazione per visualizzare in anteprima un'animazione.

- Per **avviare** un'animazione, cliccare sul pulsante **Play**.

  La riproduzione inizia dalla posizione del cursore.
- Per **sospendere** l'animazione, cliccare sul pulsante **Pausa**.
- Per **interrompere** l'animazione e spostare il cursore all'inizio dell'animazione, cliccare sul pulsante **Stop**.

È possibile passare alla sequenza di animazione successiva o precedente (in ordine cronologico) rispettivamente con gli strumenti **Vai alla sequenza successiva** e **Vai alla sequenza precedente**.

Nota:

- Durante la riproduzione, i parametri coinvolti nella scena e la posizione della fotocamera vengono cambiati direttamente nel disegno. Per i modelli complessi, il ricalcolo del modello può richiedere molto tempo. In alternativa puoi usare il cursore per controllare i momenti chiave dell'animazione.
- Durante la riproduzione, la fotocamera si sposterà con qualsiasi sequenza di fotocamera impostata. Se non sono presenti sequenze di fotocamera, verrà utilizzata la vista corrente. Il video di output sarà lo stesso del finestra corrente.
- Al termine della riproduzione, il modello tornerà automaticamente allo stato precedente all'avvio dell'animazione. In alcuni casi, è possibile che il modello si trovi in uno stato diverso da quello dettato dalla posizione del cursore. Per aggiornare il modello rispetto alla posizione corrente del cursore, cliccare su .

21.4.12.6 Rendering

1 Cliccare sul pulsante **Rendering video** per eseguire il rendering dell'animazione come video 2D.

  Viene visualizzata la finestra di dialogo *Rendering video*:

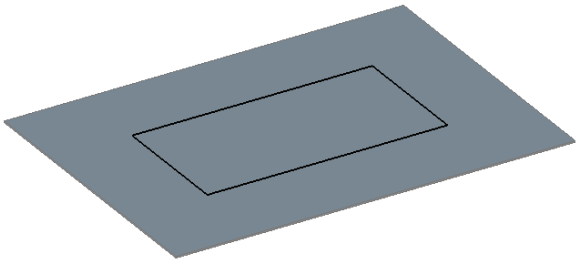


Figura 107: Rendering video

2 Regola i parametri del rendering:
   - **Frequenza fotogrammi di rendering (fps)**: La frequenza dei fotogrammi.
   - **Risoluzione**: La risoluzione del video.
   - **Formato file**: Il formato del video (sono supportati i formati AVI, MPG e WMV).
   - **Ora di inizio**: la posizione temporale in cui inizierà il rendering.

- **Ora di fine**: la posizione temporale in cui terminerà il rendering.

3 Cliccare sul pulsante **Render** per avviare il rendering.

Viene visualizzata una finestra di dialogo per selezionare la posizione del file:

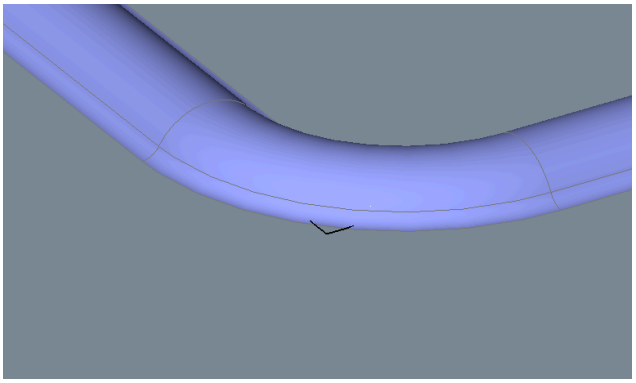


Figura 108: Salva Video Come

4 Specificare la posizione e il nome del file di output.

5 Cliccare su **Salva** per avviare il rendering.

Un'anteprima mostra l'avanzamento.

21.4.12.7 Aggiungere una nuova scena

1 Cliccare sul pulsante **Aggiungi scena**.

Viene visualizzata la finestra di dialogo Aggiungi scena:

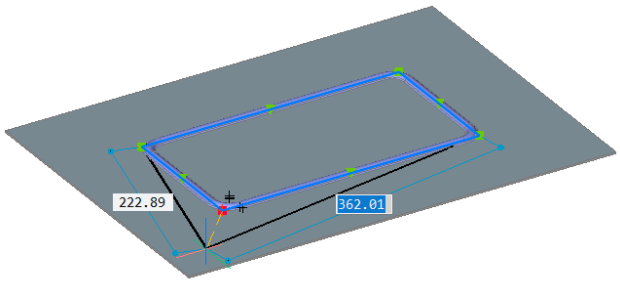


Figura 109: Aggiungi scena

2 Inserire il nome della scena.

3 Immettere la durata della scena in secondi.

4 Cliccare il pulsante **Aggiungi** per aggiungere una scena.

21.4.12.8 Rimuovere una scena

1 Selezionare la scena con il **Selettore Scene**.

2 Utilizzare lo strumento **Rimuovi Scena**.

21.4.12.9 Aggiungere un'animazione di un parametro

1 Aggiungere il parametro desiderato alla timeline:
   - Selezionare il parametro con il **Selettore Parametri**.
   - Utilizzare lo strumento **Aggiungi timeline** per aggiungere una nuova timeline.

2 Posizionare il cursore nella posizione iniziale desiderata sulla **Scala Temporale**.

3 Utilizzate lo strumento **Aggiungi Sequenza** per aggiungere una nuova sequenza di animazione,
o fare doppio clic sulla timeline in cui si desidera aggiungere una nuova sequenza.

4 Trascinare per regolare la durata desiderata della sequenza,
oppure utilizzare il **Configuratore** per impostare l'ora di inizio, l'ora di fine o la durata desidera, in secondi.

5 Utilizzare il **Configuratore** per impostare i valori iniziali e finali desiderati per il parametro.

21.4.12.1 Aggiungere un movimento della fotocamera 0

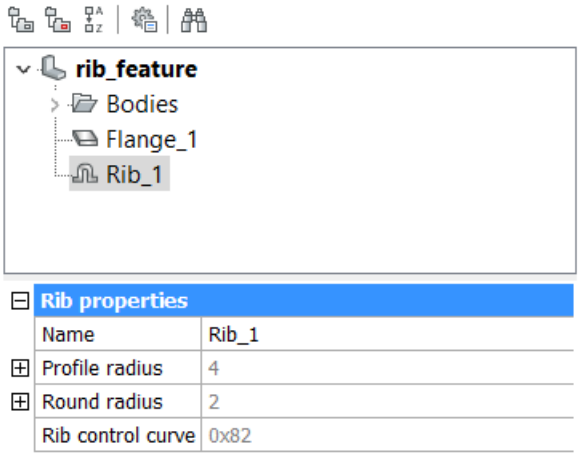
1 Se applicabile, aggiungere una timeline dalla selezione della **Fotocamera** dal **Selettore Oggetto**.

2 Posizionare il cursore nella posizione iniziale desiderata sulla **Scala Temporale**.

3 Usare lo strumento **Aggiungi Sequenza** sulla timeline della fotocamera per aggiungere una nuova sequenza di fotocamera,
o fare doppio clic sulla timeline in cui si desidera aggiungere una nuova sequenza di fotocamera.

4 Cliccare e trascinare per regolare la durata desiderata della sequenza,
oppure utilizzare il **Configuratore** per impostare l'ora di inizio, l'ora di fine o la durata desiderata in secondi.

5 Utilizzare il **Configuratore** per impostare il percorso della fotocamera:
   - Scegliere il **Tipo percorso fotocamera** tra i seguenti:
   - Specificare che la posizione della fotocamera deve essere fissa, oppure
   - **Percorso** per specificare che la fotocamera deve spostarsi lungo il percorso indicato.
   - Utilizzare la proprietà Punto fotocamera per selezionare il punto o il **Percorso telecamera** per selezionare il percorso:
   - Selezionare la proprietà per la modifica, quindi cliccare su



per selezionare il percorso o il punto sul modello,

oppure selezionare uno dei punti o percorsi precedenti dall'elenco a discesa.

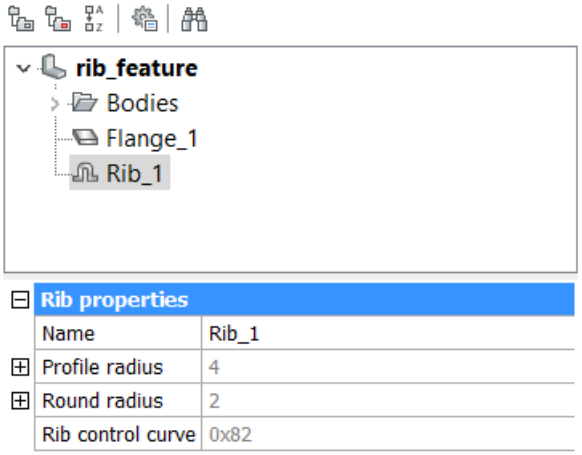
6 Utilizzare il **Configuratore** per impostare il percorso di destinazione. La **destinazione** è il punto di focus della fotocamera.

Per scegliere il **tipo percorso destinazione** eseguire una delle seguenti operazioni:

- Specificare che la destinazione deve essere fissa.
- **Percorso** per specificare che la destinazione deve spostarsi lungo il percorso indicato.

Utilizzare la proprietà **Punto destinazione** per selezionare il punto o la proprietà **Percorso destinazione** per selezionare il percorso:

Selezionare la proprietà per la modifica, quindi cliccare su



per selezionare il percorso o il punto sul modello,

oppure selezionare uno dei punti o percorsi precedenti dall'elenco a discesa.

21.4.12.1 Rimuovere una sequenza di animazione

1

1 Cliccare il pulsante destro del mouse sulla sequenza.

2 Selezionare **Elimina** dal menu di scelta rapida.

In alternativa, è possibile:

1 Cliccare il pulsante sinistro del mouse sulla sequenza.

2 Premere il tasto **Elimina**.

21.4.12.1 Disabilitare o abilitare una timeline

2

1 Controllare l'icona di stato della timeline. Se l'icona è , la timeline è disabilitata. Se l'icona è , la timeline è abilitata.

2 Cliccare sull'icona di stato della timeline per attivare/disattivare lo stato.

Le sequenze di animazione su timeline disabilitate non saranno incluse nell'animazione.

21.4.12.1 Animare una vista esplosa

3

Poiché l'editor animazione non è supportato nell'Editor Blocchi o nelle modalità di modifica dei riferimenti, è possibile animare solo le viste esplose inserite nello spazio modello.

1 Inserite la vista esplosa nello spazio modello.

2 Se applicabile, sopprimere la vista esplosa e il modello, se volete nasconderli o mostrarli durante l'animazione.

3 Selezionare la vista esplosa dell'oggetto nel **Selettore Oggetto**.

4 Cliccare su per aggiungere una timeline.

5 Utilizzare lo strumento **Aggiungi Sequenza** nella sequenza temporale della vista esplosa per aggiungere una nuova sequenza della vista esplosa,

o fare doppio clic sulla sequenza temporale in cui si desidera aggiungere una nuova sequenza.

6 Cliccare e trascinare per regolare la durata desiderata della sequenza,

oppure utilizzare il **Configuratore** per impostare l'ora di inizio, l'ora di fine o la durata desiderata in secondi.

7 Se applicabile, utilizzare il **Configurator** per impostare il passo iniziale e il passo finale della vista esplosa o per abilitare l'opzione di nascondere automaticamente.

21.4.12.1 Renderizzare un video

4

Vedere la sezione Rendering.

21.5 Lamiera

21.5.1 Metodi di Progettazione della Lamiera

La progettazione della lamiera consente di modellare parti in lamiera e generare la loro rappresentazione come sviluppo 2d completo delle informazioni per la produzione.

Si possono creare parti complesse in lamiera metallica con BricsCAD in modo semplice e rapido perché il processo di progettazione è diverso da quello di produzione. Non si pensa in termini di fogli planari che devono essere tagliati e piegati, ma si modella direttamente la parte mentre si creano corpi solidi con gli strumenti di modellazione diretta.

Esercitazione: Introduzione alla Lamiera di BricsCAD

Esercitazione: Progettare da Zero la Lamiera in BricsCAD

Con BricsCAD, si possono creare parti di lamiera in 4 modi diversi (o in qualsiasi combinazione di essi):

1 Creare una flangia di base da un profilo chiuso piano, quindi aggiungere le flange:
   - Tirare gli spigoli per creare le flange aggiuntive complete della piegatura.
   - Creare flange parziali con la creazione automatica delle corrispondenti gole di scarico.
   - Tirare più spigoli adiacenti per creare diverse flange contemporaneamente con la creazione automatica delle gole di scarico e giunzioni.
   - Piegare una flangia per creare una nuova flangia completa di piega.

2 Creare una combinazione di flange e curve con loft che collegano due profili 3D.

   Utilizzare lo strumento Lamiera Loft
   .

3 Creare una combinazione di flange e piegature da un profilo planare.

   Utilizzare lo strumento Estrudi Polilinea.

4 Crea un corpo svuotato da un solido 3D.

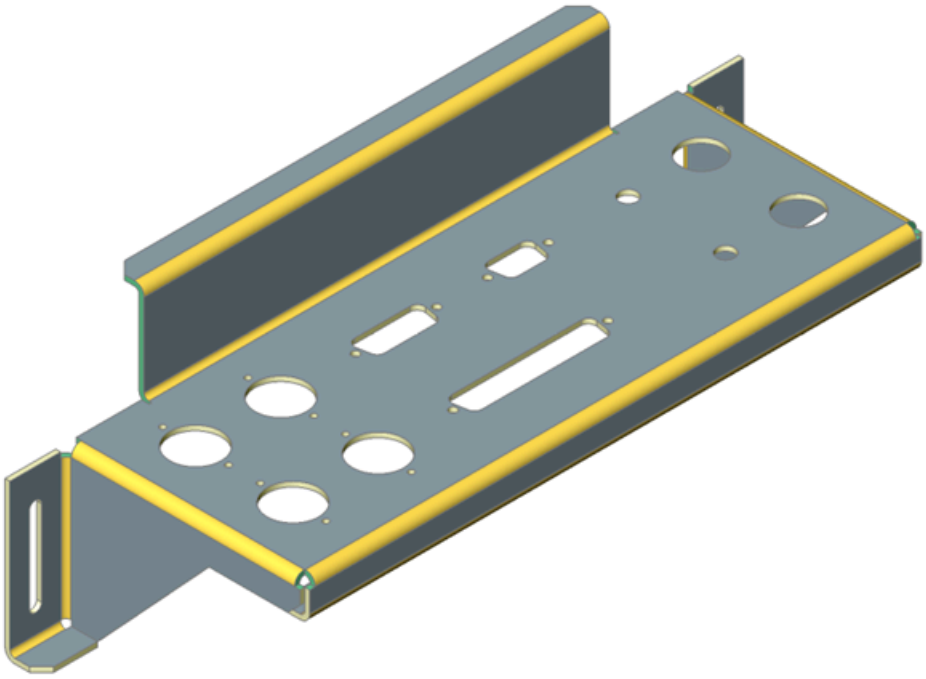
   Utilizzare l'opzione *Svuota* del comando *MODIFSOLIDI* con lo spessore desiderato, quindi convertire il risultato in una parte di lamiera con il comando *SMCONVERT* e creare piegature e giunzioni dagli spigoli vivi.
   - Praticare fori a partire da profili chiusi sulle facce delle flange e spingendoli attraverso le flange stesse.
   - Regolare lo spessore del materiale ed il raggio di piegatura utilizzando parametri predefiniti.
   - Applicare le operazioni di Modellazione Diretta e vincoli 3D per modificare ulteriormente il vostro disegno, sempre mantenendo l'intento del progetto in termini di caratteristiche della lamiera.
   - Generare automaticamente la rappresentazione dello sviluppo 2d della vostra parte ed inviarla ad un sistema CAM esportando il corrispondente disegno 2D con le annotazioni di piegatura in un file .dwg o .dxf .

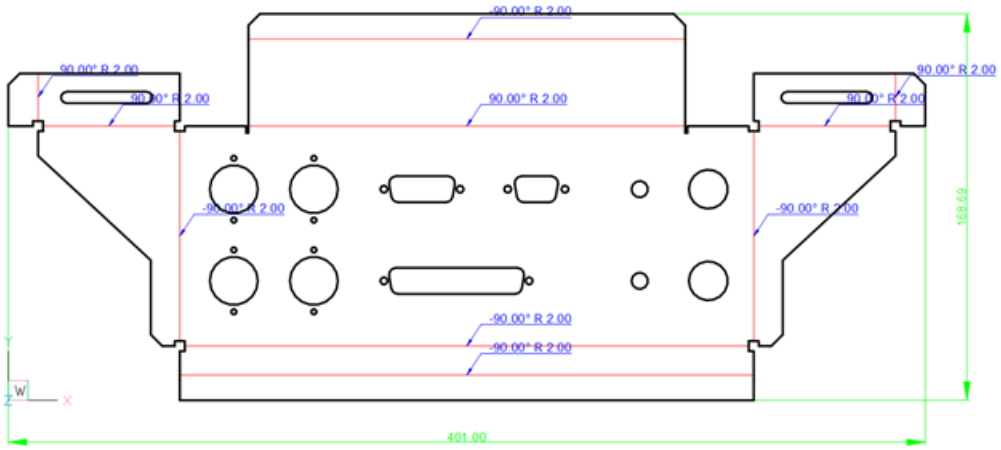
Si può ulteriormente dettagliare la parte di lamiera utilizzando le seguenti operazioni:

- Praticare fori a partire da profili chiusi sulle facce delle flange e spingendoli attraverso le flange stesse.
- Inserimento di Caratteristiche Forma da una libreria. Con BricsCAD Sheet Metal viene installata una libreria di caratteristiche di forma parametrica ed è possibile creare le proprie caratteristiche forma.

- Creare caratteristiche di nervature da una curva 2D che si trova sulla flangia utilizzando il comando *SMRIBCREATE*.
- Regolare lo spessore del materiale ed il raggio di piegatura utilizzando parametri predefiniti.
- Applicare le operazioni di Modellazione Diretta e vincoli 3D per modificare ulteriormente il vostro disegno, sempre mantenendo l'intento del progetto in termini di caratteristiche della lamiera.

Quando la parte in lamiera è pronta, è possibile dispiegarla automaticamente e inviare il risultato a un ingegnere di produzione esportando il disegno 2D corrispondente.





Esercitazione: Elaborazione Batch in BricsCAD Sheet Metal

21.5.2 Comandi e Barre degli Strumenti Lamiera

Tabella 36: Comandi e Barre degli Strumenti Lamiera

Icone della Barra degli Strumenti Lamiera	Icone Barra degli Strumenti a Comparsa	Comando	Descrizione
		SMFLANGEBASE	Crea una flangia di base (iniziale) di una parte in lamiera da un profilo piano chiuso.
		SMEXTRUDE	Crea una parte in lamiera estrudendo una polilinea nella direzione della normale del piano della curva. Lo spessore viene assegnato rispetto alle dimensioni esterne della polilinea.
		SMLOFT	Crea parte di lamiera con piegature e flange da 2 curve non complanari.
		SMCONVERT	Riconosce automaticamente flange, piegature e piegature lofted in un solido 3D.
		SMFLANGEEDGE	Crea una o più flange su una parte in lamiera tirando uno o più spigoli di una flangia esistente.
		SMFLANGECONTOUR	Crea una nuova flangia da un contorno chiuso e la collega a una parte di lamiera esistente.
		SMFLIP	Cambia il lato di riferimento per le flange selezionate con uno spostamento opzionale della geometria della flangia sullo spessore.
		SMFLANGEROTATE	Ruota una flangia.
		SMHEMCREATE	Crea una o più caratteristiche orlo chiuso su una parte in lamiera metallica tirando uno o più spigoli di una flangia esistente.
		SMHEMCREATE	Crea una o più caratteristiche orlo aperto su una parte in lamiera metallica tirando uno o più spigoli di una flangia esistente.

Tabella 36: Comandi e Barre degli Strumenti Lamiera

Icone della Barra degli Strumenti Lamiera	Icone Barra degli Strumenti a Comparsa	Comando	Descrizione
		SMHEMCREATE	Creare una o più caratteristiche orlo a goccia su una parte in lamiera tirando uno o più spigoli di una flangia esistente.
		SMHEMCREATE	Creare una o più caratteristiche orlo circolare su una parte in lamiera tirando uno o più spigoli di una flangia esistente.
		BMINSERT	Inserisce una caratteristiche forma.
		SMFORM	Crea una caratteristica forma da un determinato gruppo di facce.
		SMFORM	Esplode una caratteristica forma in flange e piegature, se possibile.
		SMRIBCREATE	Crea una caratteristica forma di nervatura associativa su una faccia della flangia da una curva 2D planare.
		SMREPLACE	Sostituisce le caratteristiche forma (comprese quelle riconosciute) presenti nelle parti in lamiera, con le caratteristiche forma della libreria utente o incorporata.
		SMBENDCREATE	Converte gli spigoli vivi (bordi taglienti tra le facce delle flange) e le giunzioni, in piegature.
		SMBENDSWITCH	Converte le piegature in piegature con loft.
		SMFLANGEBEND	Piega una flangia esistente lungo una linea, obbedendo al fattore k per un dato raggio di piegatura.
		SMJUNCTIONCREATE	Converte gli spigoli vivi in giunzioni.

Tabella 36: Comandi e Barre degli Strumenti Lamiera

Icone della Barra degli Strumenti Lamiera	Icone Barra degli Strumenti a Comparsa	Comando	Descrizione
		SMJUNCTIONSWITCH	Modifica una caratteristica giunzione simmetrica ad una con le facce sovrapposte.
		SMFLANGECONNECT	Chiude gli spazi tra due flange arbitrariamente orientate.
		SMTABCREATE	Creare una caratteristica linguetta singola in una parte di lamiera cambiando la giunzione esistente in linguetta.
		SMTABCREATE	Crea una caratteristica serie di linguette presente in una parte di lamiera trasformando una giunzione esistente in linguetta.
		SMRELIEFCREATE	Rende corretti gli scarichi d'angolo e le gole di scarico. Gli scarichi d'angolo vengono realizzati su angoli con 3 o più flange adiacenti. Le gole di scarico vengono realizzate sull'inizio e la fine di uno spigolo della flangia.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in uno scarico Circolare.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in un scarico Rettangolare.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in uno scarico Tipo V.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in uno scarico Uniforme.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in uno scarico Asolato.
		SMRELIEFSWITCH	Converte uno scarico d'angolo in uno scarico Lacerato.

Tabella 36: Comandi e Barre degli Strumenti Lamiera

Icone della Barra degli Strumenti Lamiera	Icone Barra degli Strumenti a Comparsa	Comando	Descrizione
		SMSPLIT	Divide una flangia lungo una linea disegnata sulla sua faccia.
		SMSPLIT	Divide una flangia da un singolo vertice nell'angolo.
		SMSPLIT	Dividi una flangia da un singolo vertice nell'angolo e suggerisce automaticamente più divisioni analoghe.
		SMIMPRINT	Divide le facce dello spessore di una parte in lamiera con spigoli stampati, in base alle flange e alle pieghe adiacenti.
		SMDISSOLVE	Rimuove i dati della lamiera dalle facce ed entità selezionate.
		SMDELETE	Rimuove una piegatura o una giunzione ripristinando uno spigolo vivo tra due flange; rimuove una flangia con tutte le piegature adiacenti ad essa. Le flange adiacenti vengono estese fino ad una configurazione di giunzione con la flangia da eliminare. Rimuove i tagli ad angolo ripristinando la geometria tagliata dalla caratteristica.
		SMREPAIR	Risolvi i problemi specifici delle parti in lamiera, entrambi i flussi di lavoro globali e locali sono supportati. Facoltativamente, ripara le caratteristiche di piegatura con loft: unisce le pieghe adiacenti e fornisce una connessione tangenziale con le flange.
		SMREPAIR	Corregge i problemi specifici delle parti in lamiera. Solo la flangia selezionata o la piegatura sbagliata vengono riparate.
		SMSELECT Caratteristiche forma uguale	Seleziona tutte le caratteristiche forma identiche nel modello della lamiera

Tabella 36: Comandi e Barre degli Strumenti Lamiera

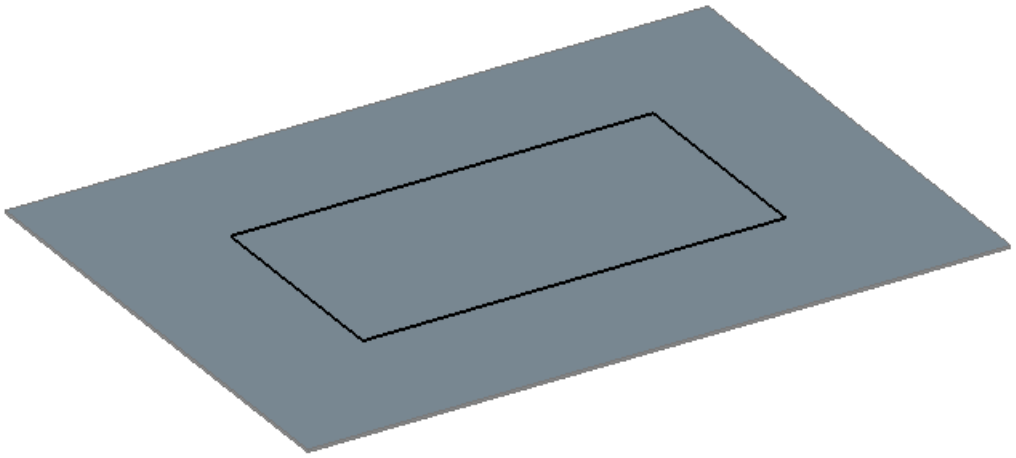
Icone della Barra degli Strumenti Lamiera	Icone Barra degli Strumenti a Comparsa	Comando	Descrizione
		SMSELECT Spigoli vivi	Seleziona tutti gli spigoli vivi delle parti in lamiera.
		SMSELECT Spigoli non ortogonali allo spessore	Seleziona tutti gli spigoli non ortogonali allo spessore sulla parte in lamiera.
		SMSELECT Spigoli piatti	Seleziona tutti gli spigoli piatti sulla parte in lamina.
		SMSELECT lato della parte in lamiera	Seleziona il lato della parte in lamiera.
		SMPARAMETRIZE	Crea un insieme consistente di vincoli 3D per la parte di lamiera selezionata.
		SMUNFOLD	Spiega il corpo in lamiera.
		SMUNFOLD Associativo	Apre una finestra aggiuntiva con la rappresentazione spiegata del modello.
		SMEXPORTOSM	Esporta un solido di lamiera in formato .osm Formato file (Open Sheet Metal).
		SMEXPORT2D	Esporta la rappresentazione spiegata di un corpo in lamiera come profilo 2D nel formato file .dxf / .dwg.
		SMASSEMBLYEXPORT	Esegue un'esportazione in batch dei componenti di un assieme per lo spiegamento, creando un .dxf per ogni solido 3D riconosciuto come parte di lamiera.
		FEATURECOLORS	Attiva o disattiva la variabile di sistema *FEATURECOLORS*.

Gli strumenti per il disegno della Lamiera sono disponibili:

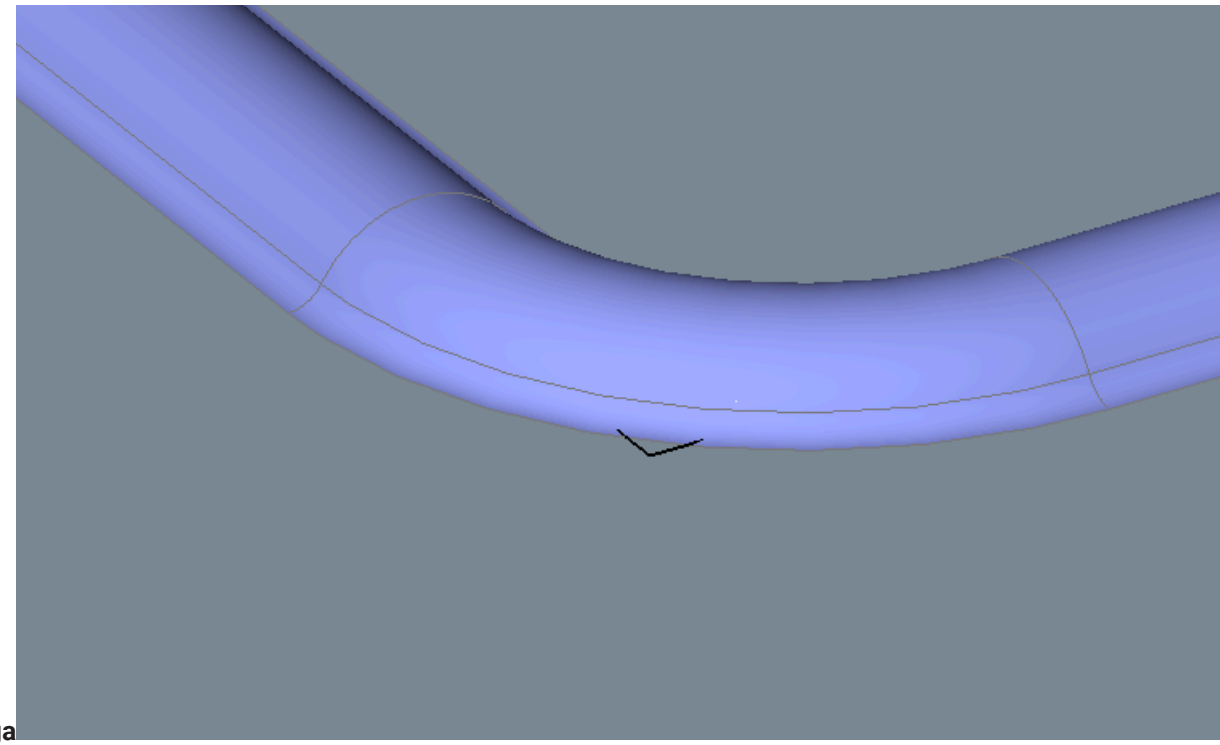
- Nel **Quad** (a seconda della caratteristica sotto il cursore):

Figura 110: Il menu cursore Quad

- Sulla barre degli strumenti **Lamiera**:

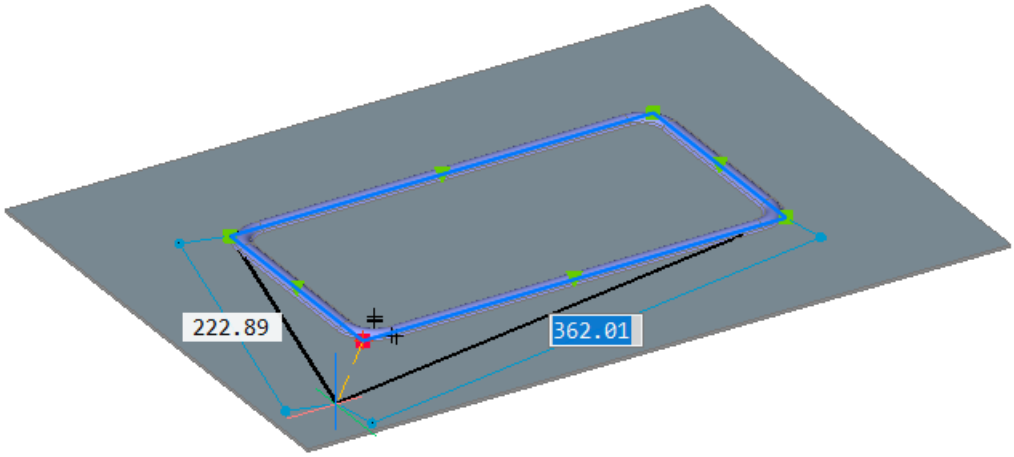


  - **Lamiera**
  - **SM**



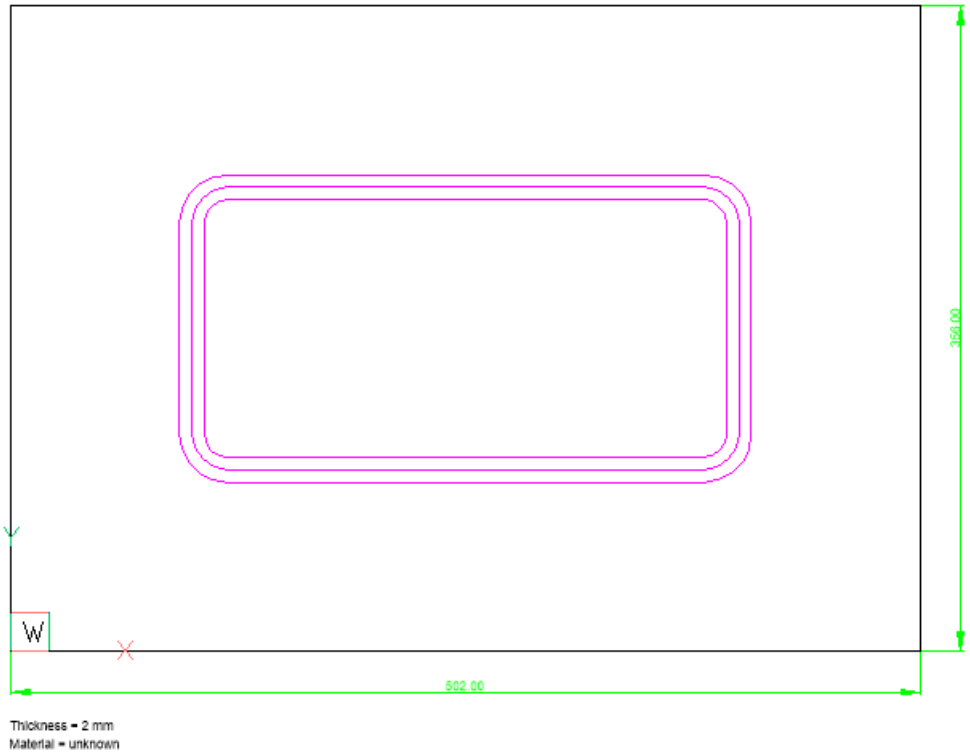
Piega

- SM Flangia
- SM Forma
- SM



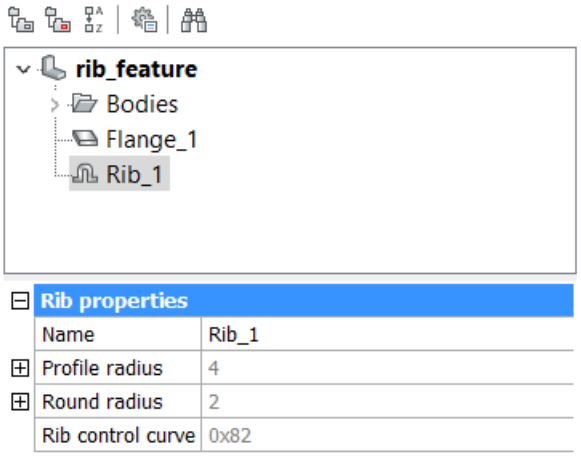
Orlo

- SM



Giunzione

- **SM Scarico**
- **SM Ripara**
- **SM Dividi**
- **SM Linguetta**

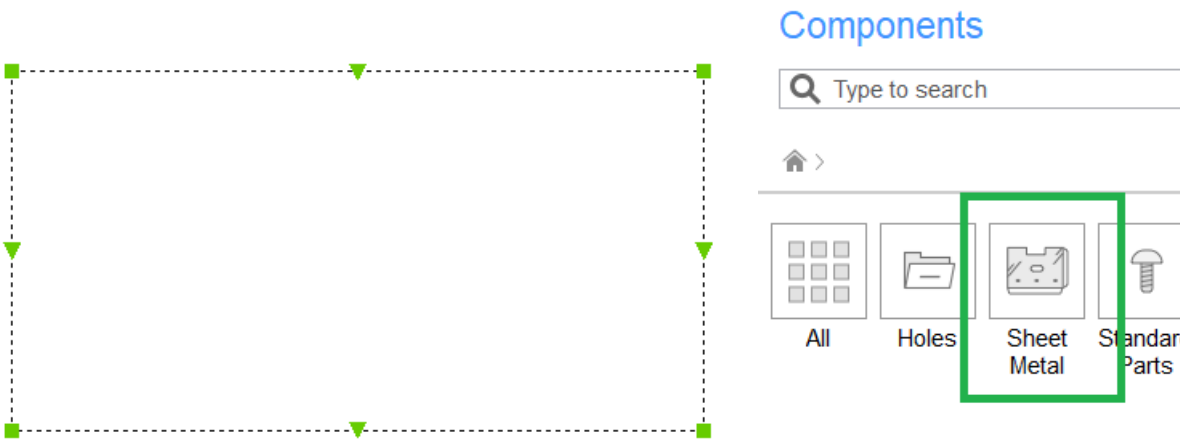


- **SM Spiegamento**

- Nel menu **Lamiera** (Solo nell'area di lavoro **Meccanico**):

Figura 111: Menu Lamiera

- Nella scheda **Lamiera** della barra multifunzione dell'area di lavoro



Meccanico.

Vedere anche Modellazione Diretta strumenti.

21.5.3 Caratteristiche della Lamiera

21.5.3.1 Panoramica

Le caratteristiche sono gruppi di facce intelligenti della parte 3D. Ogni caratteristica mantiene il rapporto spaziale e parametrico specifico tra le sue facce e alcune facce adiacenti. Le caratteristiche consentono di incorporare l'intento progettuale nel tuo modello. Le caratteristiche vengono create automaticamente da una particolare operazione geometrica applicata. Le seguenti immagini mostrano le caratteristiche della lamiera in BricsCAD:

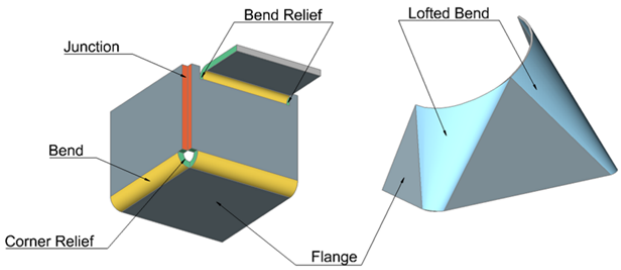


Figura 112: Esempio 1 di caratteristiche della lamiera

Figura 113: Esempio 2 di caratteristiche della lamiera

21.5.3.2 Flangia

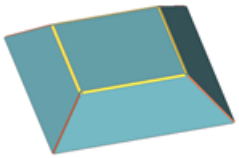
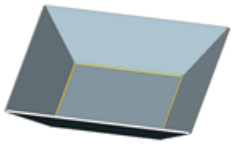
La caratteristica principale di tutte le parti di lamiera è la flangia, che consiste in 2 facce piane parallele posizionate l'una di fronte all'altra (con un volume solido tra di loro) sulla distanza, che è significativamente più piccola della dimensione lineare delle facce. La distanza tra loro corrisponde allo spessore del materiale. Quando si modifica il modello, questo rapporto di distanza viene automaticamente sempre mantenuto. Altre facce, adiacenti alla faccia della flangia e che non appartengono alle piegature vengono chiamate facce di spessore. Sono sempre perpendicolari alle facce della flangia.

Le caratteristiche Flange vengono create dai seguenti comandi: *SMFLANGEBASE*, *SMEXTRUDE*, *SMLOFT*, *SMFLANGEEDGE*, *SMFLANGEBEND* e *SMCONVERT*.

Se la variabile di sistema *FEATURECOLORS* è *ON*, il colore differenzia la faccia di riferimento di una flangia e il suo lato opposto. Se possibile, le modifiche parametriche di una parte in lamiera, come il cambio di spessore, non spostano le facce di riferimento. Il comando *SMFLIP* inverte i lati di una flangia selezionata in modo che le facce di riferimento si trovino sull'altro lato geometrico della flangia; opzionalmente, il comando sposta la flangia sullo spessore della parte in lamiera.

Le seguenti immagini illustrano la legenda dei colori delle facce della flangia della stessa parte.

Tabella 37: Flangia

	
Facce di Riferimento	Altre Facce

21.5.3.3 Gola

Due flange sono collegati da una piega. Una piega è costituita da 2 facce cilindriche coassiali, che sono sempre tangenti alle facce planari adiacenti delle flange.

Le caratteristiche Piegatura vengono create dai comandi *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND* e *SMCONVERT*.

21.5.3.4 Piegatura Zero

Questo è un concetto di progettazione della piegatura, ma la geometria ha una faccia cilindrica per il lato esterno della piegatura con un raggio uguale allo spessore. Invece di una faccia interna, c'è uno spigolo. Così la geometria rappresenta efficacemente una curvatura di raggio zero, che viene utilizzata per lo scopo della progettazione. L'operazione *SMUNFOLD* supporta piegature di questo tipo e genera quotature come se queste piegature fossero normali piegature con un raggio uguale allo spessore.

21.5.3.5 Piegatura Errata

Le caratteristiche di piegatura errate vengono riconosciute dal comando *SMCONVERT* e riflettono ipotesi sull'intento di progettazione dell'utente. Come le normali piegature, le piegature errate collegano le flange, ma presentano avarie geometriche, come raggi o disallineamenti angolari che impediscono l'uso della geometria come normali caratteristiche di piegatura. Prima di spiegare la parte, tutte le piegature errate devono essere riparate da *SMREPAIR* o rimosse da *SMDELETE* e quindi ricreate da *SMBENDCREATE*.

21.5.3.6 Piegatura con Loft

Una caratteristica piegatura loft è costituita da due superfici rigate poste una di fronte all'altra (con offset uguale allo spessore del materiale) con un volume solido tra loro. Una superficie rigata può essere descritta come l'insieme di punti trascinati da una linea retta mobile. Esempi sono cilindri, coni, elicoidi. Le piegature loft possono essere create ispessendo le superfici rigate. Una piegatura con loft può collegare 1 o 2 flange o essere autonoma, senza flange. Sono consentite anche alcune combinazioni di adiacenza tra piegatura normale e piegatura loft.

Differenze tra piegatura e piegatura con loft:

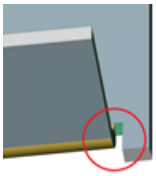
Tabella 38: Differenze

	Piegatura	Piega con Loft
Superfici e	CIlindrico	Cilindrica, conica o spline
Spiegam ento	Esatto	Lo spiegamento approssimativo delle piegature con loft viene controllato da un numero impostabile di operazioni di piegatura, che viene denominato Numero di Suddivisioni. Più operazioni vengono eseguite, più uniforme sarà la produzione della parte reale e migliore sarà l'approssimazione del progetto iniziale. Le piegature con loft cilindriche e coniche hanno un'opzione per lo spiegamento analitico corretto.

Le caratteristiche di piegatura con loft vengono create dai comandi *SMLOFT* e *SMCONVERT*.

21.5.3.7 Gola di Scarico

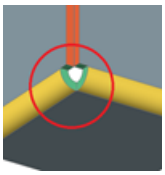
Gruppo di facce che rappresentano un taglio tecnologico tra 2 flange di diversa larghezza collegate con una piegatura. La caratteristica gola di scarico mantiene la distanza tra le due facce opposte del taglio.



Le caratteristiche gole di scarico vengono create dai comandi *SMFLANGEEDGE*, *SMBENDCREATE* e *SMRELIEFCREATE*.

21.5.3.8 Scarico d'Angolo

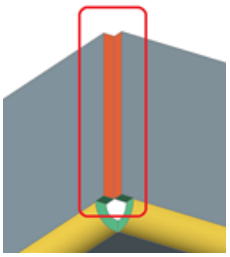
Un gruppo di facce che rappresentano un taglio nell'angolo dove si incontrano 3 o più flange. Questo taglio è necessario per evitare la deformazione indesiderata del materiale durante la piegatura della foglio di lamiera. Una caratteristica scarico d'angolo mantiene la forma e le dimensioni di questo taglio.



Le caratteristiche scarico d'angolo vengono create dai comandi *SMFLANGEEDGE*, *SMBENDCREATE* e *SMRELIEFCREATE*.

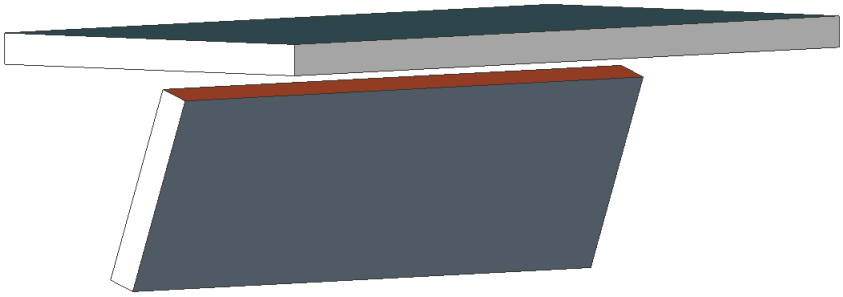
21.5.3.9 Giunzione e Giunzione a T

La caratteristica giunzione è costituita da due facce di spessore di flange adiacenti, che non sono collegate tramite una piegatura. Una giunzione può essere simmetrica o sovrapposta. La caratteristica di giunzione mantiene la distanza minima tra le flange: il vuoto della giunzione.



Una caratteristica di giunzione a T è costituita da una faccia di spessore di una singola flangia, che si trova vicino a una faccia di un'altra flangia. Di solito, una caratteristica di giunzione a T precede la creazione di una limguetta a T.

Per maggiore chiarezza, il vuoto della giunzione nell'immagine sottostante è intenzionalmente piuttosto grande.

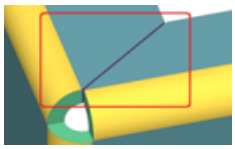


Le caratteristiche di giunzione vengono create dai comandi *SMFLANGEEDGE*, *SMJUNCTIONCREATE* e *SMFLANGECONNECT*.

Le caratteristiche di giunzione a T vengono create dai comandi *SMJUNCTIONCREATE* e *SMFLANGECONNECT*.

21.5.3.10 Funzione Taglio ad Angolo

Una caratteristica taglio ad angolo rappresenta una divisione tra le flange complanari create dai comandi *SMFLANGEEDGE* e *SMSPLIT*. Una caratteristica taglio ad angolo mantiene lo spazio del taglio ad angolo.



21.5.3.11 Caratteristiche Nervatura e Forma

Le caratteristiche di forma e nervatura rappresentano una ricca varietà di forme che sono prodotte dalla punzonatura.

Sono fondamentali due rappresentazioni delle caratteristiche di forma: una in 3D ed un'altra come simbolo 2D nella geometria ottenuta dallo spiegamento. Nel caso di geometria importata il simbolo viene ottenuto dalla proiezione delle facce 3D della caratteristica di forma. Nel caso in cui la caratteristica di forma è stato inserita/sostituita dalla caratteristica forma della libreria, il simbolo utilizzato nello spiegamento viene preso dal file della libreria.

Le caratteristiche nervatura rappresentano l'impronta sulla flangia creata da una ruota che segue una traiettoria arbitraria.

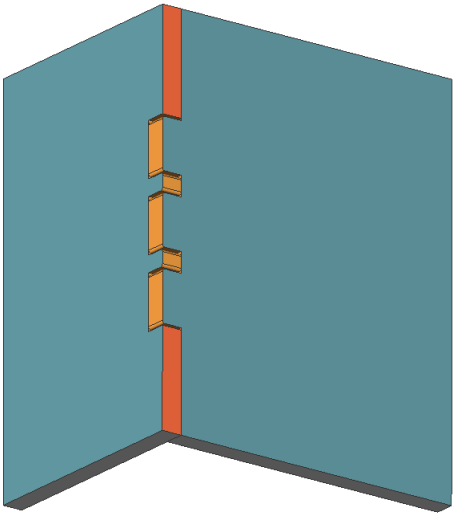
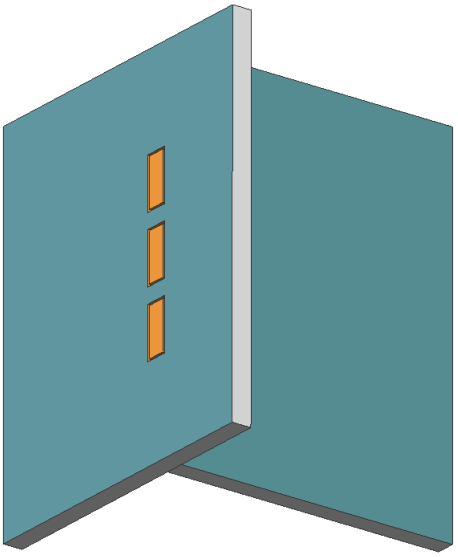
Le caratteristiche di forma e nervatura vengono create dai comandi *BMINSERT*, *SMRIBCREATE* e *SMCONVERT*.

21.5.3.12 Linguette e Linguette a T

Entrambe le caratteristiche vengono utilizzate per collegare 2 flange, limitando il loro spostamento reciproco.

Le caratteristiche linmguetta e linguetta a T vengono create dal comando *SMTABCREATE*.

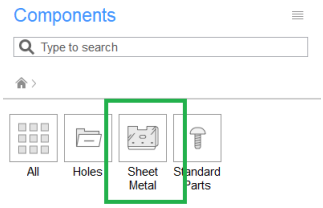
Tabella 39: Linguette e Linguette a T

 **Figura 114:** Esempio di linguetta	 **Figura 115:** Esempio di linguetta a T
Caratteristica Linguetta	Caratteristica Linguetta a T

21.5.3.13 Smusso

Le caratteristiche smusso rappresentano la cianfrinatura sulle facce di spessore delle parti in lamiera. Sia le facce inclinate che quelle ortogonali definiscono una caratteristica smusso. Le caratteristiche smusso devono essere condivise rigorosamente da un'unica flangia e non possono essere adiacenti alle caratteristiche di piegatura.

Le caratteristiche smusso possono essere riconosciute o cancellate dal comando *SMCONVERT* e sono supportate dal comando *SMUNFOLD*.



21.5.4 Colori Caratteristiche

Per un'esperienza migliore e una facile distinzione tra le caratteristiche, vengono applicati diversi colori ai vari tipi di caratteristiche nella geometria. Questi colori sono solo per la visualizzazione e non verranno mai visualizzati nei documenti generati, nelle viste dello spazio carta, negli inserti di blocchi, ecc.

La visualizzazione dei colori delle caratteristiche è controllata tramite la variabile di sistema *FEATURECOLORS* e un'altra serie di variabili di sistema, una per ogni colore di caratteristiche. Sebbene i colori predefiniti delle caratteristiche siano selezionati con attenzione per combinarsi tra di loro, è possibile impostarli in modo diverso nel gruppo delle impostazioni **Lamiera** nella finestra di dialogo *Impostazioni*.

Figura 116: Lamiera

Attivare o disattivare la visualizzazione dei colori delle caratteristiche:

- Fare clic sul pulsante dello strumento **Colora Caratteristiche** () sulla barra degli strumenti **Lamiera**.
- Fare clic sul pulsante dello strumento **Colora Caratteristiche** () sulla barra multifunzione **Lamiera**.
- Scegliere **Colora Caratteristiche** nel menu **Lamiera**.

Esercitazione: Colorazione delle Caratteristiche nelle Lamiere in BricsCAD

21.5.5 Il Navigatore Meccanico per la Lamiera

21.5.5.1 Panoramica

Il Navigatore Meccanico si apre automaticamente quando si passa all'area di lavoro **Meccanico**.

Per aprire manualmente il Navigatore Meccanico eseguire le seguenti operazioni:

1 Spostare il cursore su una barra degli strumenti, quindi fare clic con il tasto destro del mouse.
Viene visualizzato un menu contestuale

:

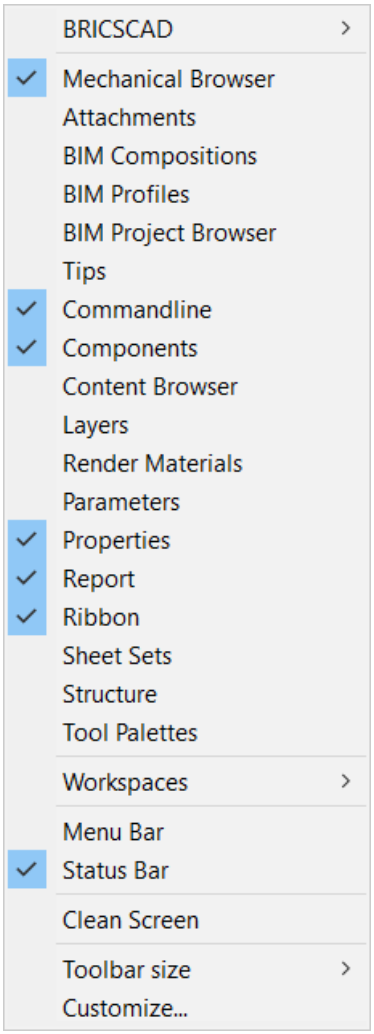


Figura 117: Menù contestuale

2 Scegliere **Navigatore Meccanico**.

21.5.5.2 Contesto della Lamiera

Se si lavora con una parte in lamiera (se si è creata una caratteristica lamiera), le proprietà del nodo radice nel **Navigatore Meccanico** contengono la sezione "Lamiera" (anche chiamata Contesto Lamiera associato al documento).

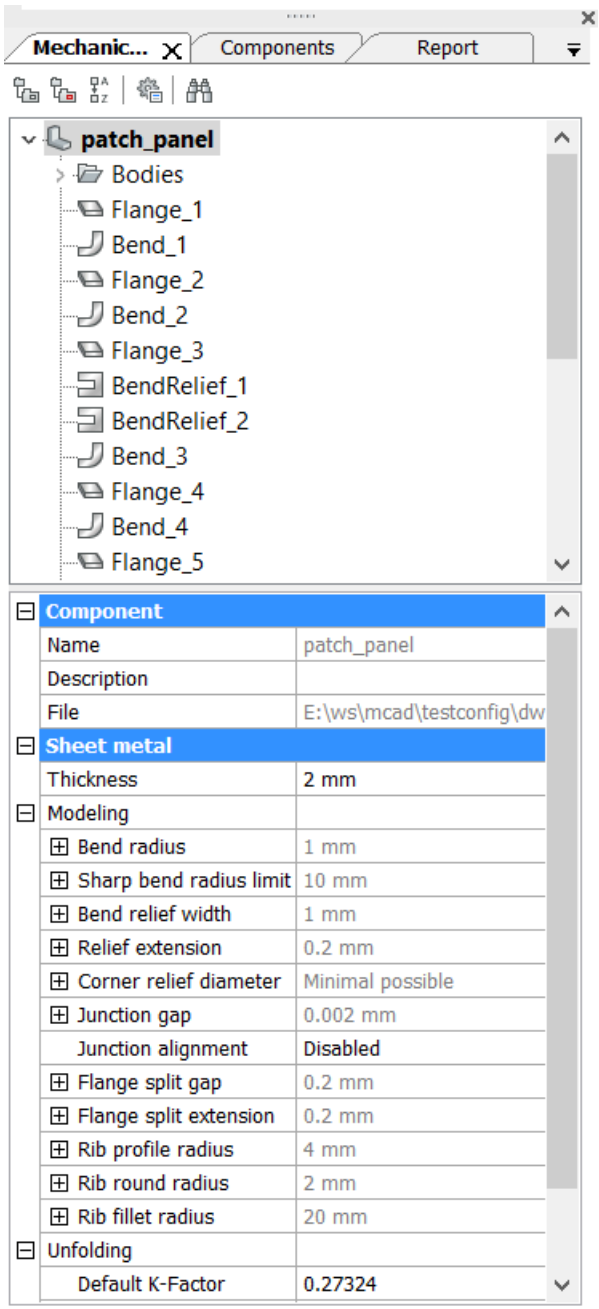


Figura 118: Navigatore Meccanico

Le Proprietà della Lamiera sono rappresentate come campi raggruppati.

Il campo **Spessore** controlla lo spessore del materiale dei solidi della lamiera che deve essere uguale per tutti i solidi del documento.

Il gruppo Modellazione svolge due ruoli:

- Offre impostazioni predefinite per i comandi della Lamiera, come *SMFLANGEBASE*, *SMEXTRUDE*, *SMFLANGEEDGE*, *SMFLANGEBEND*, *SMLOFT* e *SMCONVERT* in un modello appena creato. I parametri iniziali vengono salvati nel disegno, il che consente di creare un gruppo di template .dwt per ogni tipo di modello di lamiera creato in un'azienda.
- Le nuove caratteristiche create dai comandi derivano di solito le loro proprietà dal Contesto della Lamiera. Ad esempio, il valore predefinito della proprietà **Tipo Raggio** della caratteristica di piegatura è "Usa valore globale". Pertanto, quando si modifica il **Raggio Piegature** nel Contesto della Lamiera, tutte le caratteristiche di piegatura con la proprietà derivata ne modificheranno il raggio. Quindi ne deriva il fatto che le proprietà della Contesto della Lamiera consentono di modificare a livello globale le proprietà della parte in lamiera.

Il gruppo **Spiegamento** delle proprietà del Contesto della Lamiera elenca alcune proprietà dei comandi *SMUNFOLD*, *SMEXPORT2D* e *SMASSEMBLYEXPORT*.

Il Contesto della Lamiera viene creato quando la prima parte della lamiera viene visualizzata nel documento ed è inizializzata dalle impostazioni elencate nei Parametri Iniziali. I flussi di lavoro tipici per la creazione di parti in lamiera sono elencati in Metodi di Progettazione della Lamiera. Il Contesto della Lamiera viene rimosso quando tutte le caratteristiche presenti nel documento vengono dissolte dal comando *SMDISSOLVE*.

Nota: Se selezioni tutte le parti di lamiera nel documento e le cancelli, il Contesto della Lamiera viene mantenuto, nonostante non vi siano solidi in lamiera e non ci sono elementi in lamiera.

21.5.5.3 Lavorare con le caratteristiche

Tutte le caratteristiche di una parte di lamiera vengono elencate nel **Navigatore Meccanico**. Quando si seleziona una caratteristica nel **Navigatore Meccanico**, le sue facce vengono evidenziate nello spazio modello.

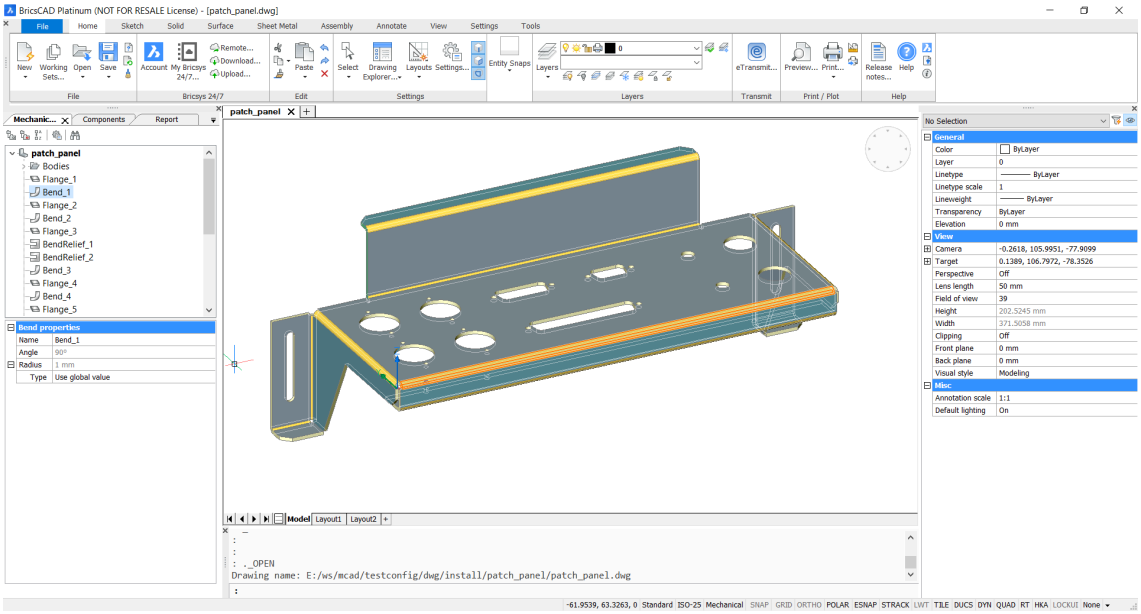


Figura 119: Navigatore Meccanico

Nota: Il gruppo di proprietà **Lamiera** è stato modificato in proprietà **Piegatura**. Cliccando sulla caratteristica nel Navigatore Meccanico è possibile:

- Individuare la caratteristica sulla parte evidenziandone la geometria.
- Vedere le proprietà di una determinata caratteristica

21.5.5.4 Cambiare lo spessore

Per modificare lo spessore di una parte di lamiera:

1 Selezionare il nodo principale nel **Navigatore Meccanico**.

2 Digitare un valore nel campo **Spessore**.

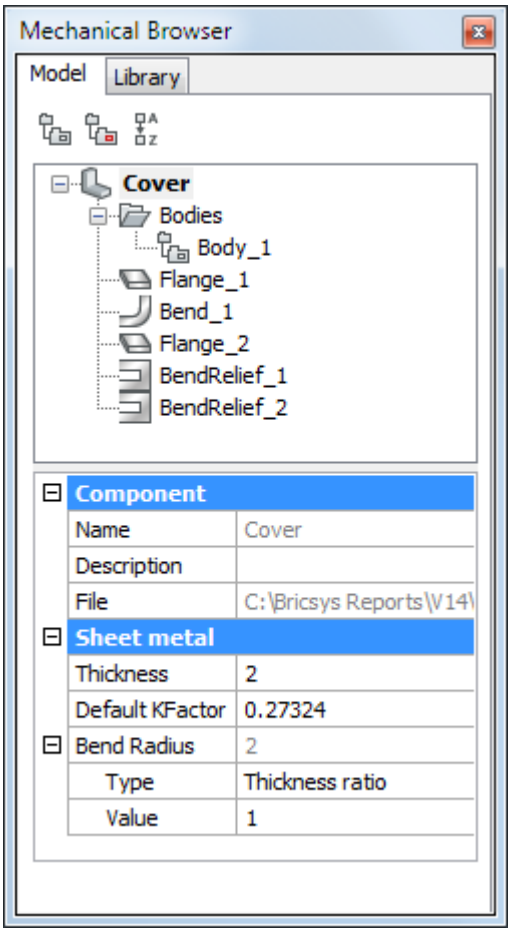
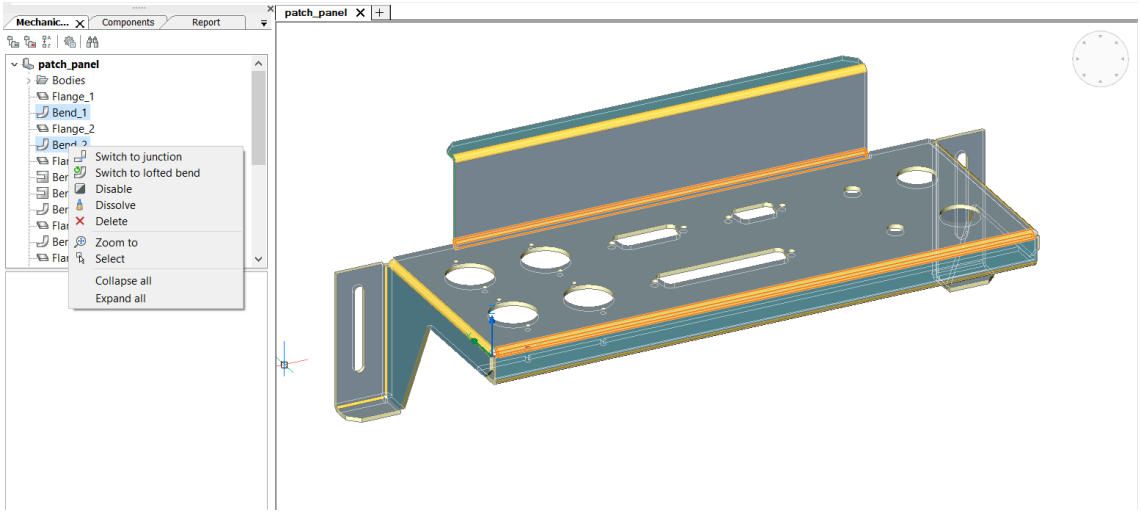


Figura 120: Spessore

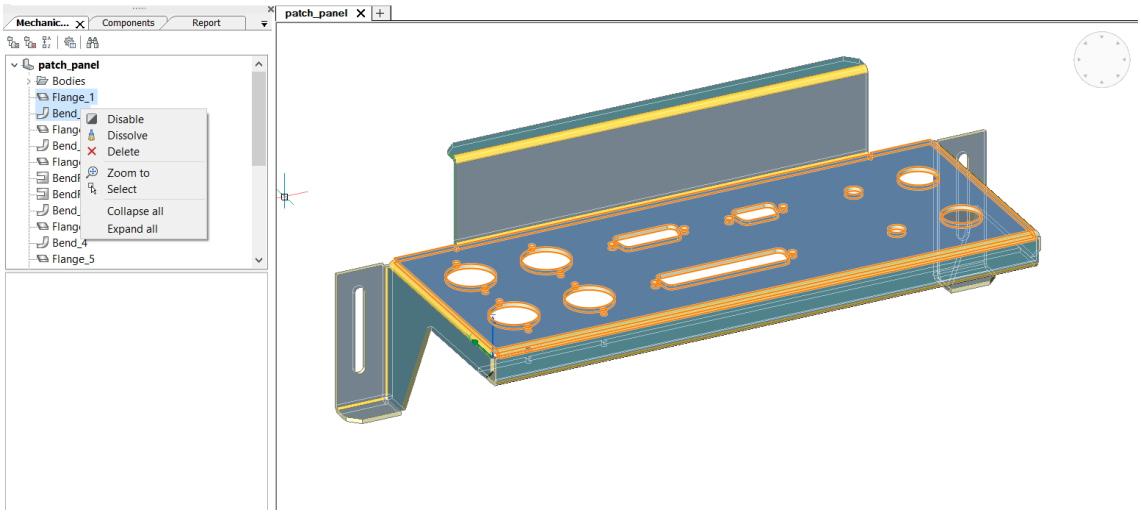
21.5.5.5 Operazioni sulle caratteristiche

Il **Navigatore Meccanico** fornisce un accesso rapido alle operazioni su una determinata caratteristica della lamiera, l'elenco dipende dal tipo di caratteristica. L'elenco delle operazioni dipende dalle caratteristiche selezionate e contiene le operazioni disponibili per la selezione.

Selezionando due piegature si ottengono operazioni per la caratteristica piegatura:



Selezionando una flangia ed i limiti della piegatura, vengono elencate le operazioni disponibili per entrambe le caratteristiche:



21.5.5.6 Comprendere la differenza tra Disattiva, Dissolvi ed Elimina

Cliccando il pulsante destro del mouse su una caratteristica nel **Navigatore Meccanico**, e selezionando **Dissolvi** dal menu contestuale la caratteristica selezionata viene rimossa dalla parte, ma manterrà la sua geometria. Tuttavia, l'intento di progettazione (relazioni spaziali e parametriche tra le facce della caratteristica) associato alla geometria di una caratteristica dissolta viene rimosso.

Se il tuo intento è temporaneo per dimenticarti di una determinata caratteristica, puoi scegliere l'opzione **Disabilita**. La caratteristica rimarrà nel **Navigatore Meccanico**, ma la parte si comporterà come se la caratteristica sia fosse stata dissolta. Al contrario di dissolvi è molto utile, in quanto la caratteristica si aggiorna ancora sulle modifiche della geometria e puoi attivarla quando vuoi, evitando di ricrearla o riconoscerla con il comando *SMCONVERT*.

Alcune caratteristiche della Lamiera supportano l'operazione di **Elimina** nel **Navigatore Meccanico**, che è l'analogo del comando *SMDELETE*. In questo caso, la caratteristica viene rimossa dal Navigatore Meccanico e la geometria viene modificata a seconda del tipo di caratteristica.

21.5.6 Lamiera Globale e Proprietà Caratteristiche

21.5.6.1 Panoramica

Le caratteristiche della lamiera e i menu contestuali si trovano nel **Navigatore Meccanico**. Vedi Navigatore Meccanico per la Lamiera per comprendere i principi generali. Questo articolo descrive le proprietà e i menu di scelta rapida particolari per ogni tipo di nodo nel **Navigatore Meccanico**.

Quando si lavora con parti in lamiera, cliccando sul nodo principale nel **Navigatore Meccanico**, vengono visualizzate le proprietà dell'intero disegno. A volte queste proprietà sono riferite al contesto globale della Lamiera associato al disegno: ogni parte di Lamiera condivide questo contesto per impostazione predefinita.

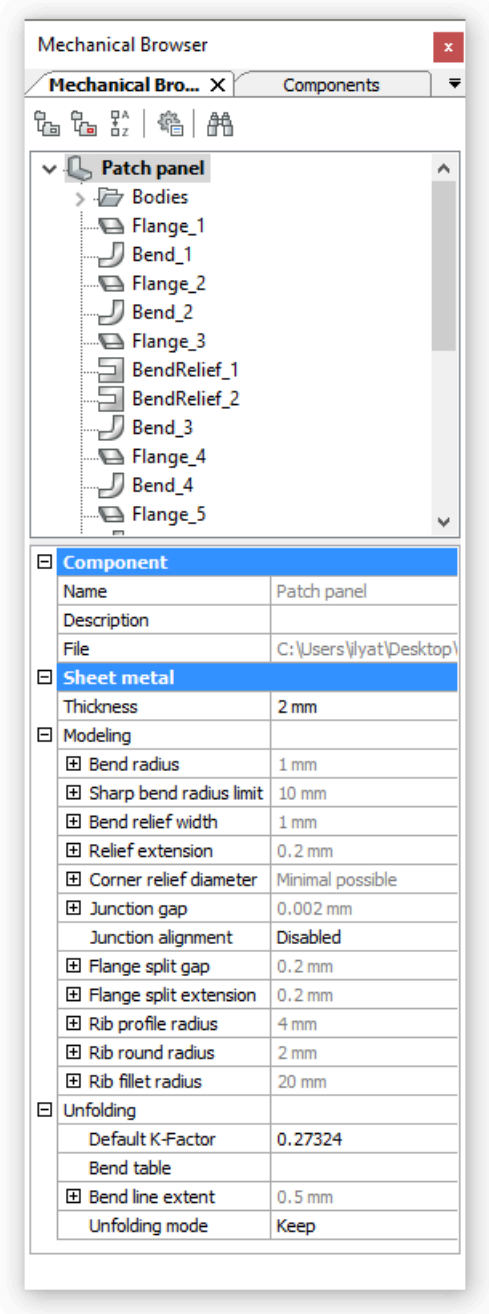


Figura 121: Navigatore Meccanico

A partire da BricsCAD V21, è possibile assegnare un contesto locale di Lamiera per un dato solido, consentendo quindi, di definire parti in lamiera con diversi parametri di progettazione e, inoltre, di modificare lo spessore della singola parte. Prima di BricsCAD V21, tutte le parti nello spazio di modellazione dovevano avere lo stesso spessore, costringendo l'utente a distribuire le parti con uno spessore diverso in diversi blocchi o componenti. Anche gli scenari con più corpi importati che coinvolgono *SMCONVERT* e *SMASSEMBLYEXPORT* beneficiano di questa nuova funzionalità. Un altro vantaggio è che il contesto locale della Lamiera può definire la propria Tabella di Piegatura.

21.5.6.2 Passare da un contesto solido a quello locale

Eseguire le seguenti operazioni:

1 Selezionare il corpo nel **Navigatore Meccanico**.

2 Nella sezione Lamiera, cambiare **Tipo contesto** da **Contesto globale** a **Contesto per solido**.

Nota: Il comando *SMCONVERT*, in caso di input multipli con spessori diversi, imposta automaticamente alcuni contesti su locale.

Le proprietà globali della lamiera sono posizionate nella parte inferiore del **Navigatore Meccanico**.

Tabella 40: Proprietà Globali

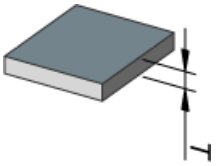
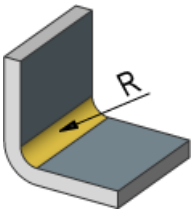
Proprietà Globali	Descrizione
Spessore	Spessore globale del materiale della lamiera. 
Raggi Piegature	Raggio predefinito delle caratteristiche di piegatura; può essere modificato anche per singola piegatura. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Limite del raggio di piegatura acuta	Raggio di piegatura massimo per considerare una piegatura acuta (normale). Utilizzato nei comandi *SMCONVERT* e *SMRELIEFCREATE* per distinguere tra piegature normali e loft. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore.

Tabella 40: Proprietà Globali

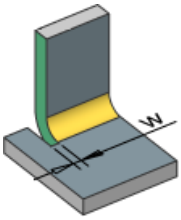
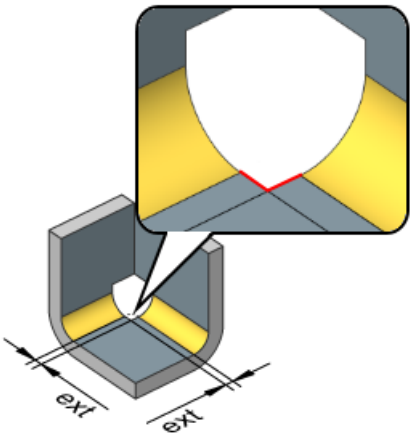
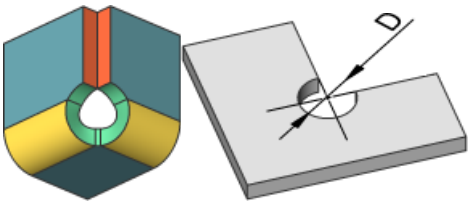
Proprietà Globali	Descrizione
Larghezza gola di scarico	Larghezza del taglio in prossimità di una singola piega. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Estensione scarico	Dimensione dell'estensione della piegatura o dello scaricato d'angolo. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Diametro scarico d'angolo	Diametro predefinito di uno scarico d'angolo circolare. **Tipo**: valore minimo o assoluto. 

Tabella 40: Proprietà Globali

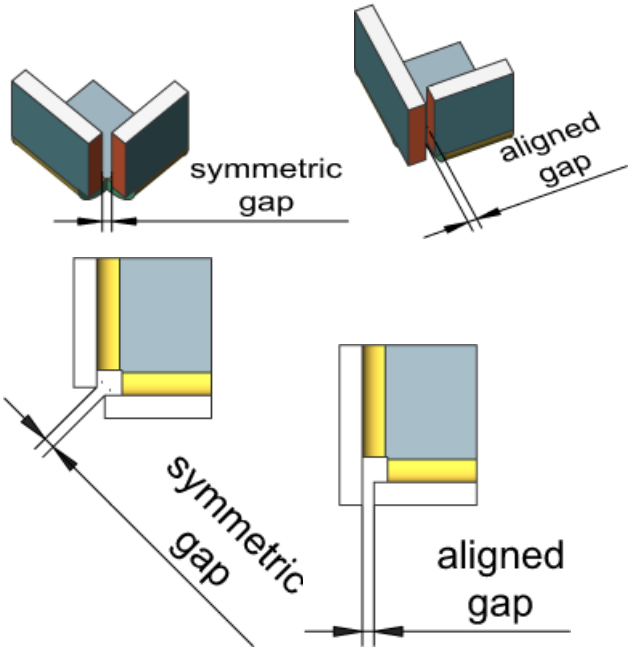
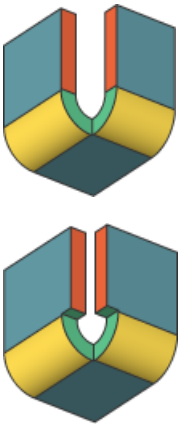
Proprietà Globali	Descrizione
Vuoto giunzione	Spazio tra 2 flange adiacenti. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Allineamento della giunzione	Controlla l'allineamento delle facce della giunzione con le corrispondenti facce dello scarico d'angolo. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. • **Allineamento della giunzione**: abilitato o disabilitato  **Figura 123:** Disabilitato **Figura 122:** Abilitato

Tabella 40: Proprietà Globali

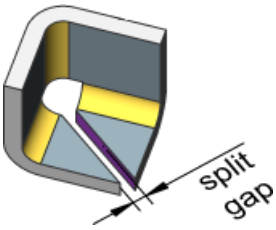
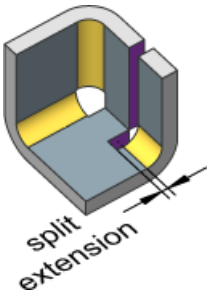
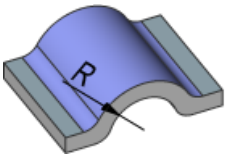
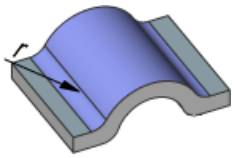
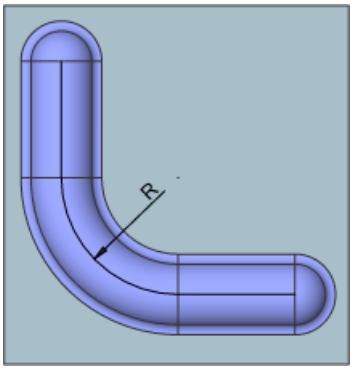
Proprietà Globali	Descrizione
Divario taglio ad Angolo	Spazio tra le facce dello spessore di una flangia o la piegatura adiacente durante la divisione, vedere *SMSPLIT*. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Estensione taglio ad angolo	Controlla la profondità del taglio di una flangia adiacente in caso di divisione della flangia. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Raggio nervatura profilo	Raggio dell'utensile, che crea la nervatura. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 
Raggio nervatura arrotondata	Raggio del raccordo tra la flangia e una nervatura. • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore. 

Tabella 40: Proprietà Globali

Proprietà Globali	Descrizione
Raggio di raccordo della nervatura	Raggio della curva di controllo della nervatura. Vedere Lavorare con le Caratteristiche Forma e Nervatura. • **Tipo**: rapporto raggio profilo o valore assoluto. • **Valore**: valore numerico. 
Spiegamento	Parametri per la rappresentazione spiegata.
Fattore-K predefinito	Fattore-K per il raggi delle piegature uguale allo spessore. Vedere Fattore K.
Tabella di piegatura	Visualizza la finestra di dialogo Apri per le tabelle di piegatura in formato file CSV.
Limite linea di piegatura	Controlla fino a che punto una linea di piegatura si estende oltre il confine della piegatura (sono consentiti valori zero e negativi). • **Tipo**: rapporto spessore o valore assoluto. • **Valore**: entrambi rapporto o valore.
Modalità spiegamento della caratteristica forma	Controlla l'aspetto delle caratteristiche forma nella rappresentazione spiegata. • **Mantieni**: conserva la rappresentazione 3D della caratteristica forma. • **Rimuovi**: rimuove completamente le caratteristiche forma dalla rappresentazione sviluppo in 2D. • **Proietta**: proietta tutti gli spigoli delle caratteristiche forma come entità 2D. • **Contorno**: proietta solo il contorno delle caratteristiche forma come entità 2D. • **Simbolo**: posiziona i simboli 2D nei punti di inserimento nella rappresentazione spiegata.

Le proprietà delle caratteristiche vengono visualizzate automaticamente quando si seleziona una caratteristica nel **Navigatore Meccanico**.

Cliccare il pulsante destro del mouse su una caratteristica per visualizzare il menu contestuale.

Tabella 41: Menu Contestuali delle Caratteristiche

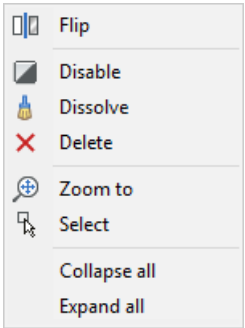
Menu Contestuali delle Caratteristiche	Descrizione
	Strumenti comuni: • **Disabilita**: disabilita temporaneamente una caratteristica e salta il suo vincolo. • **Dissolvi**: rimuove una caratteristica ma ne conserva la geometria. L'intento progettuale associato alla geometria di una caratteristica dissolta verrà rimosso. • **Elimina**: Vedi il comando *SMDELETE*. Rimuove le caratteristiche dal Navigatore Meccanico ed elimina la geometria corrispondente nel modello. • **Zoom a**: ingrandire la caratteristica selezionata. • **Comprimi tutto**: collassa il componente principale inclusi tutti i componenti e sottocomponenti. • **Espandi tutto**: espande il componente principale e tutti i componenti e sottocomponenti.
Caratteristica Flangia	Cliccare il pulsante destro del mouse su una caratteristica flangia:  **Inverti**: scambia i lati di una flangia selezionata in modo che la faccia di riferimento si trovi sul lato geometrico opposto della flangia. Questa operazione sposta la flangia in base allo spessore, per preservare la posizione della faccia di riferimento.

Tabella 41: Menu Contestuali delle Caratteristiche

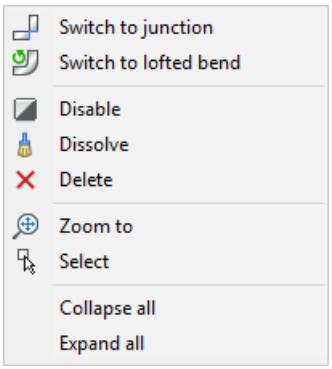
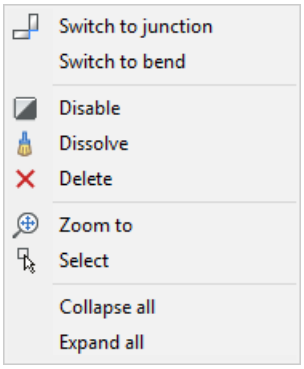
Menu Contestuali delle Caratteristiche	Descrizione
Caratteristica Piegatura	Cliccare il pulsante destro del mouse sulla caratteristica piegatura:  • **Commuta in Giunzione**: passa a un tipo di caratteristica giunzione con uno spazio tra le flange adiacenti. • **Commuta in piegatura loft**: passa a una caratteristica piegatura loft.
Caratteristica Piegatura Lofted	Cliccare il pulsante destro del mouse su una caratteristica piegatura loft:  • **Commuta in Giunzione**: passa a un tipo di caratteristica giunzione con uno spazio tra le flange adiacenti. • **Commuta in piegatura**: passa alla caratteristica piegatura.

Tabella 41: Menu Contestuali delle Caratteristiche

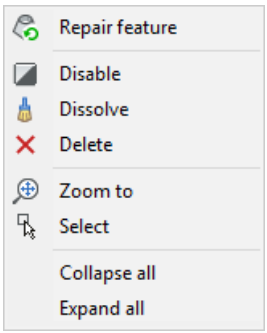
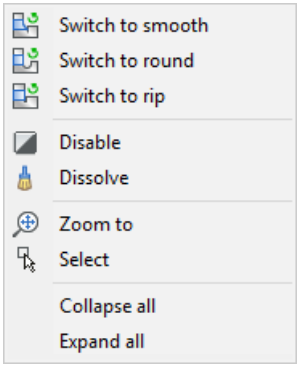
Menu Contestuali delle Caratteristiche	Descrizione
Caratteristica Piegatura Errata	Cliccare il pulsante destro del mouse su una caratteristica piegatura errata:  **Ripara caratteristica**: ripara una caratteristica piegatura errata creando una caratteristica piegatura con geometria corretta, se possibile.
Caratteristica Gola di Scarico	Cliccare il pulsante destro del mouse sulla caratteristica gola di scarico:  **Commuta a uniforme/asolato/lacerato**: commuta una gola di scarico di piegatura sul tipo di caratteristica selezionata.

Tabella 41: Menu Contestuali delle Caratteristiche

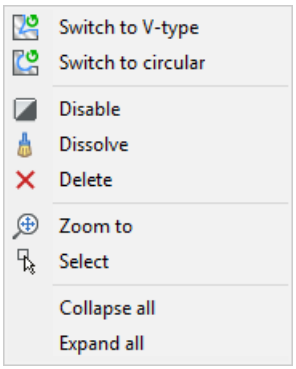
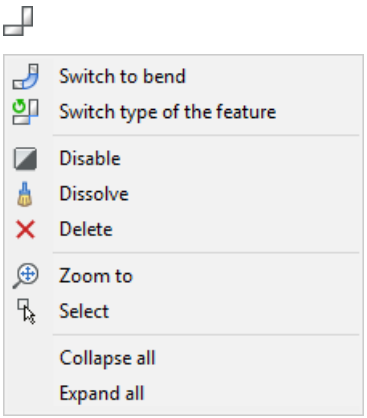
Menu Contestuali delle Caratteristiche	Descrizione
Caratteristica Scarico d'Angolo	Cliccare il pulsante destro del mouse sulla caratteristica scarico d'angolo:  **Commuta in tipo-V/cerchio**: consente di commutare una caratteristica scarico d'angolo con il tipo di caratteristica selezionato.
Caratteristica Giunzione	Fare clic con il tasto destro del mouse sulla caratteristica giunzione:  • **Commuta in piegatura**: passa alla caratteristica piegatura. • **Commuta il tipo di caratteristica**: commuta la modalità di allineamento tra simmetrica e allineata con una delle flange.

Tabella 41: Menu Contestuali delle Caratteristiche

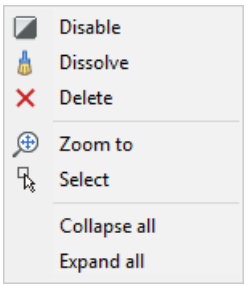
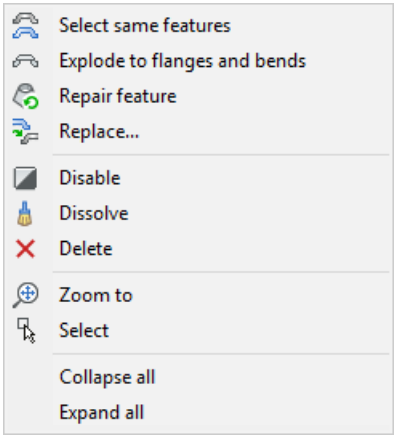
Menu Contestuali delle Caratteristiche	Descrizione
Funzione Taglio ad Angolo	Cliccare il tasto destro del mouse sulla caratteristica taglio ad angolo: 
Caratteristica Forma	Cliccare il tasto destro del mouse sulla caratteristica forma:  • **Selezionare le stesse caratteristiche**: seleziona tutte le caratteristiche forma dello stesso tipo nel modello. • **Esplodi in flange e piegature**: esplode una caratteristica forma in un insieme di flange e piegature, se possibile. • **Ripara caratteristica**: ricalcola il punto di inserimento di una caratteristica forma. • **Sostituisci ...:** visualizza la finestra di dialogo Apri per scegliere una caratteristica forma per la sostituzione.

Tabella 41: Menu Contestuali delle Caratteristiche

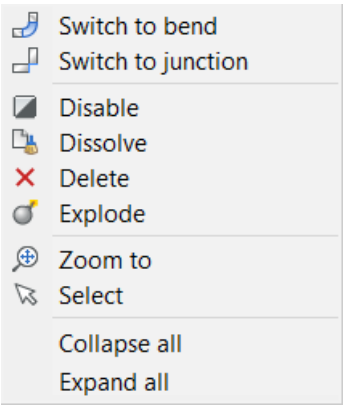
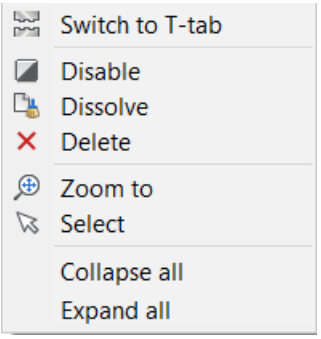
Menu Contestuali delle Caratteristiche	Descrizione
Caratteristica Linguetta	Cliccare il tasto destro del mouse sulla funzione linguetta:  • **Commuta in piegatura**: converte una caratteristica linguetta in una caratteristica piegatura • **Commuta in giunzione**: converte una caratteristica linguetta in una caratteristica giunzione
Caratteristica giunzione T	Cliccare il pulsante destro del mouse sulla caratteristica giunzione a T:  **Commuta in Linguetta-a-T**: converte una caratteristica giunzione a T in una caratteristica linguetta a T.

Tabella 41: Menu Contestuali delle Caratteristiche

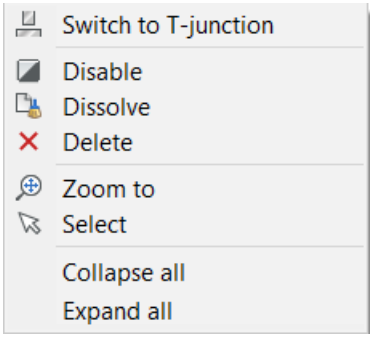
Menu Contestuali delle Caratteristiche	Descrizione
Caratteristica Linguetta a T	Cliccare il pulsante destro del mouse sulla caratteristica linguetta a T:  **Commuta in Giunzione-a-T**: converte una caratteristica linguetta a T in una caratteristica giunzione a T

Tabella 42: Proprietà Caratteristica

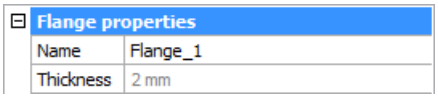
Proprietà Caratteristica	Descrizione
	Proprietà Comuni: **Nome**: il nome della caratteristica di lamiera selezionata. Opzionalmente, digitare un nome per la caratteristica. Il nome predefinito di una caratteristica è il tipo di caratteristica più un numero incrementale, ad es Flangia_1, Flangia_2, ...
Caratteristica Flangia	Cliccare su una caratteristica flangia:  **Spessore**: spessore attuale. Il valore è di sola lettura e corrisponde alla proprietà dello spessore del componente lamiera.

Tabella 42: Proprietà Caratteristica

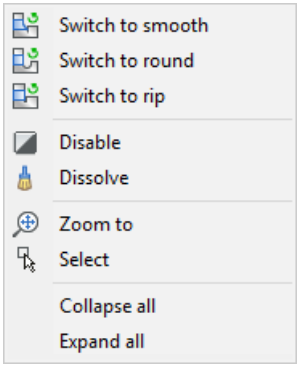
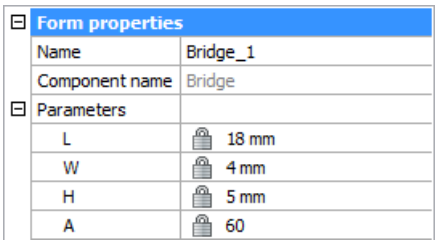
Proprietà Caratteristica	Descrizione
Caratteristica Piegatura	Cliccare su una caratteristica piegatura: • **Angolo**: angolo corrente della caratteristica piegatura; può essere modificato dal comando *SMFLANGEROTATE*. • **Raggio**: raggio attuale della piegatura. • **Tipo di raggio**: - Rapporto spessore: rapporto dello spessore moltiplicato per un coefficiente - Valore assoluto: il valore globale (proprietà del componente lamiera principale).
Gola di scarico	Fare clic su una caratteristica gola di scarico:  • **Tipo scarico**: tipo di scarico corrente; rettangolare, circolare, uniforme o lacerato. • **Larghezza**: larghezza del taglio vicino alla piegatura; globale o rapporto spessore o assoluto.

Tabella 42: Proprietà Caratteristica

Proprietà Caratteristica	Descrizione
Scarico d'Angolo	Cliccare una caratteristica scarico d'angolo: **Corner relief properties** Name: CornerRelief_1 Type: Rectangular 2 bends Extension: 0.2 mm Type: Use global value • **Tipo scarico**: tipo scarico corrente; rettangolare 2 piegature o rettangolare o tipo-V o circolare • **Larghezza**: larghezza del taglio vicino all'angolo; o globale, rapporto di spessore o assoluto.
Caratteristica Giunzione	Cliccare su una caratteristica giunzione: **Junction properties** Name: Junction_1 Junction gap: 0.002 mm Type: Use global value Connection type: Symmetric Junction alignment: Use global value • **Vuoto giunzione**: spazio tra 2 flange adiacenti. • **Tipo di vuoto giunzione**: globale, rapporto spessore o valore assoluto. • **Tipo di connessione**: controlla la posizione relativa delle flange adiacenti simmetrico o adiacente a una delle flange. • **Allineamento della giunzione**: controlla la posizione delle facce di una caratteristica giunzione; - globale - abilitato (allineato con le facce corrispondenti della caratteristica di scarico d'angolo) - disattivato.
Funzione Taglio ad Angolo	Cliccare una caratteristica taglio ad angolo: **Miter properties** Name: Miter_1 Miter gap: 0.2 mm Type: Use global value • **Divario taglio ad angolo**: spazio tra le facce laterali del taglio ad angolo. • **Tipo**: rapporto spessore o valore assoluto.

Tabella 42: Proprietà Caratteristica

Proprietà Caratteristica	Descrizione
Caratteristica Forma	Cliccare su una caratteristica forma:  • **Nome Componente**: nome della definizione del componente della caratteristica forma. • **Parametri**: contiene i parametri di una caratteristica forma parametrica.
Caratteristica Smusso	Cliccare su una caratteristica smusso: • Tipo di smusso: definisce un tipo di smusso, a seconda della geometria. Valori possibili: Smusso su un lato, Smusso a 45°, smusso X e K. In base a un tipo di smusso, i parametri possono essere alterati. • Smusso su un lato: parametri di cianfrinatura su un lato di questo smusso. • Smusso ortogonale: parametri del lato ortogonale.

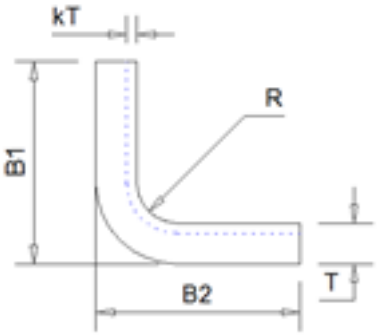
21.5.7 Fattore K

21.5.7.1 Panoramica

Quando una lamiera viene piegata con una pressa, il suo materiale si deforma, a causa della differenza della lunghezza complessiva di un foglio nello stato piegato rispetto al precedente stato non piegato. BricsCAD prende automaticamente in considerazione questa deformazione quando si appiattisce in

2D una parte di lamiera. Sono utilizzati due metodi per calcolarlo: uno semplificato (fattore K) e uno più avanzato (Tabella di Piegatura).

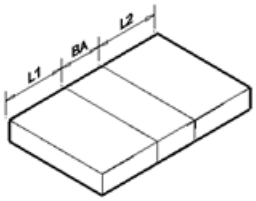
Le proprietà di deformazione del materiale derivante dalla piegatura sono basate sul presupposto che esiste una superficie interna della parte di lamiera che non si deforma quando il foglio di lamiera viene piegato. Dalle infinite superfici interne ne viene selezionata una, equidistante da un lato della piegatura. La superficie è locale per ogni piegatura, e per casi semplici può essere propagata su tutta la parte, come quella sull'immagine. Partiamo dal presupposto che questa superficie non viene stirata/allungata durante la piegatura.



T: spessore della parte in lamiera

R: raggio interno della piega

I fattore K (k) è il rapporto della posizione della superficie neutra rispetto lo spessore del materiale. Di conseguenza la superficie neutra si trova ad una distanza kT dalla superficie interna della piega. Il raggio di piegatura della superficie neutra è uguale a R + kT. Il fattore K corrisponde ad un semplice calcolo geometrico della posizione della superficie neutra. Lo stress ed altri sconosciuti (errore) fattori non sono presi in considerazione. Il Fattore K dipende da molti fattori, tra cui il tipo di materiale, il tipo di piegatura e la strumentazione. Il Fattore K è, in genere, compreso tra 0,3 e 0,5. Il valore predefinito del fattore K per un raggio di piegatura uguale allo spessore del materiale (T) è 0.27324.



Nel foglio sviluppato in 2d di una parte di lamiera, le flange (L1 e L2) non vengono allungate. La parte piegata è indicata come BA: bend allowance = tolleranza di piegatura. BA è uguale alla lunghezza della superficie neutra sviluppata:

BA = AngoloPiega * (R + kT)

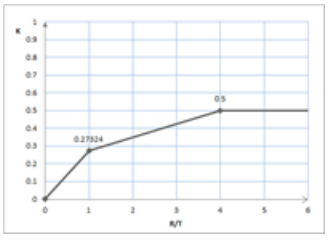
Infatti è difficile misurare il fattore K o il limite di tolleranza. La seguente formula permette di calcolare la Riduzione della Piega tangenziale (BD):

BD = B1 + B2 - Lunghezza Spiegata = B1 + B2 - (L1 + L2 + BA)

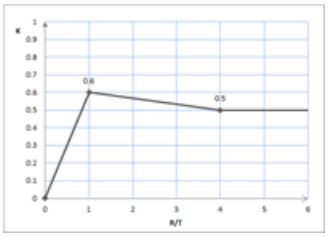
Nei casi semplici il valore del Fattore K può essere sostituito nel navigatore meccanico. Per la massima precisione deve essere prevista una tabella di piegatura contenente la riduzione della piega tangenziale. Per ogni parte di lamiera, si può specificare un Fattore K oppure usare quello predefinito.

21.5.7.2 Modificare il fattore K di una Parte in Lamiera

Per cambiare il fattore K per una parte di lamiera, selezionare il nodo principale del Navigatore Meccanico e digitare un valore nel campo **Fattore-K**. Il valore deve essere compreso nell'intervallo [0, 1], in quanto la superficie neutra si trova all'interno della parte di lamiera. In BricsCAD il fattore K è definito per il raggio di piegatura uguale allo spessore del materiale (R/T = 1). Per calcolare il valore del **Fattore K** per un raggio di piegatura arbitrario, BricsCAD utilizza una speciale tecnica di interpolazione, che si è rivelata valida per le applicazioni nell'industria della lamiera. L'immagine sotto mostra la strategia di interpolazione: per R/T < 1 l'interpolazione lineare viene eseguita tra 0 e 0.27324, per 1 < R/T < 4 il Fattore K viene interpolato linearmente tra 0.27324 e 0.5 e se R/T è superiore a 4, il Fattore K è costante ed uguale a 0.5. Fisicamente significa che se il raggio di piegatura è molto più grande rispetto allo spessore (almeno 4 volte), lo stiramento del materiale può essere trascurato.



Il solo parametro fattore K che si può controllare nel navigatore è per R/T. Significa che se impostiamo il valore a 0.6, la regola di interpolazione conduce al seguente risultato:

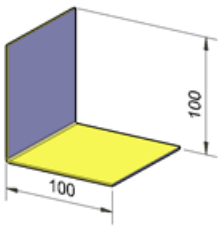


Pic: il Fattore-K variabile nel navigatore porta alla seguente curva efficace.

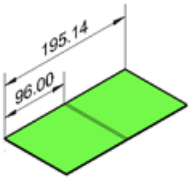
La caratteristica distintiva della curva è che variando R nel range (0, T], L1 + L2 + BA fornisce lo stesso valore. Ciò è spiegato dal fatto che per raggi inferiori allo spessore a cui si riferiscono sono per la progettazione del sistema piuttosto che per alcuni modelli fisici di processo, poiché durante la piegatura il raggio dello strumento della piegatrice è solitamente maggiore o uguale allo spessore.

Rivediamo l'esempio con il fattore K variabile mediante l'impostazione della curva di controllo (il raggio di piegatura viene mantenuto costante).

T = 2, R = 2, B1 = 100, B2 = 100



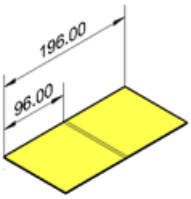
Impostazione di K=0 fornisce L1=L2=96, Lunghezza spiegata= 96, quindi BA è 3,14.



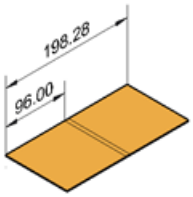
Questo valore rispetta esattamente la formula: un angolo piatto in radianti equivale a circa 1,57 (PI/2). Il valore predefinito del fattore K 0.27324 dà una Lunghezza Spiegata = 196

Oppure: L1=L2=96 (le flange non vengono deformate), il BA = 4 come possiamo calcolare dalle dimensioni (196 - 2*96) o dalla formula di tolleranza della piegatura:

BA = AngoloPiega * (R + kT) = 1.57 * (2 + 2*0.27324) = 4



Infine, massimizzando i risultati del fattore K in una Lunghezza Sviluppata = 198.28. Questo è il valore massimo che si può ottenere in questo esempio, poiché la superficie neutra è presa dal lato esterno della piega.



La Deduzione di Piegatura (BD) per k = 0.27324:

BD = B1 + B2 - Lunghezza Spiegata = 100 + 100 - 196 = 4

21.5.8 Tabella di Piegatura

21.5.8.1 Panoramica

Le tabelle di piegatura sono un modo più affidabile per esprimere le proprietà di deformazione del materiale rispetto il Fattore K. Quando si piega un foglio di un particolare materiale, è possibile misurare la sua lunghezza prima e dopo questo processo. E' possibile ripetere questa procedura per differenti angoli di piegatura, differenti raggi di piegatura e differenti spessori della lamiera e salvare le misurazioni in una tabella di piegatura. Quindi queste misurazioni possono essere prese in considerazione per calcolare la lunghezza corretta sviluppata in 2d per tutte le parti realizzate con lo stesso materiale.

In una tabella di piegatura, è possibile aggiungere il numero desiderato di risultati per angoli di piegatura differenti. Tuttavia, nella maggior parte dei casi, è sufficiente misurare la lunghezza per un angolo di 90 gradi. BricsCAD calcola automaticamente la lunghezza per altri angoli di piegatura usando un'affidabile tecnica di interpolazione.

Le tabelle di piegatura vengono memorizzate nel formato di file *.CSV (valori separati da virgola). È possibile creare tali tabelle in un formato ASCII con un editor di testo come il Blocco Note. Tuttavia, un modo più semplice ed affidabile è quello di creare una tabella in un foglio di calcolo, come Microsoft Excel o LibreOffice ed esportarlo in formato CSV.

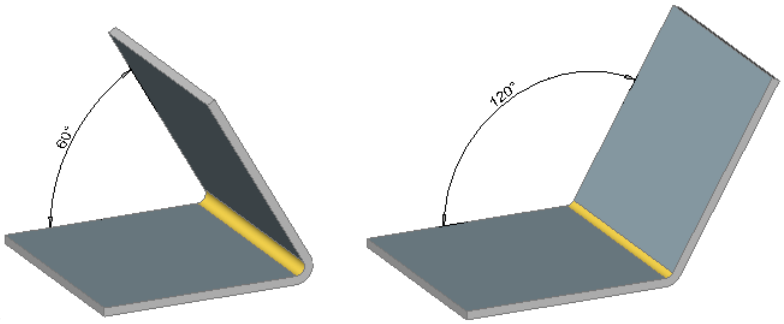
Le tabelle di piegatura in BricsCAD hanno la seguente struttura:

	A	B	C	D	E	F	G
1	BricsCAD						
2	Version	1					
3	AngleType	Internal					
4	LengthType	BendDeductionTangent					
5							
6	Thickness	1					
7	AngleValues		45	60	90	120	135
8	Radius	1	1	1.333	2	0.976	0.657
9	Radius	2	0.466	1.081	2.31	1.004	0.641
10	Radius	3	-0.069	0.828	2.621	1.033	0.624
11							
12	Thickness	1					
13	DieWidth	10					
14	AngleValues		45	60	90	120	135
15	Radius	1	1.225	1.533	2.15	1.076	0.732
16	Radius	2	0.616	1.214	2.41	1.071	0.691
17	Radius	3	0.006	0.895	2.671	1.066	0.649

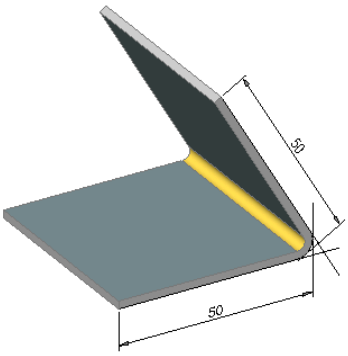
Figura 124: Esempio di tabella di piegatura

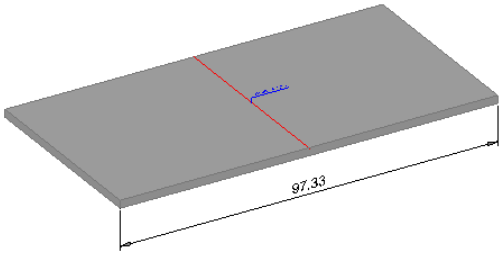
Si applicano le seguenti regole:

- Il contenuto della prima cella è "BricsCAD".
- **Version**: la versione attualmente supportata è 1.
- **AngleType**: Attualmente sono supportati solo gli angoli delle piegature interne:

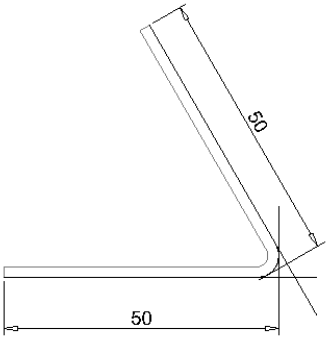


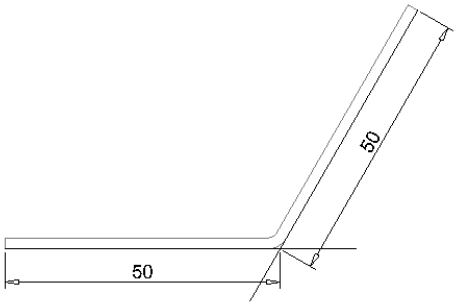
- **LengthType**: la semantica del valore nelle celle della tabella di piegatura corrispondente ad un particolare angolo e raggio di piegatura. Ora è supportata solo la deduzione della piegatura misurata dal punto di tangenza ("BendDeductionTangent"). Bend deduction (BD) è la differenza tra la somma delle lunghezze di 2 flange misurate al punto di tangenza su un modello 3D e la lunghezza dello stesso frammento nello stato spiegato:





Approfondimento: metodiche per calcolare la deduzione della piegatura tangenziale per angoli acuti e ottusi.





- All'interno dell'intestazione definita possono esistere diverse tabelle. Per ogni tabella, la prima chiave è lo **Thickness** (spessore) della parte in lamiera. La seconda chiave, **DieWidth** (RiduzioneLarghezza), è facoltativa. Le coppie **Thickness**/**DieWidth** devono essere uniche. Se la **DieWidth** non è definita, è consentito solo una chiave con valore **Thickness** univoco, ma è possibile aggiungere più valori di thickness. Quando si utilizza una tabella di piegatura per controllare il processo di piegatura, **Thickness** e **DieWidth**, definiti nella parte di lamiera, devono esistere nella tabella di piegatura. Se la tabella non viene trovata, viene utilizzato il criterio Fattore-K predefinito.
- Una volta trovata la tabella di piegatura in .CSV, viene utilizzata per spiegare la parte in lamiera. Utilizzando la tabella di piegatura di esempio qui sopra, verifichiamo alcuni casi:

  T = Thickness (spessore), R = Internal Bend Radius (raggio interno della piegatura), A = Bend Angle (angolo della piegatura)
  - T = 1, R = 2, A = 45. Successivamente, dopo lo spiegamento, otterrà BD = 0,466, perché sia R che A sono stati trovati nella tabella di piegatura per lo spessore fornito.
  - T = 1, R = 2.5, A = 45. A = 45 è stato trovato, ma R = 2,5 manca: viene utilizzata l'interpolazione di BD tra R=2 e R=3.

- T = 1, R = 2.5, A = 75. Né R né A sono presenti nella tabella: le seguenti celle vengono utilizzate per calcolare il valore di BD: (R=2, A=60), (R=3, A=60), (R=2, A=90) e (R=3, A=90).

L'algoritmo è utilizzato per calcolare l'interpolazione. Ad esempio, l'interpolazione lineare su valori adiacenti di BD realizzano risultati innaturali, che non corrispondono al reale risultato della piegatura. Inoltre tenere a mente che per un determinato T, R ed A non tutti i valori di BD hanno senso: un valore casuale potrebbe inserire la superficie neutra fuori dalla parte di lamiera, oppure espressa nel Fattore K (che può essere dedotto da BD), il fattore K può essere compreso nell'intervallo [0, 1]. In questo caso, verrà utilizzato il BD per il fattore K predefinito.

- Se si producono parti in lamiera con strumenti diversi, è possibile descriverli in una tabella di piegatura utilizzando il parametro **DieWidth**. Per comodità viene fornita una tabella di piegatura di esempio in **[Cartella Installazione BricsCAD]\Samples\Mechanical\bend_tables**.

  Nota: I diritti di scrittura di questa cartella potrebbero essere limitati. In tal caso, è necessario copiare le tabelle di piegatura in un'altra cartella in cui si dispone dei diritti di accesso completi.

21.5.8.2 Impostare la tabella di piegatura per la tua parte di lamiera:

1 Selezionare il nodo principale nel Navigatore Meccanico.

2 Selezionare il campo **Tabella di piegatura**.

3 Fare clic sul pulsante **Sfoglia** ( ).

4 Selezionare il file *.CSV.

5 Fare doppio clic o premere il pulsante **Apri**.

21.5.8.3 Aggiornare la tabella di piegatura impostata in precedenza:

1 Selezionare il nodo principale nel Navigatore Meccanico.

2 Selezionare il campo **Tabella di piegatura**.

3 Cliccare sul pulsante **Ricarica** ( ).

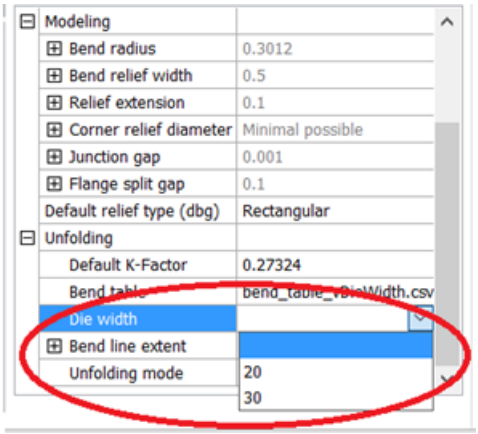
Se la tabella di piegatura non è assegnata al documento o non dispone della chiave secondaria **DieWidth**, il parametro **DieWidth** non può essere assegnato a una singola piegatura o a tutte le piegature. Per effettuare l'assegnazione, attenersi alla seguente procedura:

	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 Selezionare le chiavi secondarie **DieWidth** del file .csv e lo spessore corrispondente alla parte corrente. Nell'esempio fornito assumiamo che la parte abbia uno spessore di 1 mm e che il raggio di piegatura sia uguale a 1. Verrà calcolata la deduzione di piegatura con diversi strumenti: la prima sottotabella viene utilizzata quando lo strumento non è specificato, la seconda sottotabella è valida quando lo strumento **DieWidth** è uguale a 20 e l'ultima è per lo strumento con **DieWidth** uguale a 30.

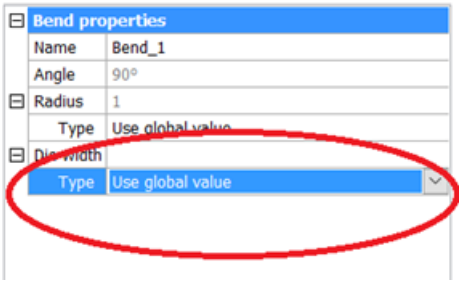
2 Assegnare la tabella di piegatura alla parte in lamiera.

3 Verificare che nel Navigatore Meccanico siano comparsi nuovi

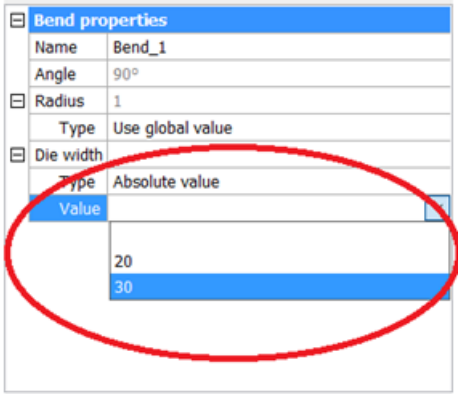


campi:

In questa impostazione globale è possibile selezionare qualsiasi valore predefinito **DieWidth** adatto allo spessore della parte. Il valore è assegnato per impostazione predefinita alle piegature della parte, che possono essere controllate dalle proprietà della piegatura (cliccare sulla piegatura nel Navigatore Meccanico):



4 Tuttavia, possiamo selezionare uno strumento specifico che verrà utilizzato per la piegatura data espandendo l'elenco a discesa:



5 Con la selezione data, lo spiegamento di Bend_1 utilizzerà una terza sottotabella dalla nostra tabella di piegatura per la deduzione della piegatura ed utilizzerà il valore 1.8. Per il resto delle piegature, a meno che **DieWidth** non sia impostato su di esse, verrà utilizzato il valore predefinito senza riduzione della larghezza, che attiverà prima la tabella di piegatura ed utilizzerà il valore 2.1 per 90 gradi con un raggio di curvatura uguale a 1.

Nota: Dopo aver assegnato **DieWidth** alle piegature e salvato il documento, il valore viene salvato nelle piegature. Tuttavia, la prossima volta che si aprirà il disegno, per assegnare il valore **DieWidth** ad un'altra piegatura, è necessario prima ricaricare la tabella di piegatura (premere il pulsante freccia verde), vedere Ricaricare la tabella di piegatura impostata in precedenza. Dopo il ricaricamento saranno disponibili i valori della tabella di piegatura.

Nota: A partire da BricsCAD V21 è possibile modificare le tabelle di piegatura per diversi solidi nello spazio modello definendo un contesto locale della Lamiera per ciascun solido.

21.5.9 Impostazioni della Lamiera

21.5.9.1 Panoramica

La sezione Lamiera nella finestra di dialogo *Impostazioni* controlla le diverse proprietà della lamiera all'interno di BricsCAD. Le proprietà sono suddivise in quattro gruppi: **Comandi**, **Colori caratteristiche**, **Parametri iniziali** e **Modellazione**.

Figura 125: Impostazioni: Lamiera

Nota: La maggior parte delle impostazioni di lamiera metallica sono memorizzate nel disegno. Per risultati ottimali, inizia sempre con uno dei file modello DWT della lamiera o creane uno tuo.

21.5.9.2 Comandi

La sezione **Comandi** contiene impostazioni che controllano il comportamento dei vari comandi della lamiera.

Nota: Prima di BricsCAD V19 questi controlli erano disponibili come opzioni all'interno dei singoli comandi. Sono stati spostati nella finestra di dialogo delle impostazioni per semplificare l'interfaccia di comando.

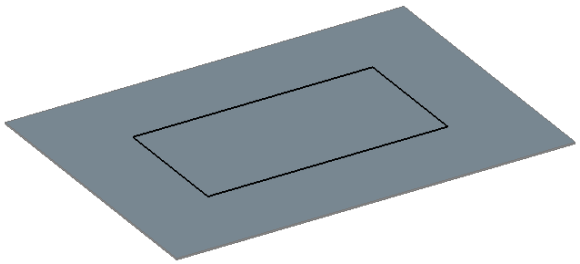


Figura 126: Impostazioni: Comandi

I layer utilizzati per lo Spiegamento di Lamiere determinano l'aspetto delle parti spiegate.

21.5.9.3 Colori caratteristiche

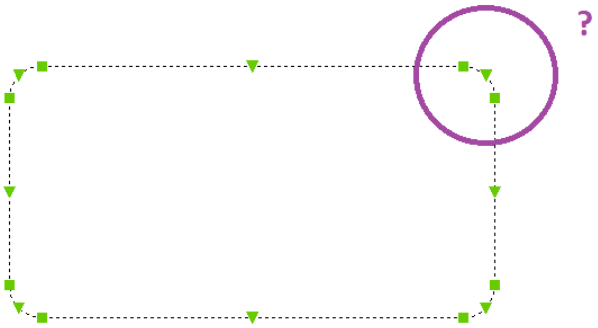


Figura 127: Impostazioni: Colori caratteristiche

Le impostazioni sotto **Colori caratteristiche** distinguono visivamente i diversi tipi di caratteristiche della lamiera in base al colore. È possibile regolare i colori per ogni tipo di caratteristica.

21.5.9.4 Parametri iniziali

Questi parametri sono responsabili dell'inizializzazione del **Contesto di Lamiera** associato a un disegno.

Modellazione

Le impostazioni sotto Modellazione controllano il comportamento delle caratteristiche della lamiera. Le impostazioni sono raggruppate per tipo di caratteristica.

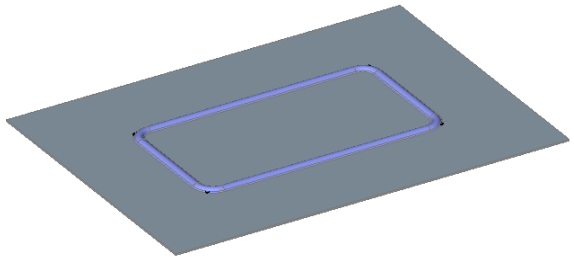


Figura 128: Impostazioni: Modellazione

Spiegamento

Le impostazioni di spiegamento controllano il comportamento del **Valore del Fattore K**, della **Modalità di spiegamento della caratteristica forma** e del **tipo/valore dell'estensione della linea di piegatura**.

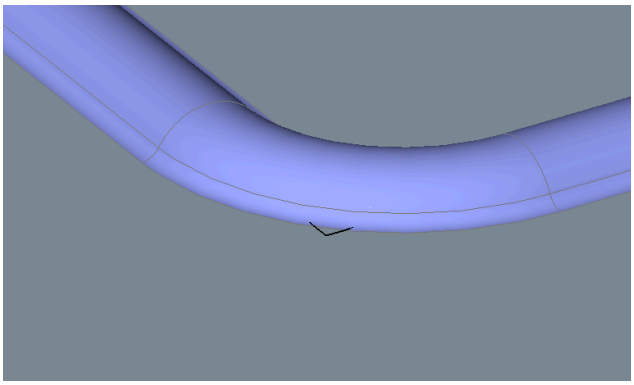


Figura 129: Impostazioni: Spiegamento

21.5.9.5 Modellazione

Le impostazioni sotto Modellazione includono la preferenza utente di *SMSMARTFEATURES* la quale controlla l'aggiornamento locale e intelligente della geometria dopo l'esecuzione dei comandi della Lamiera.

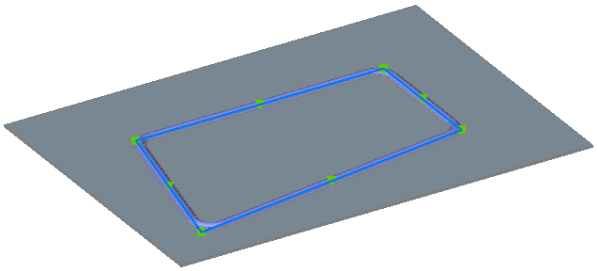


Figura 130: Impostazioni: Modellazione

Esercitazione: Parametrizzazione di parti in lamiera

21.5.10 Creare Flange

21.5.10.1 Panoramica

Una flangia è la caratteristica principale della maggior parte delle parti in lamiera. In BricsCAD puoi creare flange utilizzando molte tecniche diverse:

- Creare una flangia di base da un profilo 2D chiuso, che ne definisce il contorno.
- Creare una o più flange di bordo tirando gli spigoli delle flange esistenti.
- Creare più flange contemporaneamente estrudendo una polilinea 2D (ogni segmento lineare della polilinea diventa una flangia).

Suggerimento: Esistono anche altre tecniche che è possibile utilizzare per creare flange. Queste sono descritte in:

- Le flange possono essere create come elementi di parti in lamiera loft.
- Le flange possono essere riconosciute nella geometria esistente.

A differenza dei MCAD basati sulla cronologia, BricsCAD applica gli stessi strumenti per modificare qualsiasi flangia in modo indipendente.

21.5.10.2 Creare Flangia di Base

In generale, si crea una flangia di base per avviare il progetto della parte in lamiera. Per creare una flangia di base, selezionare un profilo planare chiuso ed eseguire il comando *SMFLANGEBASE*.

Eseguire una delle seguenti operazioni:

- Fare clic sul pulsante **Crea Flangia di Base** ( ) sulla barra degli strumenti **Lamiera**.
- Scegliere **Crea Flangia di Base** nel menu **Lamiera**.
- Scegliere **Crea Flangia di Base** nel gruppo **Lamiera** del menu cursore **Quad**:

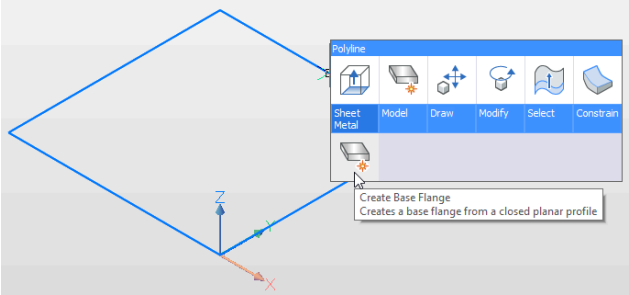
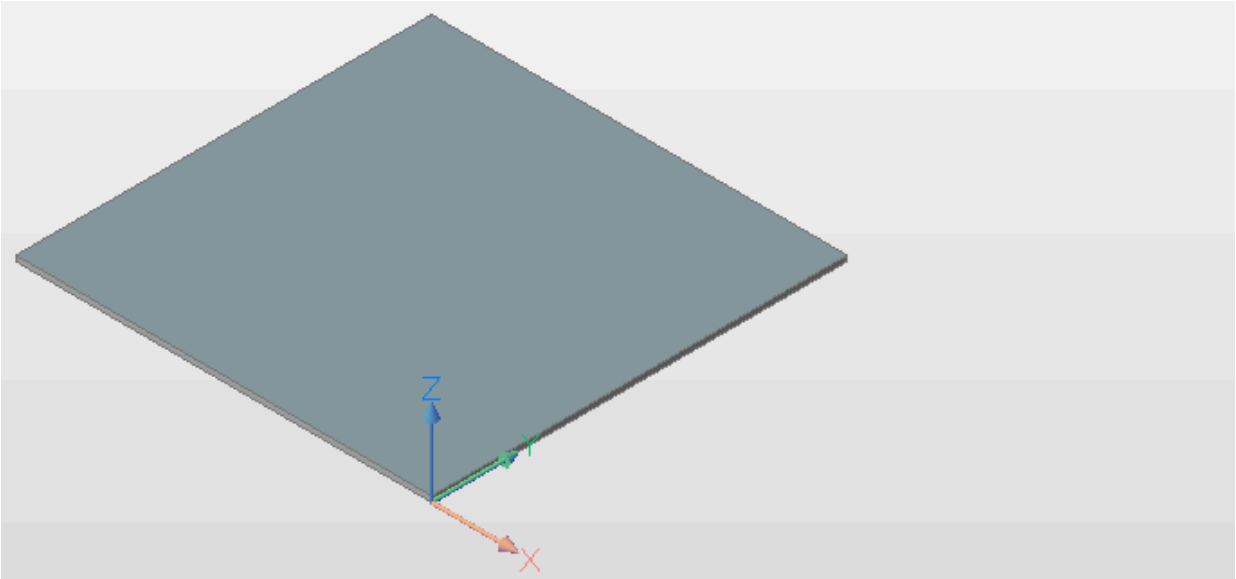


Figura 131: Creare Flangia di Base



Per creare una flangia di base, il comando *SMFLANGEBASE* estrude un profilo.

L'altezza della flangia di base corrisponde alla proprietà **Spessore** predefinito della parte in lamiera. Per modificare lo spessore della parte in lamiera, digitare il valore appropriato nel campo **Spessore** nel Navigatore Meccanico.

21.5.10.3 Creare Flangia da Spigolo

Puoi aggiungere un'ulteriore flangia alla tua lamiera:

1 Selezionare uno spigolo lineare.

2 Selezionare il comando *Crea Flangia da Spigolo* nella sezione **Lamiera** del menu del cursore **Quad**.

3 Spostare il puntatore del mouse per definire la lunghezza desiderata della flangia e l'angolo tra le 2 flange. È anche possibile utilizzare i campi delle dimensioni dinamiche per inserire valori precisi.

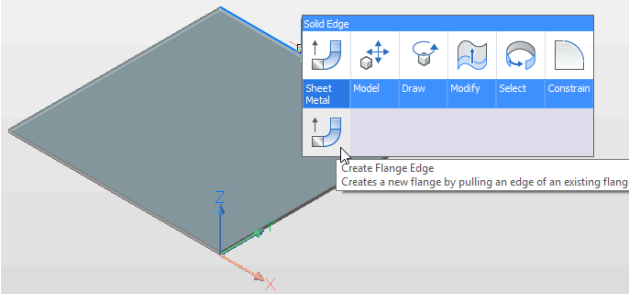
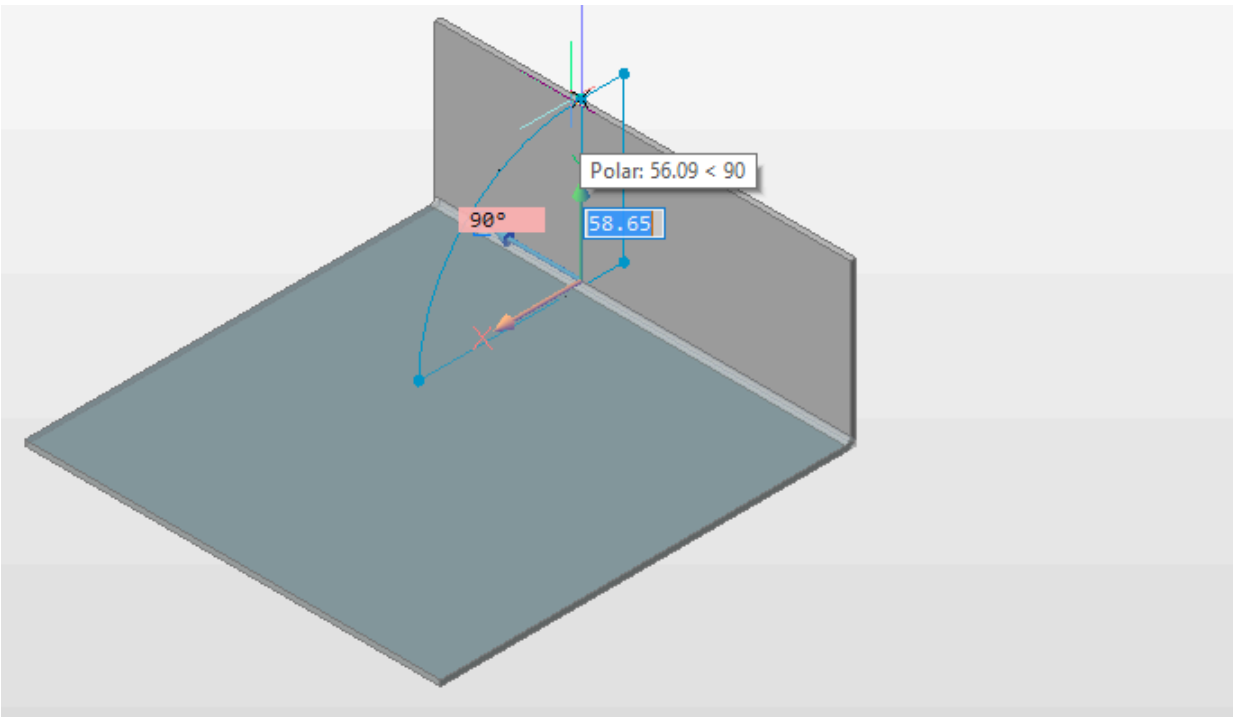
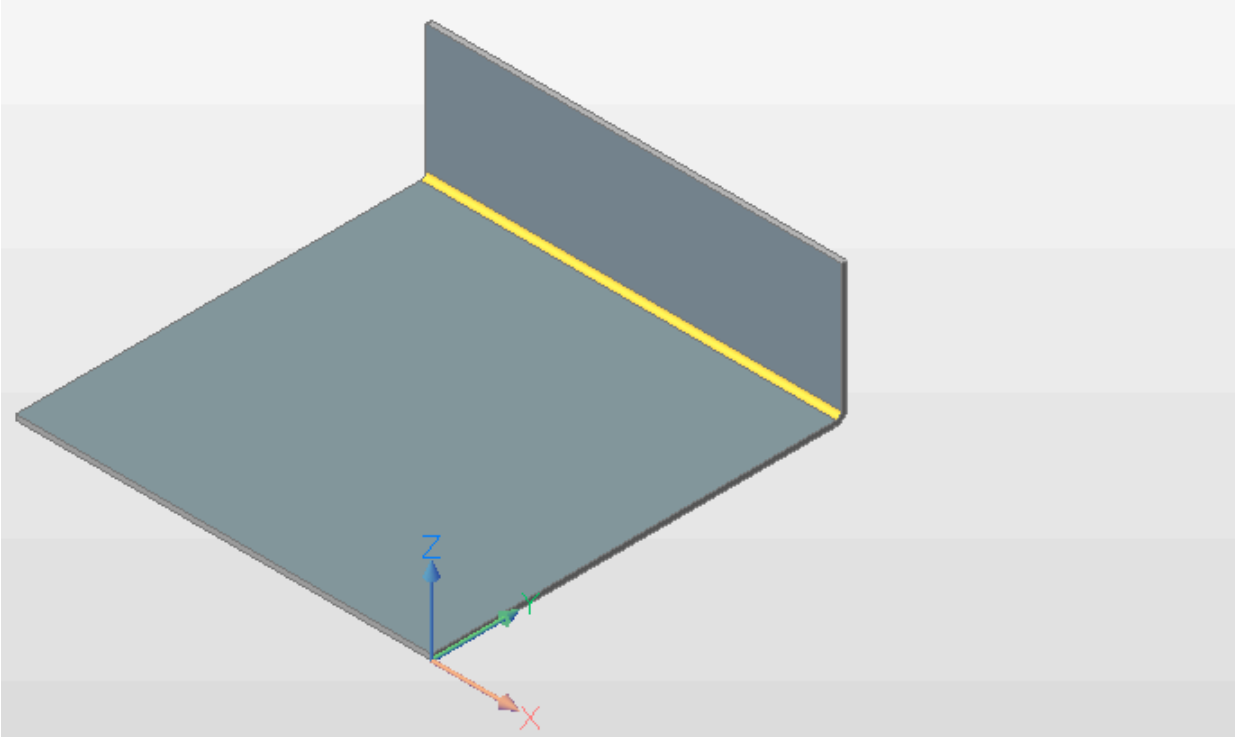
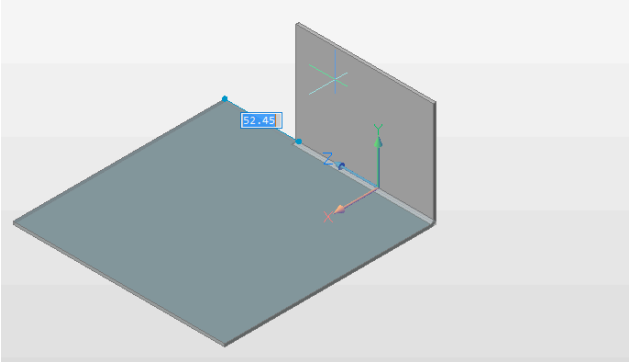


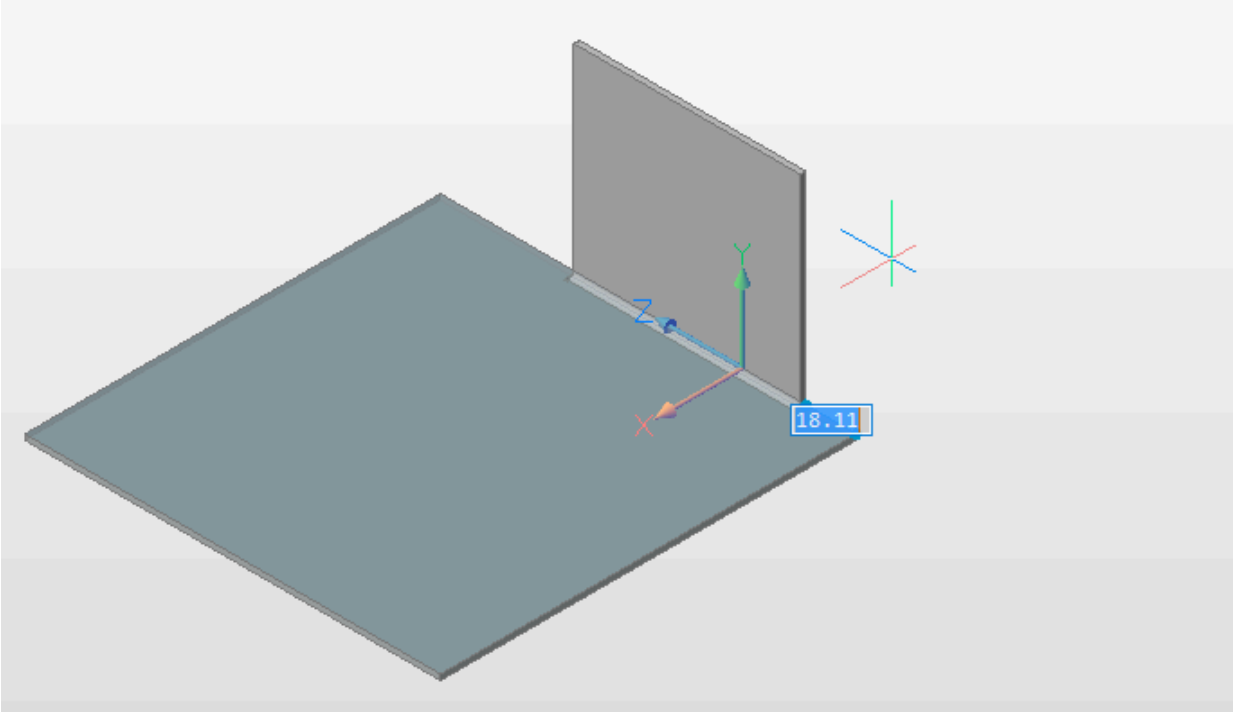
Figura 132: Creare Flangia da Spigolo

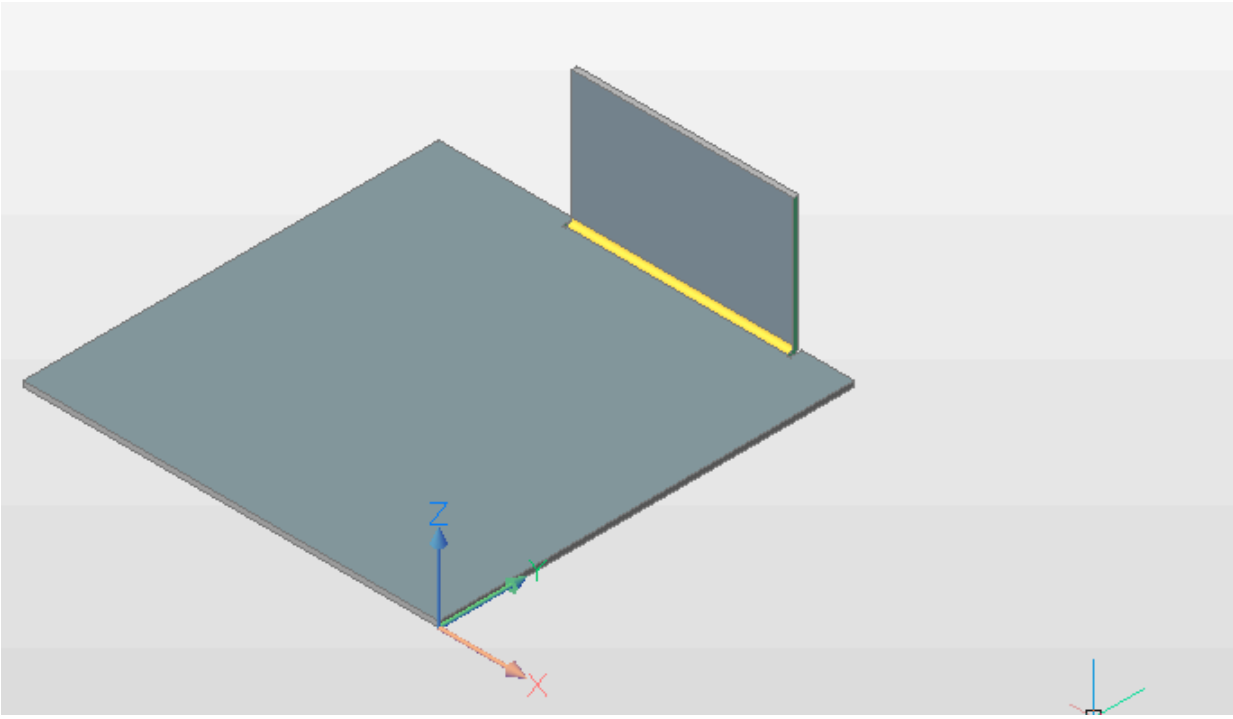




Per creare una flangia da spigolo in cui la larghezza è diversa dalla larghezza dello spigolo, selezionate l'opzione **Larghezza** del comando *SMFLANGEEDGE* e definite la distanza di offset da uno o entrambi i lati dello spigolo.



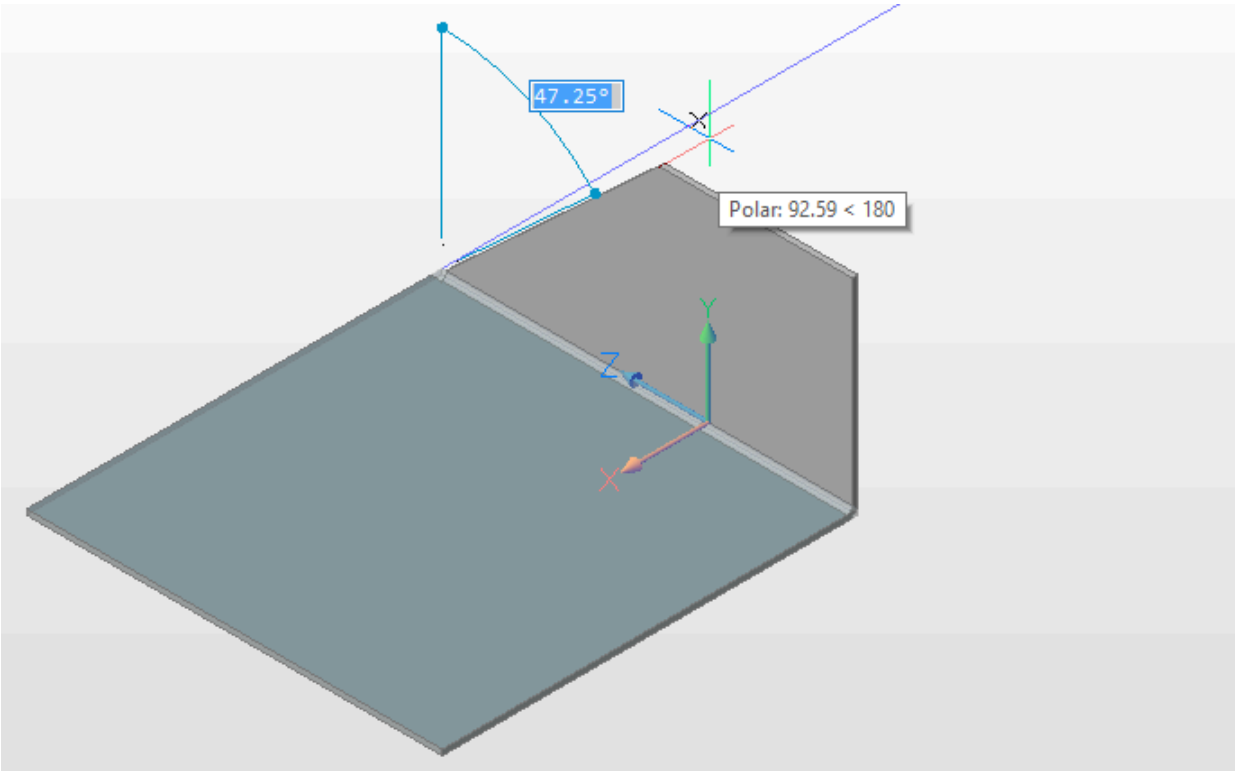


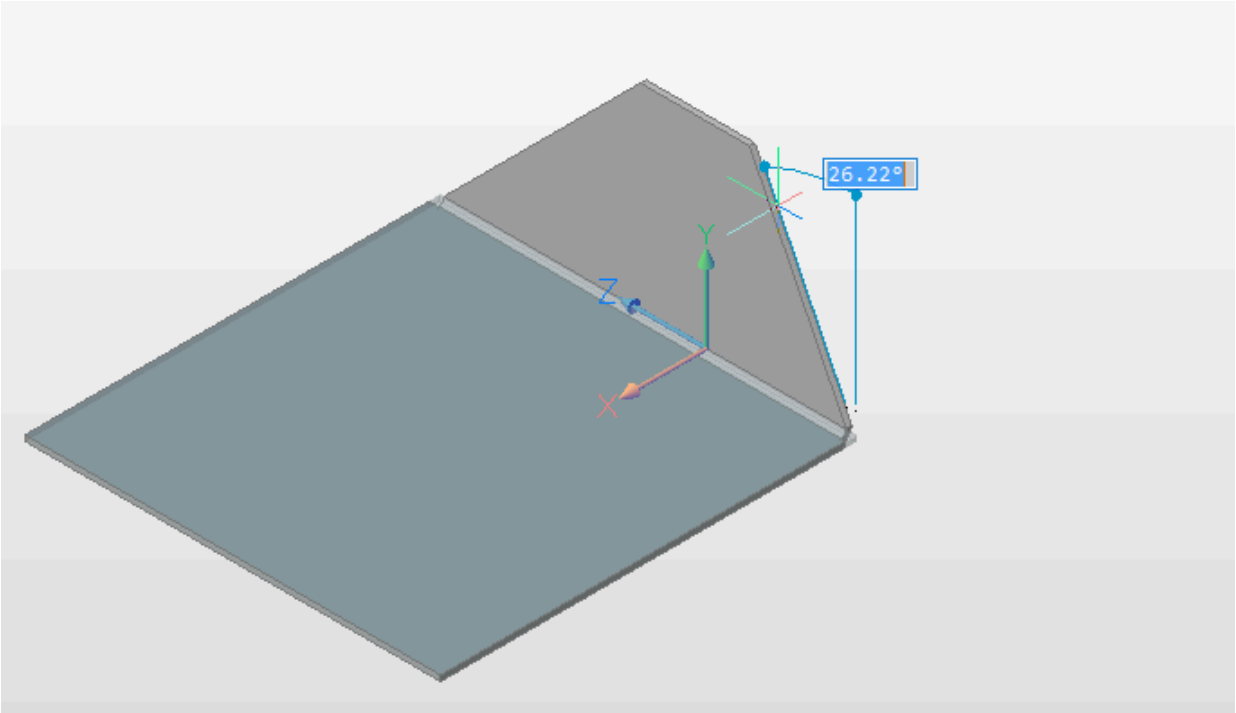


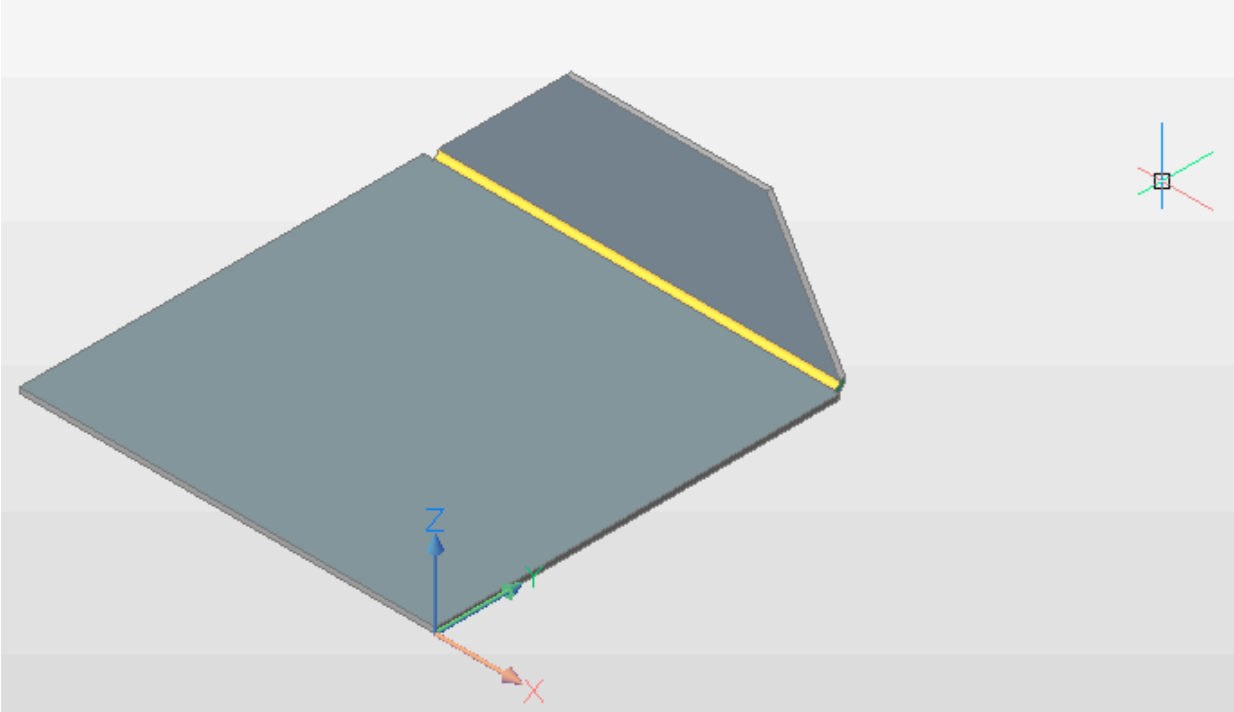
Nota: Quando si crea una flangia da spigolo di larghezza diversa rispetto allo spigolo, le gole di scarico per la piegatura vengono create automaticamente.

21.5.10.4 Creare Flangia da Spigolo con Angoli di Rastremazione

È possibile creare flange trapezoidali da spigoli con l'opzione **Angolo di rastremazione** del comando *SMFLANGEEDGE*:

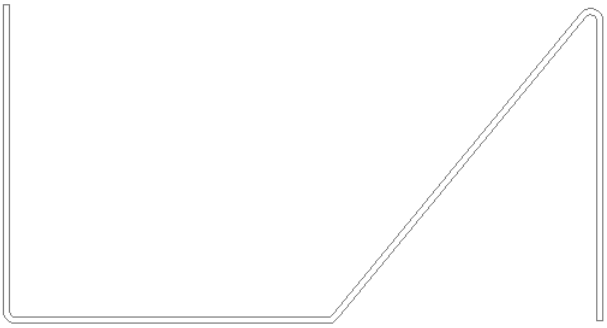






21.5.10.5 Creare Flange Multiple mediante l'Estrusione di una Polilinea

Esiste una classe semplice ma importante di parti in lamiera che sono viste come polilinee con spessore se le guardiamo "lateralmente":

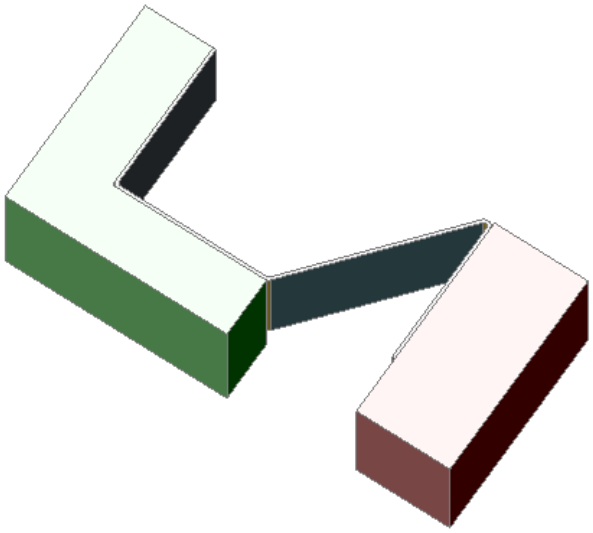


Per creare questa parte è possibile:

- utilizzare chiamate sequenziali del comando *SMFLANGEEDGE*, o
- disegnare una polilinea e utilizzare il comando *DMEXTRUDE*, seguito dai comandi *DMTHICKEN*, *SMCONVERT* e *SMBENDCREATE*.

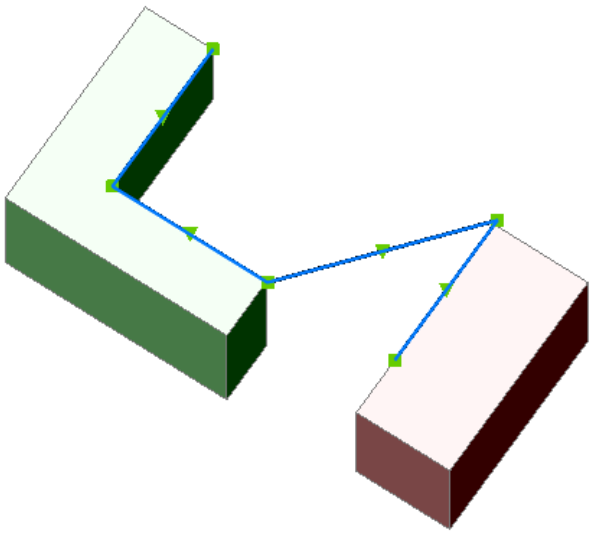
Tuttavia, consigliamo la seguente procedura:

BricsCAD consente di disegnare una parte in un assieme complesso con altre parti in modo che non interferisca con altre entità nell'assieme.



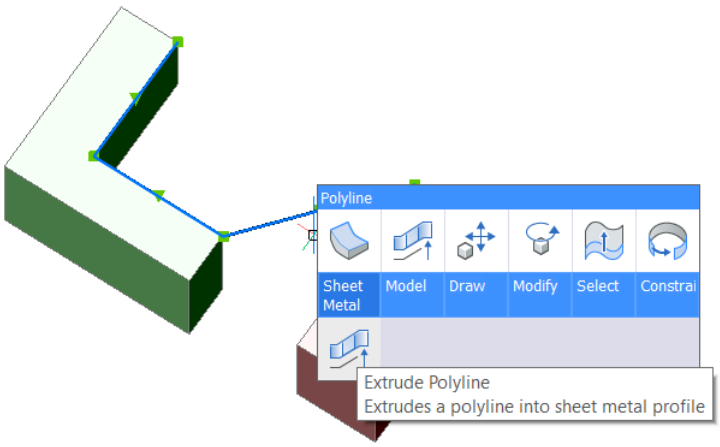
È possibile controllare individualmente la direzione di ispessimento di ogni segmento di linea con *SMEXTRUDE*.

1 Disegnare una polilinea che rappresenta uno schizzo della futura parte di lamiera:



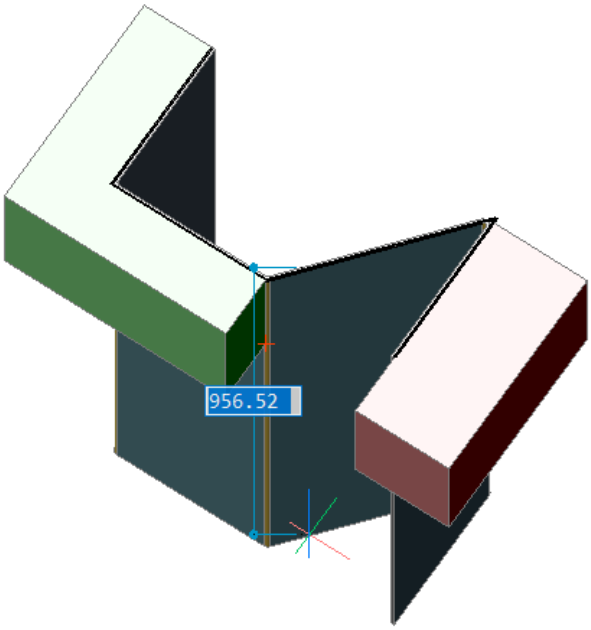
effettuare una delle seguenti operazioni:

- Cliccare sul pulsante **Estrudi Polilinea** ( ) nella barra degli strumenti **Lamiera**.
- Scegliere **Estrudi Polilinea** nel menu **Lamiera**.
- Scegliere **Estrudi Polilinea** nel gruppo **Lamiera** del menu del cursore



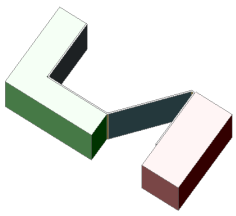
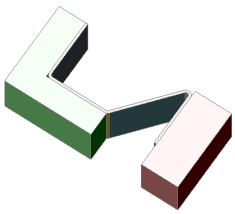
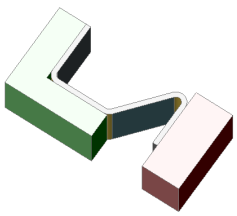
Quad:

2 Spostare il cursore del mouse per selezionare l'**altezza di estrusione** o inserire il valore e premere **Invio**



per accettarlo:

Il corpo viene creato e le estensioni vengono rispettate. La parte in lamiera non supera le dimensioni della polilinea originale. Il comportamento è lo stesso, anche con spessori diversi:

Spessore 1 mm	Spessore 2 mm	Spessore 4 mm
		

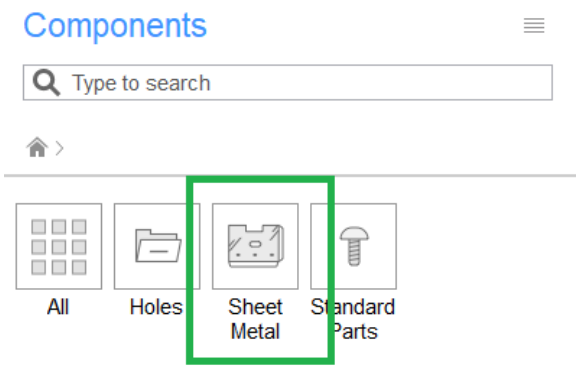
Nota: La politica di conservazione delle dimensioni esterne funziona anche per le parti in lamiera create da altri flussi di lavoro: costruzione sequenziale di flange con *SMFLANGEEDGE*, piegature di parti con *SMFLANGEBEND* e riconoscimento della progettazione della lamiera con *SMCONVERT*. Se si desidera modificare la direzione di estrusione di una flangia, applicare il comando *SMFLIP*.

21.5.10.6 Estraudere un Contorno per Creare una Flangia

Il comando *SMFLANGECONTOUR* consente di estrudere un determinato contorno in una nuova flangia e di fissarla al modello. Può essere considerato come una sequenza dei comandi *SMFLANGEBASE*, *SMFLANGECONNECT* e *SMBENDCREATE*. In realtà, sceglie in modo intelligente le facce di spessore sulle flange selezionate e le collega con le facce di spessore su una nuova flangia.

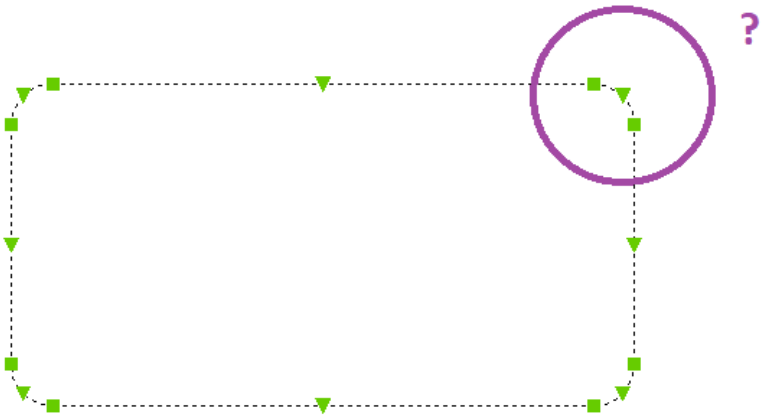
Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Crea Flangia da Contorno** ( ) sulla barra degli strumenti **Lamiera**.
- Scegliere **Crea Flangia da Contorno** nel menu **Lamiera**.
- Scegliere **Crea Flange da Contorno** nel gruppo **Lamiera** del menu del cursore

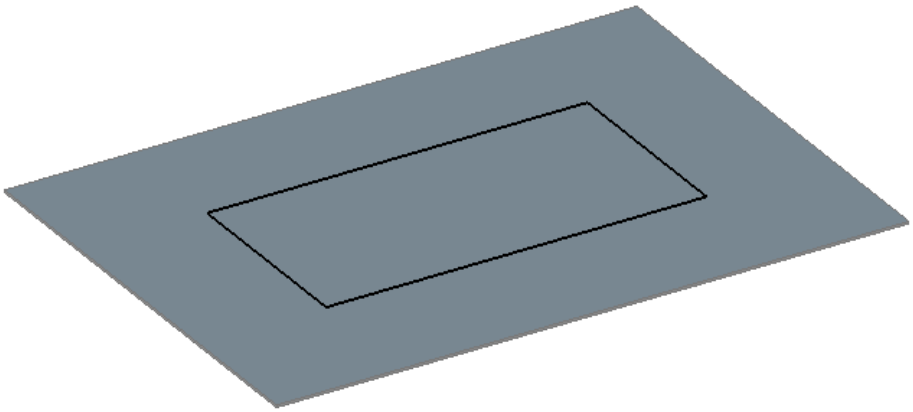


Quad:

Una volta selezionate tutte le flange richieste, fare clic sul contorno:

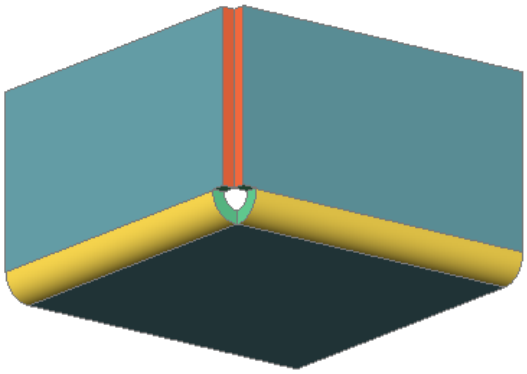


Il comando crea una nuova flangia e la fonde con le flange selezionate sulla parte:



21.5.10.7 Piegature, Scarichi d'Angolo e Giunzioni

Quando si tira uno spigolo di una flangia con il comando *SMFLANGEEDGE*, viene automaticamente creata una caratteristica di piegatura tra lo spigolo della flangia e la flangia di base. Una caratteristica scarico d'angolo viene creata automaticamente in un angolo in cui si incontrano 3 flange. Questa operazione crea anche una giunzione tra due flange, le quali non sono collegate con una piegatura.



Esercitazione: Progettare da Zero la Lamiera in BricsCAD

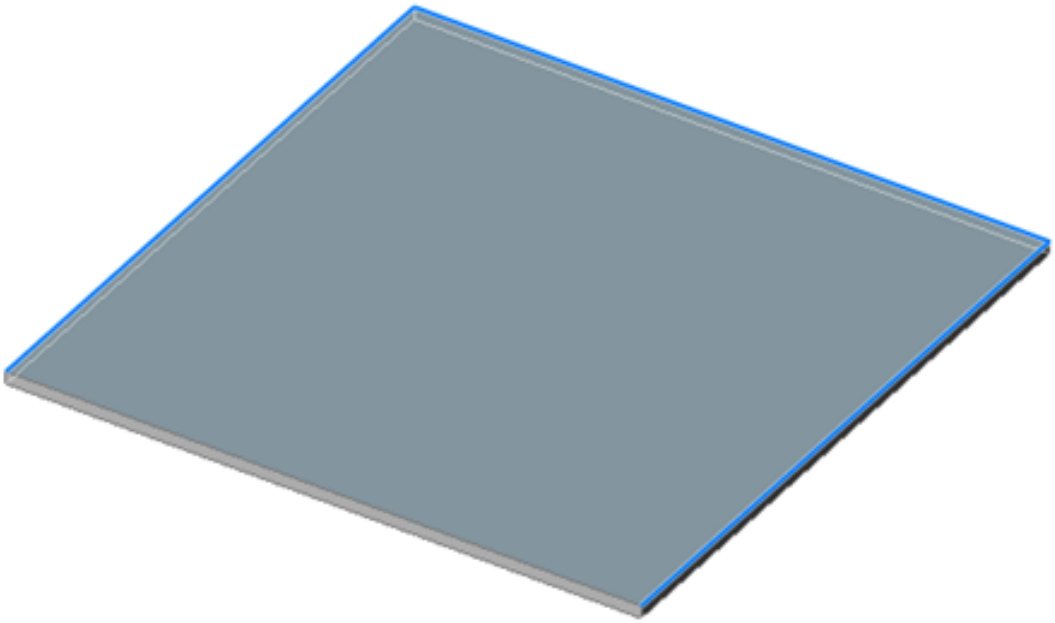
21.5.10.8 Creare Flange Multiple

È possibile creare più flange contemporaneamente.

Eseguire una delle seguenti operazioni:

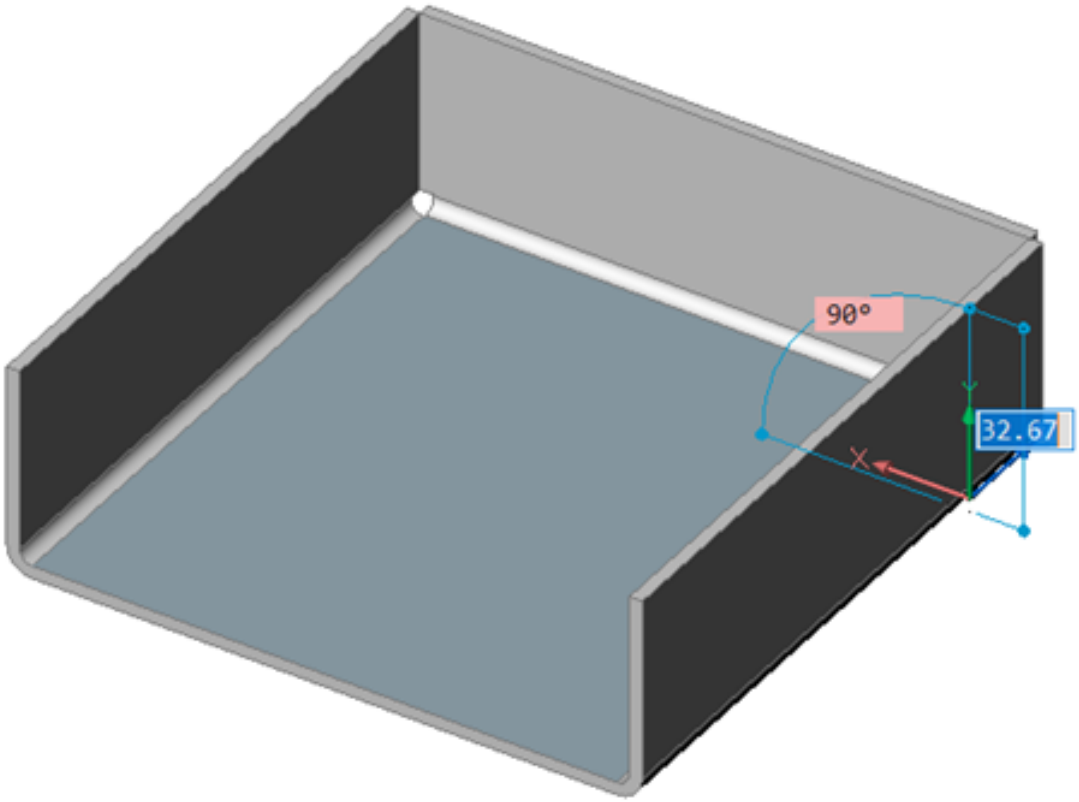
- Selezionare più spigoli, quindi scegliere **Crea Flangia da Spigolo** ( ) nel gruppo di comandi **Lamiera** del menu del cursore **Quad**.

- Avviare il comando *SMFLANGEEDGE* e selezionare più

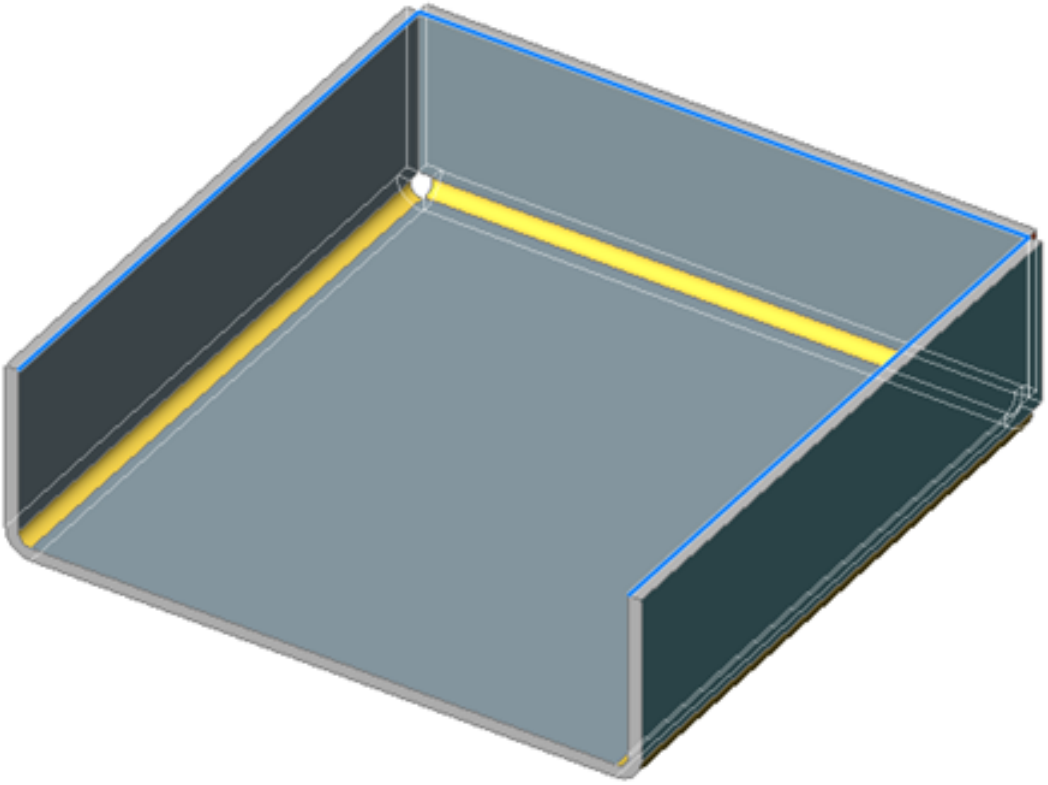


  spigoli.
- Crea dinamicamente le flange.

  Definire l'angolo e l'altezza della flangia utilizzando i campi delle dimensioni dinamiche. Premere il tasto **TAB** per passare dal campo **Angolo** a quello della **Distanza**.


90°
32.67

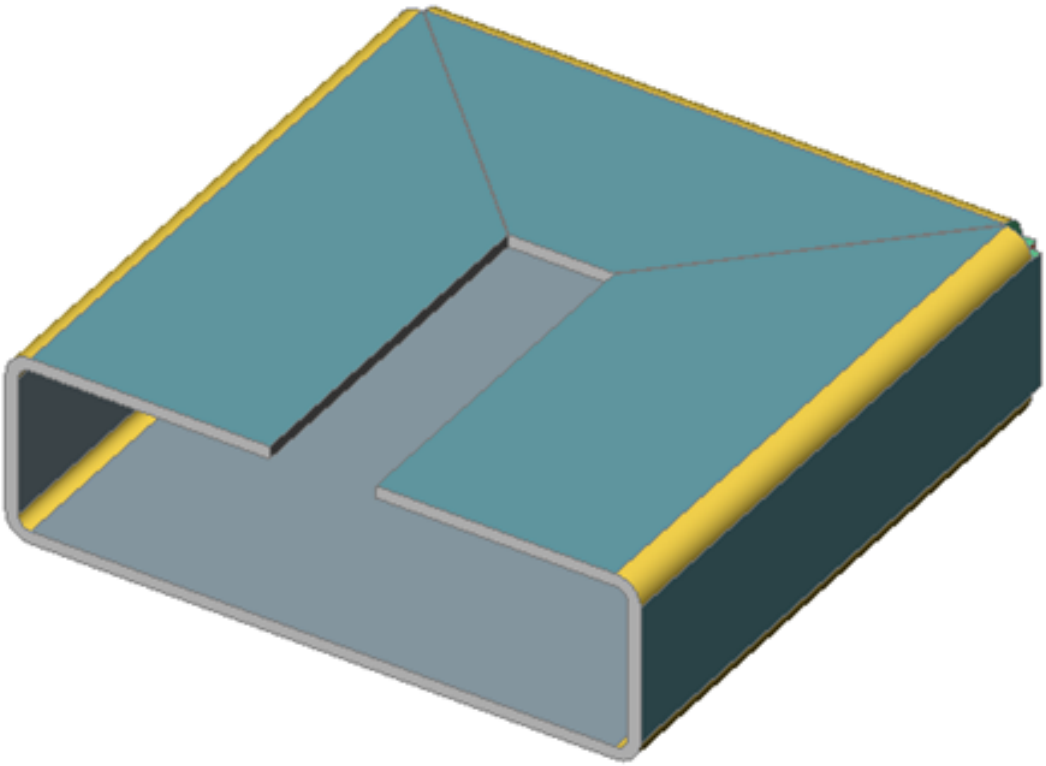
- Selezionare gli spigoli interni delle flange appena



create:

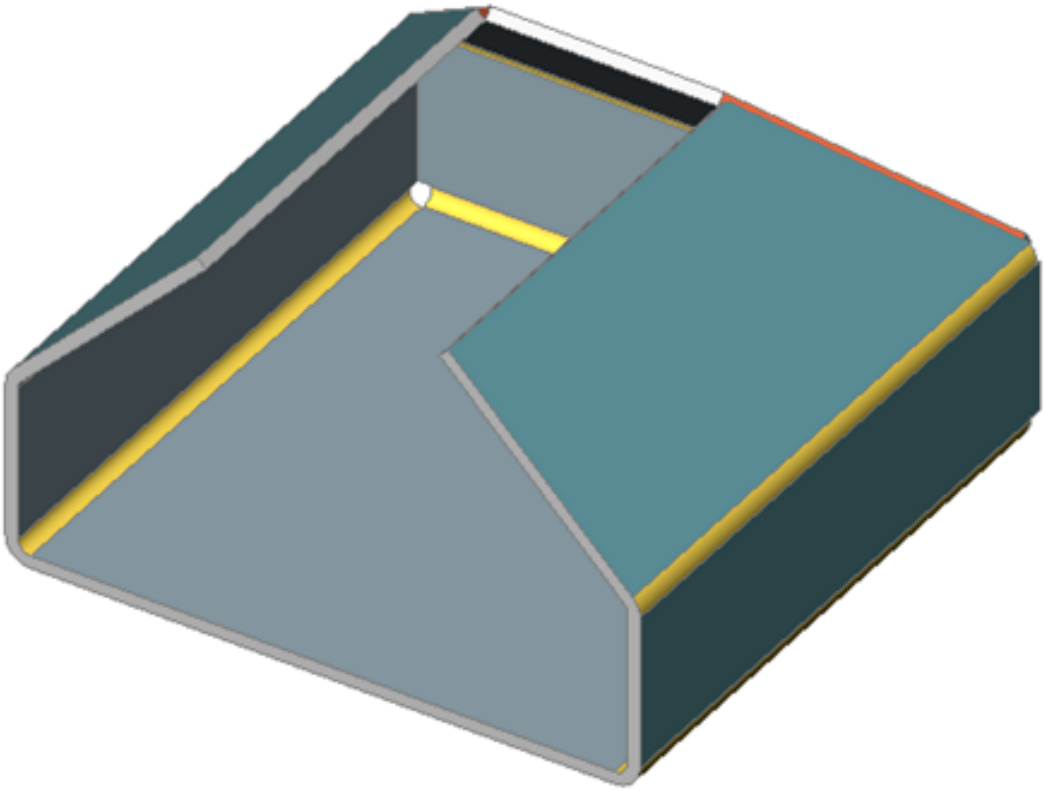
- Creare le flange.

Il comando calcola automaticamente l'intersezione delle flange e crea una giunzione o caratteristica taglio ad angolo. Sui 90 gradi vengono creati i tagli ad

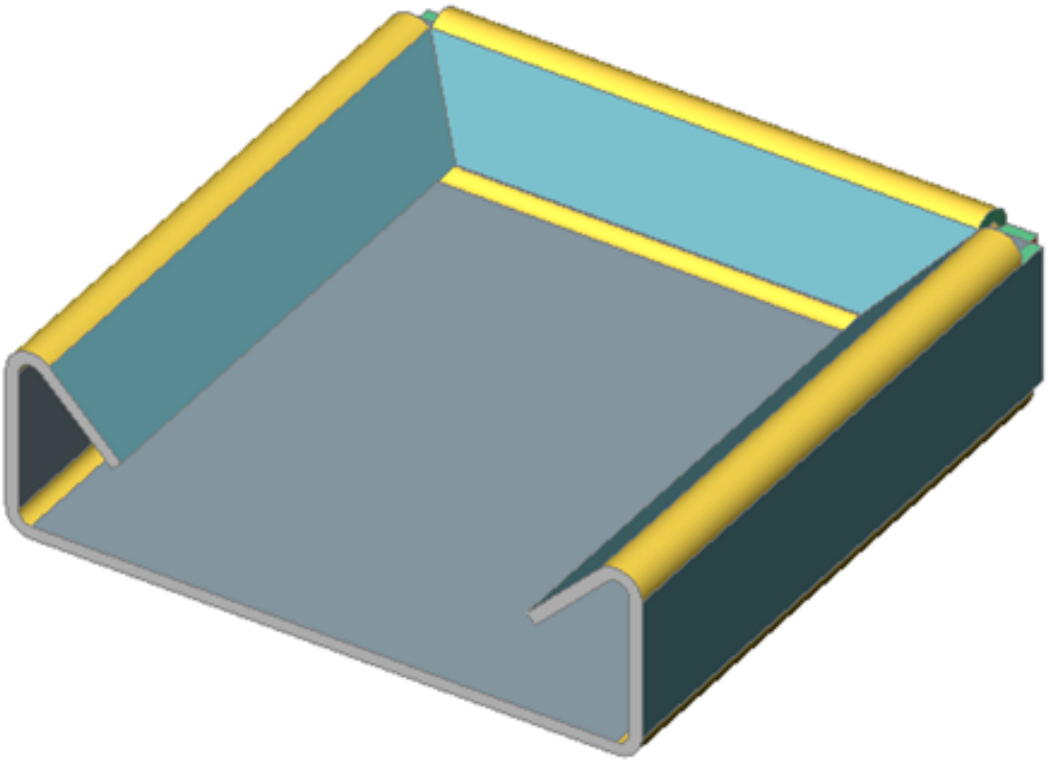


angolo:

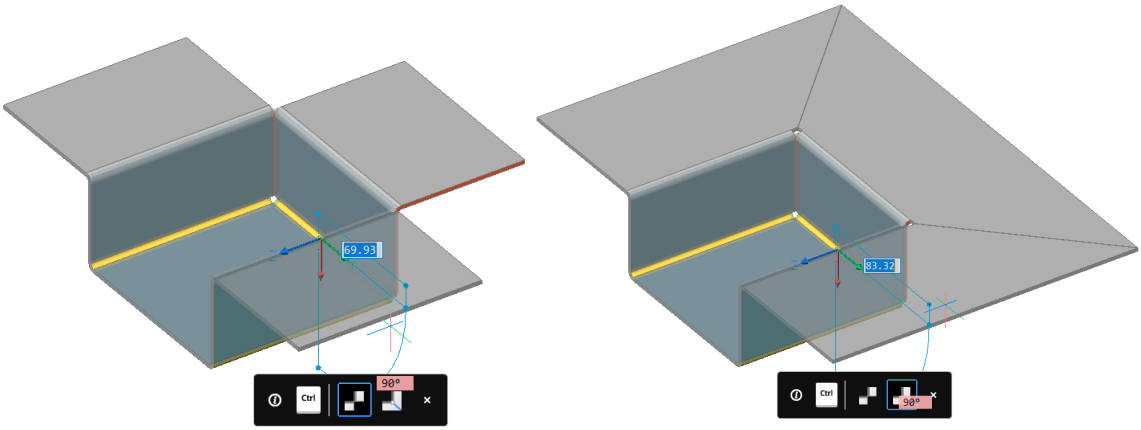
Per altri angoli vengono create delle giunzioni:



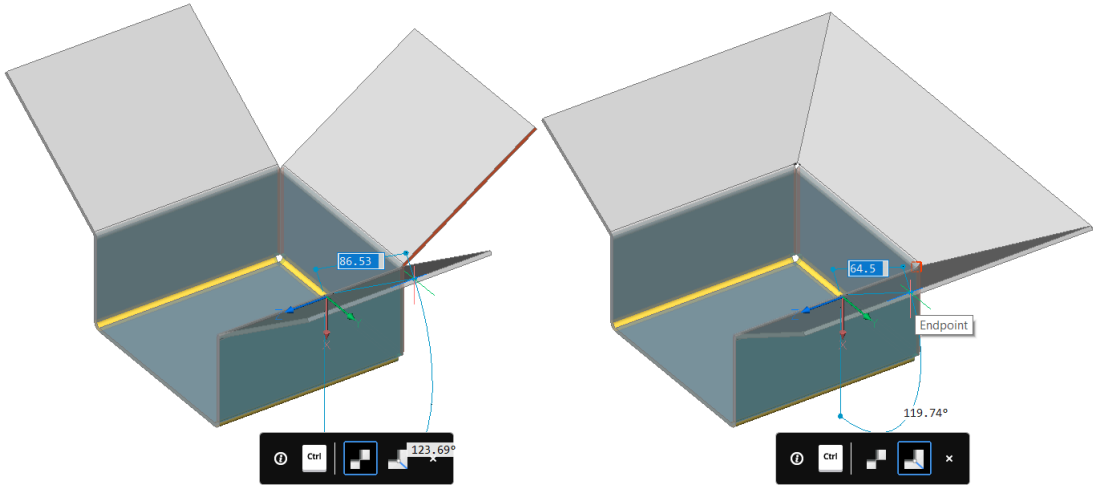
In tutti i casi si evitano conflitti di materiale:



Se le flange vengono create esternamente, utilizzare l'opzione **Commuta** (oltre al **widget Ctrl**) per controllare il tipo di connessione per i tagli ad angolo:



e le giunzioni:



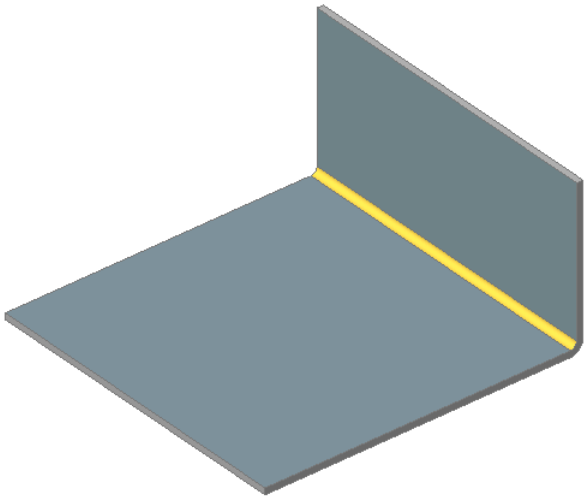
21.5.11 Modificare le flange

BricsCAD offre diversi strumenti per modificare le flange nelle parti in lamiera. Questi strumenti possono essere applicati in modo coerente a qualsiasi parte, indipendentemente dalla sua cronologia di progettazione. In questo articolo vengono utilizzati gli strumenti di Modellazione Solida e gli strumenti specifici della lamiera. Quasi tutti gli strumenti di modellazione solida si applicano alle parti in lamiera, pur preservando l'intelligenza della progettazione fornita dalle Caratteristiche della Lamiera.

21.5.11.1 Cambiare la dimensione e la forma della flangia

Su un determinato semplice modello applicheremo diverse operazioni di Modellazione Solida per modificare la forma delle sue flange.

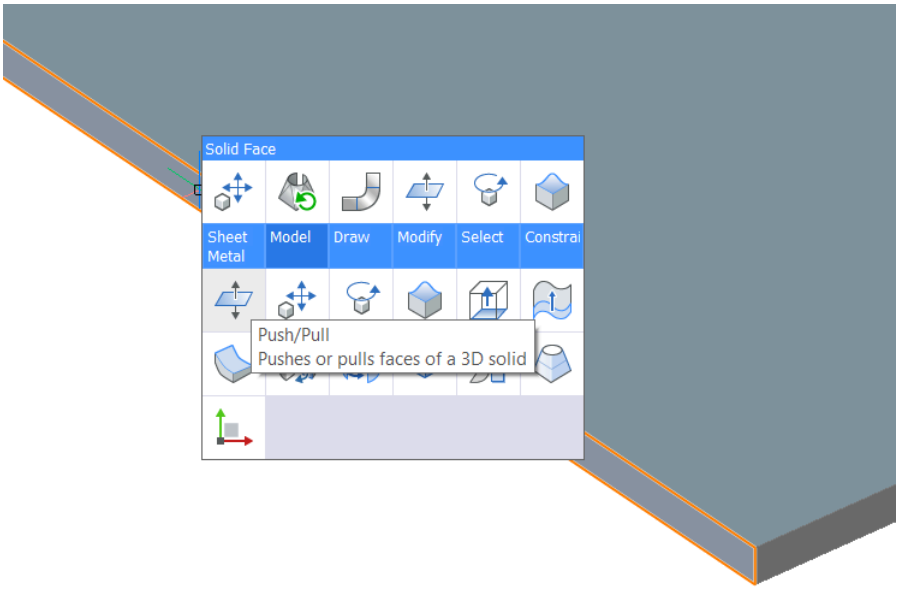
Nota: Lavoriamo solo su facce dello spessore del modello



21.5.11.2 Stirare le flange

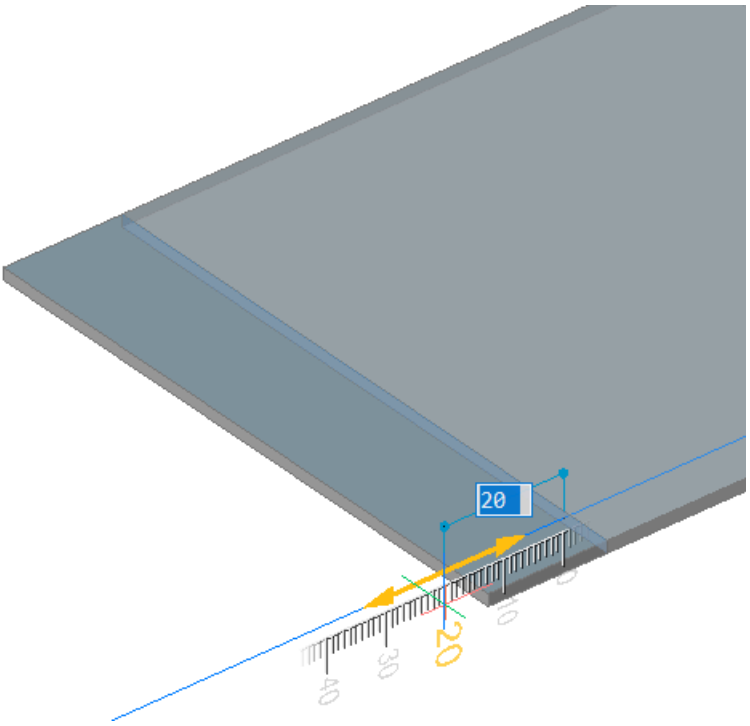
1 Passare il puntatore del mouse sulla faccia dello spessore, quindi

selezionare **PremiTira** nel gruppo di comandi **Modello** nel menu



Quad.

2 Trascinare la faccia, quindi Cliccare il pulsante destro del

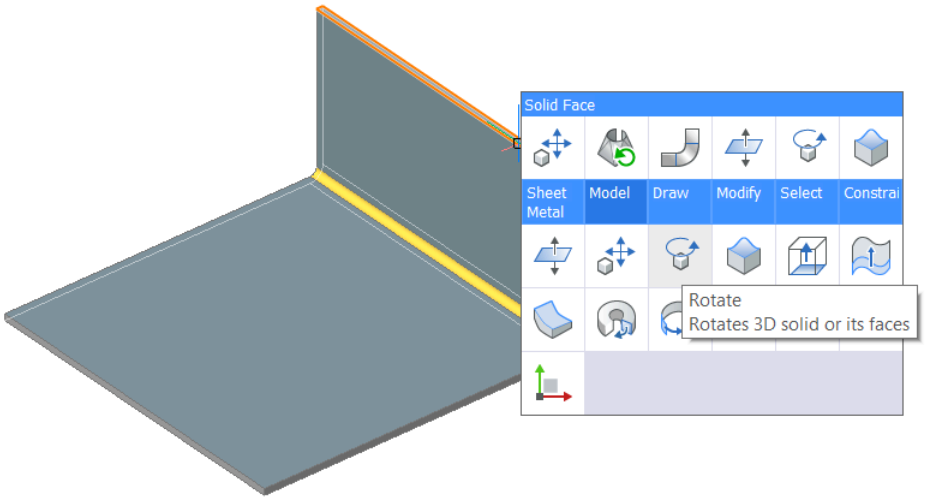


mouse.

Le modifiche verranno applicate alla parte, la flangia viene allungata sulla distanza specificata.

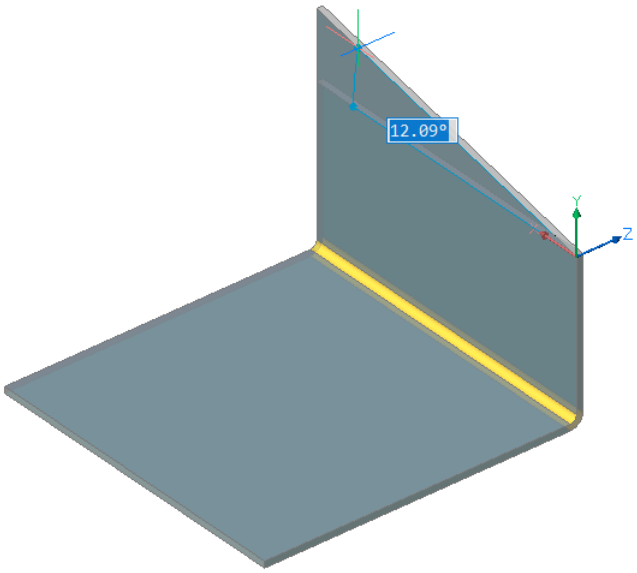
21.5.11.3 Ruotare una faccia dello spessore

1 Passare il mouse sulla faccia e scegliere **Ruota** nel gruppo di comandi **Modello** nel menu



Quad.

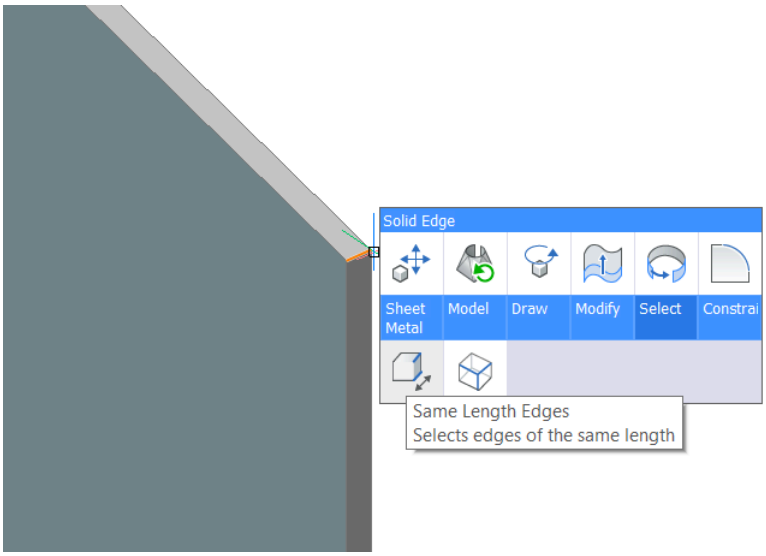
2 Specificare l'asse di rotazione, il punto base e l'angolo (per ulteriori dettagli, vedere DmRotate nella



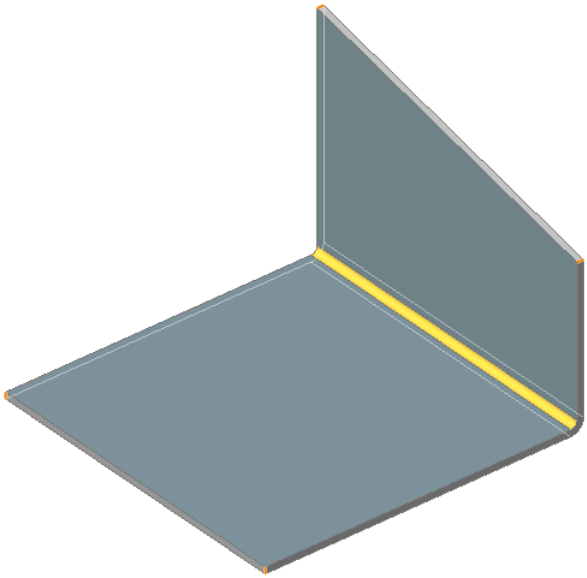
Guida di Riferimento ai Comandi):

3 Raccordare gli spigoli vivi.

Passare il mouse su uno spigolo dello spessore e scegliere **Spigoli con Stessa Lunghezza** ( ) nel gruppo di comandi **Seleziona** nel menu **Quad** .



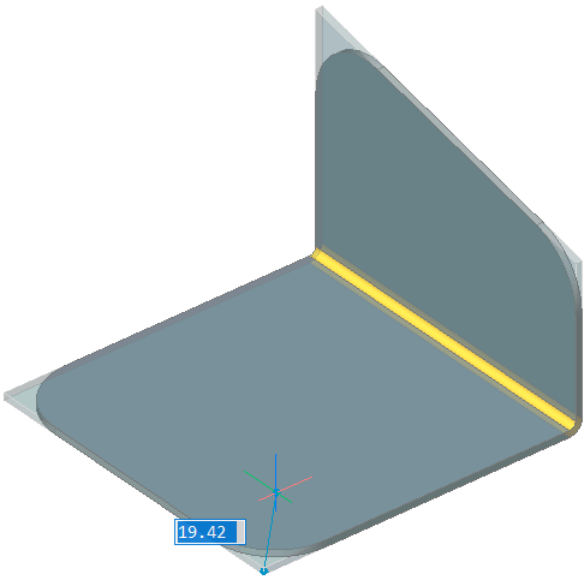
4 Premere **Invio**. Gli spigoli con la stessa lunghezza sono stati

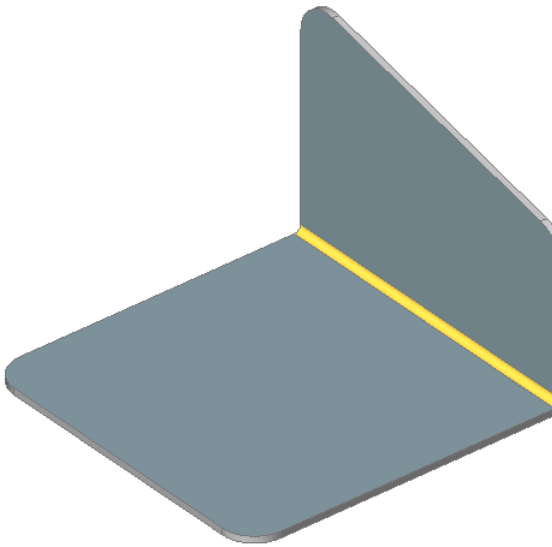


selezionati.

5 Scegliere **Raccordo** nel gruppo di comandi **Modello** nel menu **Quad**.

6 Specificare il raggio di raccordo

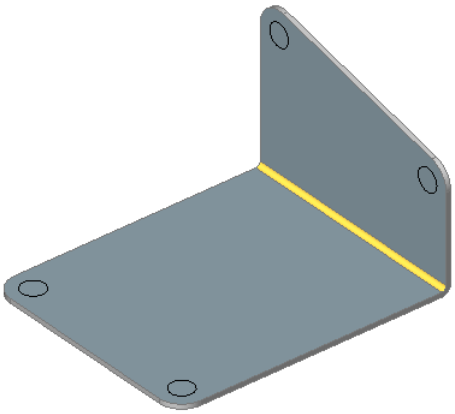




desiderato:

21.5.11.4 Realizzare un foro in una flangia

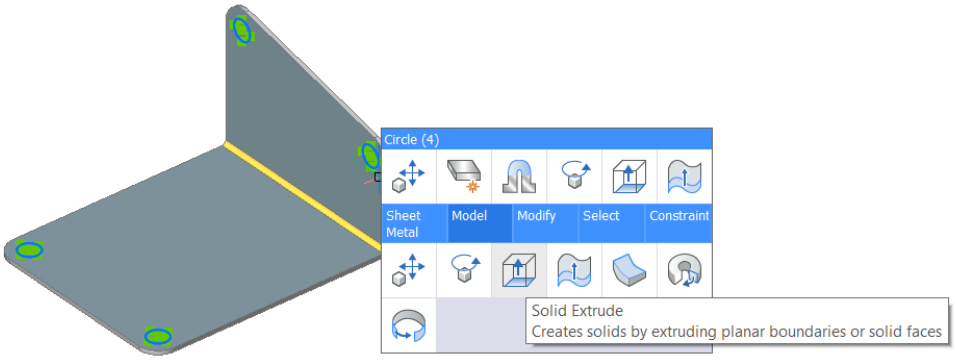
1 Creare 4 cerchi come indicato nell'immagine qui sotto. Usare lo snap dell'entità Snap a Centro per



definire i punti centrali.

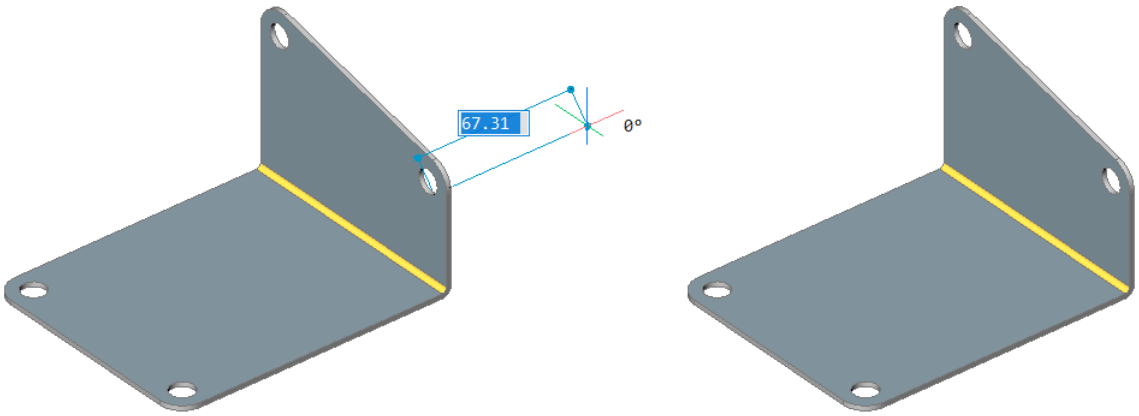
2 Selezionare i cerchi.

3 Scegliere **Solido di Estrusione** nel gruppo di comandi **Modello** nel menu



Quad.

4 Spostare il cursore per creare i

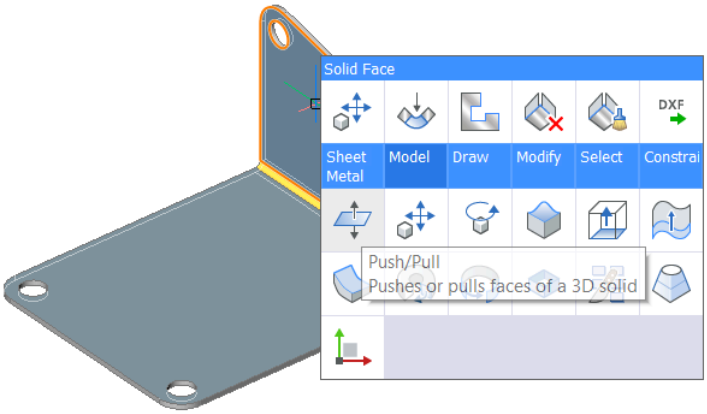


fori.

21.5.11.5 Spostare una flangia

1 Eseguire una delle seguenti operazioni:

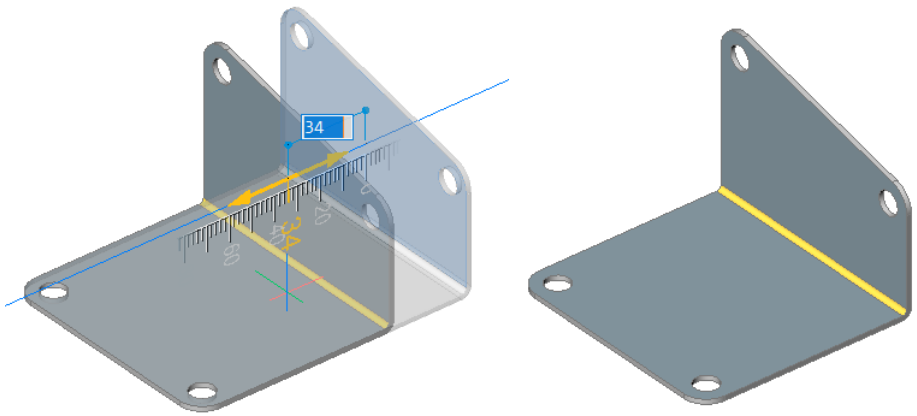
- Cliccare il pulsante **Premi/Tira** ( ) sulla barra degli strumenti **Modellazione Diretta**
- Scegliere **Modellazione Diretta > Premi/Tira** nel menu **Modellazione**.
- Scegliere **Premi/Tira** ( ) nel pannello **Edita** della scheda **Lamiera** nella barra multifunzione.
- Eseguire il comando *DMPUSHPULL* per la faccia specificata.
- Passare il mouse sulla faccia che si desiddera spostare e scegliere **Premi/Tira** nel gruppo di comandi **Modello** nel menu



Quad.

2 Selezionare la nuova posizione della flangia.

Nota: A causa della presenza di **Caratteristiche della Lamiera**, lo spessore della parte viene mantenuto.



21.5.11.6 Ruotare una flangia

Anche se è possibile ruotare una flangia utilizzando il comando *DMROTATE* ( ) si raccomanda di utilizzare invece il comando *SMFLANGEROTATE* ( ), poiché questo comando seleziona automaticamente l'asse di rotazione per rispettare l'intento della progettazione della parte in lamiera.

1 Avviare il comando *SMFLANGEROTATE* ( ).

   Viene richiesto: Selezionare una faccia della flangia da ruotare [opzioni di selezione (?)]:

   La faccia sotto il cursore di evidenzia.

2 Cliccare per selezionare la faccia della flangia.

   La flangia ruota in modo dinamico.

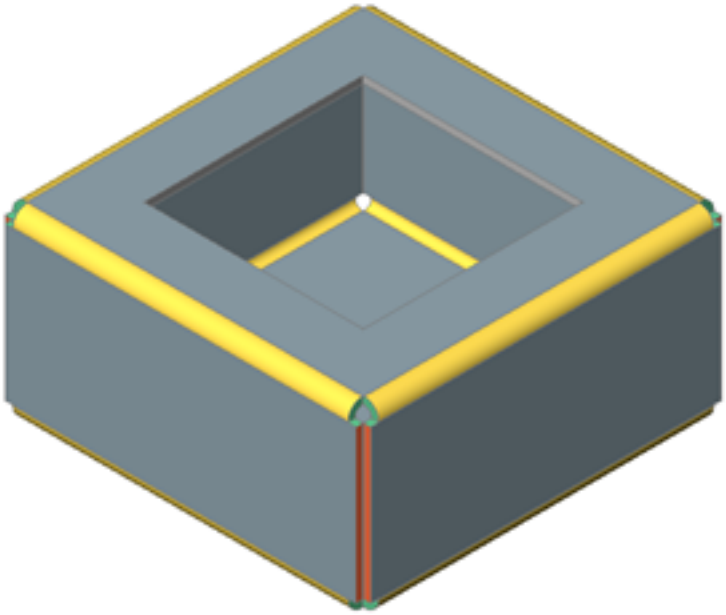
   Una quota dinamica visualizza l'angolo corrente rispetto alla flangia di base.

3 Eseguire una delle seguenti operazioni:
   - Selezionare un punto.
   - Immettere un valore nel campo quota dinamica.
   - Premere il tasto **TAB**, quindi specificare l'angolo di rotazione assoluto nel campo della quota dinamica.

Nota: Quando il comando *DMROTATE* viene usato per ruotare una flangia, la quale è connessa ad altre flange attraverso le caratteristiche giunzione, queste caratteristiche vengono automaticamente dissolte prima della rotazione.

21.5.11.7 Dividere una flangia

In alcuni casi è meglio dividere una flangia per ridurre al minimo il consumo di materiale, ad esempio quando una flangia ha un foro molto grande come nell'immagine qui sotto:



1 Avviare il comando *SMSPLIT* ( ).

Viene richiesto: Selezionare una flangia o piegatura con loft:

2 Selezionare una faccia della flangia.

Viene richiesto:

Selezionare linee o spigoli per dividere la flangia o [divisione INtelligente/disegna una Nuova linea/opzioni di selezione (?)] <disegna una Nuova linea>:

3 Se non esiste ancora nessuna linea sulla faccia della flangia, cliccare il pulsante destro del mouse.

Viene richiesto: Punto iniziale della linea:

4 Specificare il punto iniziale della linea.

Viene richiesto: Punto finale della linea:

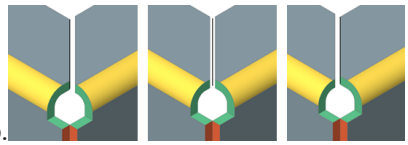
5 Specificare il punto finale della linea.

Viene richiesto: Esegui divisione [Centro/Sinistra/Destra/Accetta] <Accetta>:

6 Per regolare la posizione del taglio rispetto alla linea, effettuare una delle seguenti operazioni:

- Premere il tasto **CTRL** per attivare o disattivare le opzioni **Centro**/**Sinistra**/**Destra**.

  Se **HKA** è attivo, l'opzione attualmente selezionata è indicata nel widget Suggerimenti.
- Selezionare **Centro**, **Sinistra** o **Destra** nel menu contestuale.
- Immettere un'opzione nella barra dei comandi, quindi premere

  Invio. 

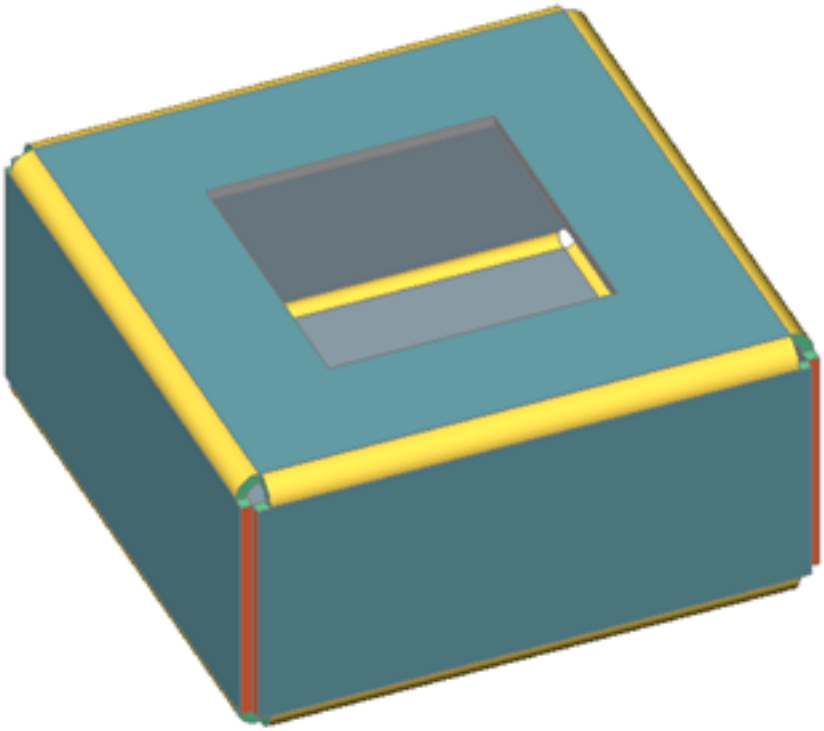
  La flangia viene tagliata dalla linea: sinistra, centro o destra.

Nota: Un'operazione di divisione crea una caratteristica di taglio ad angolo. È possibile ripristinare la flangia iniziale rimuovendo la caratteristica taglio ad angolo dall'elenco delle caratteristiche nel **Navigatore Meccanico**.

21.5.11.8 Strumenti intelligenti per dividere una flangia

Lo scenario seguente illustra le altre funzionalità del comando *SMSPLIT*:

- Divisione intelligente selezionando un singolo vertice nell'angolo.
- Divisione della piegatura e conversione della sua parte adiacente all'angolo in una



giunzione.

1 Eseguire una delle seguenti operazioni:
   - Cliccare sullo strumento **Divisione Intelligente** ( ) sulla barra multifunzione **Lamiera**.
   - Scegliere **Dividi Intelligente** nel menu **Lamiera**.
   - Digitare *SMSPLIT* nella barra dei comandi.

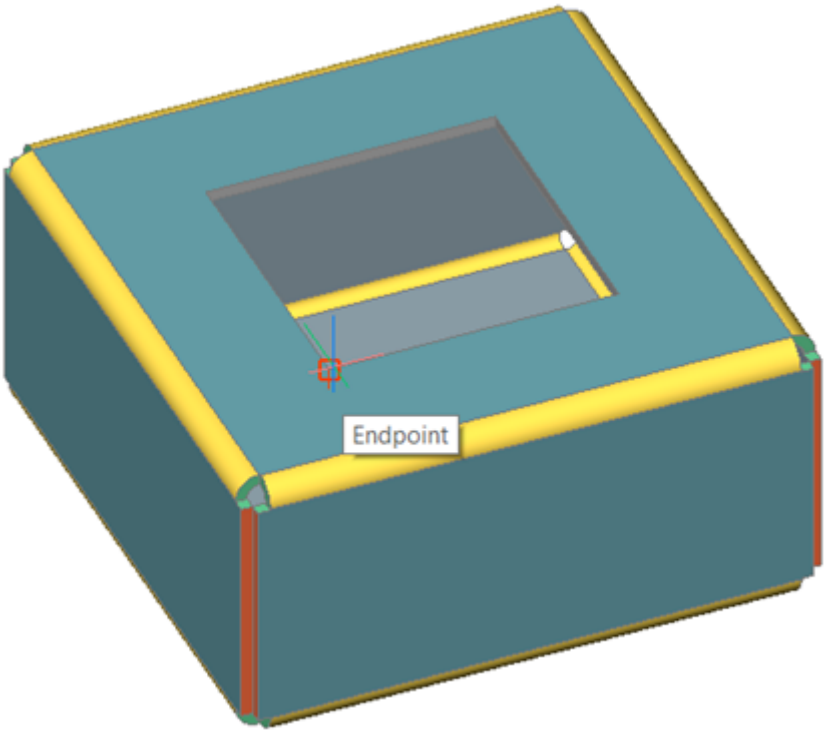
     Viene richiesto: Selezionare flangia o faccia piegatura con loft:

2 Selezionare una faccia della flangia.

   Viene richiesto:

   Selezionare linee o spigoli per dividere la flangia o [divisione INtelligente/disegna una Nuova linea/opzioni di selezione (?)] <disegna una Nuova linea>:

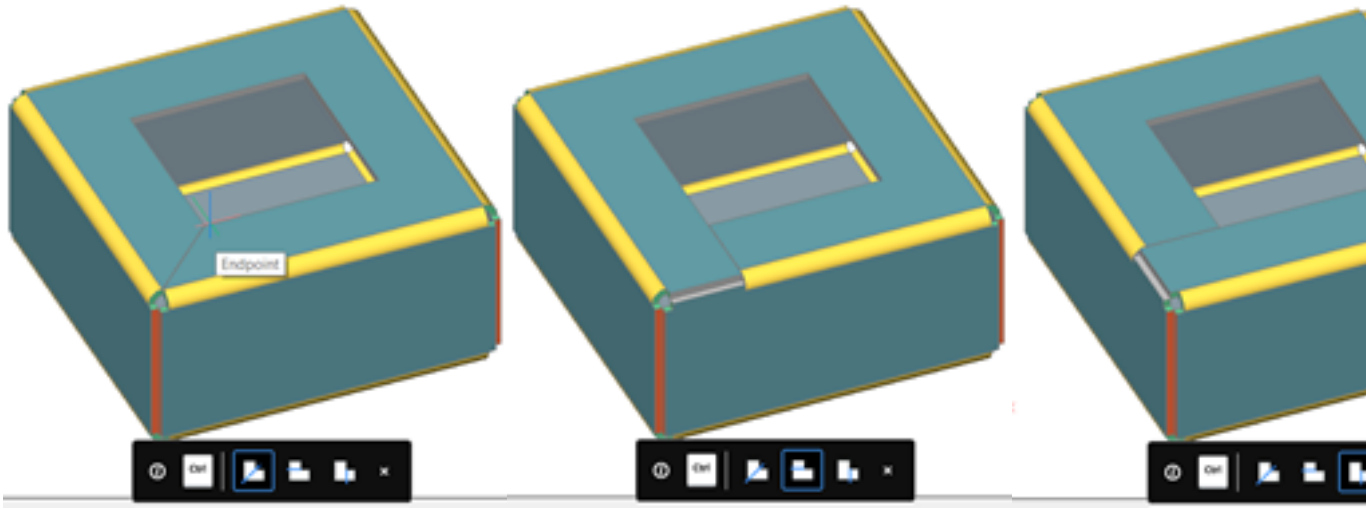
3 Selezionare l'opzione **Divisione Intelligente**, quindi selezionare un vertice



d'angolo:

Viene richiesto: Esegui divisione [Centro/Sinistra/Destra/Accetta] <Accetta>:

Le opzioni sono:



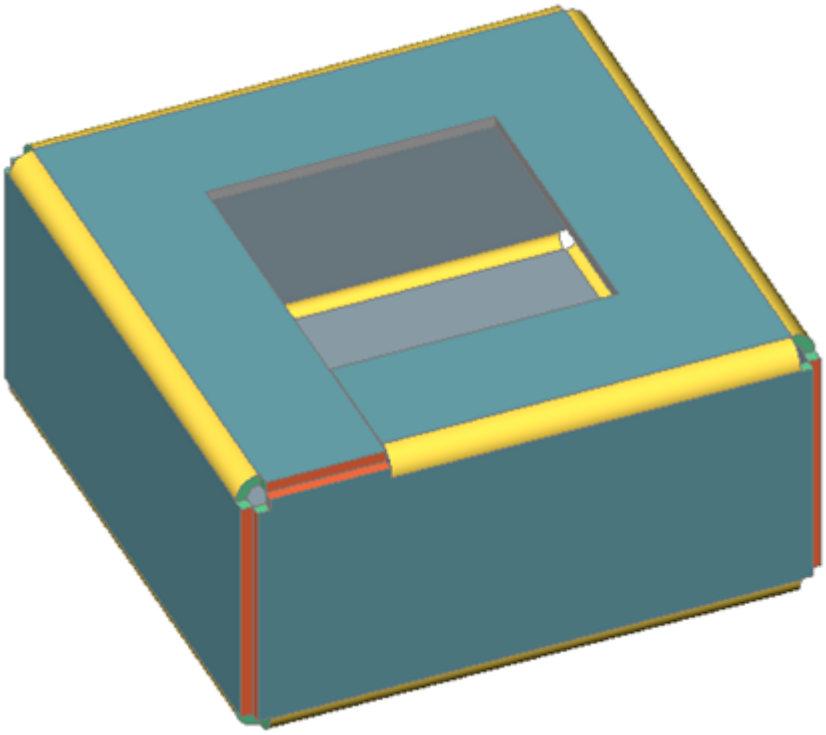
Nota: La conversione di una parte della piegatura in un giunzione non è correlata alla modalità **Divisione Intelligente**.

4 Per commutare le opzioni, effettuare una delle seguenti operazioni:

- Premere ripetutamente il tasto **CTRL**.

  Se **HKA** è attivato, l'opzione attualmente selezionata è indicata nel widget Suggerimenti.
- Selezionare **Centro**, **Sinistra** o **Destra** nel menu contestuale.
- Immettere un'opzione nella barra dei comandi, quindi premere **Invio**.

La flangia viene tagliata da una linea, la piegatura viene parzialmente convertita in giunzione:



Nel caso dell'opzione **Centra** se le **Caratteristiche Intelligenti** sono attivate, gli scarichi d'angolo adiacenti alla divisione verranno convertiti in gole di scarico.

21.5.11.9 Piegare una flangia

Il comando *SMFLANGEBEND* piega una o più flange utilizzando una linea. È possibile disegnare prima la linea, quindi avviare il comando o utilizzare l'opzione **Nuova linea** del comando.

Piegare più flange contemporaneamente

1 Avviare il comando *SMFLANGEBEND*.

   Viene richiesto: Selezionare una faccia della flangia [opzioni di selezione (?)]:

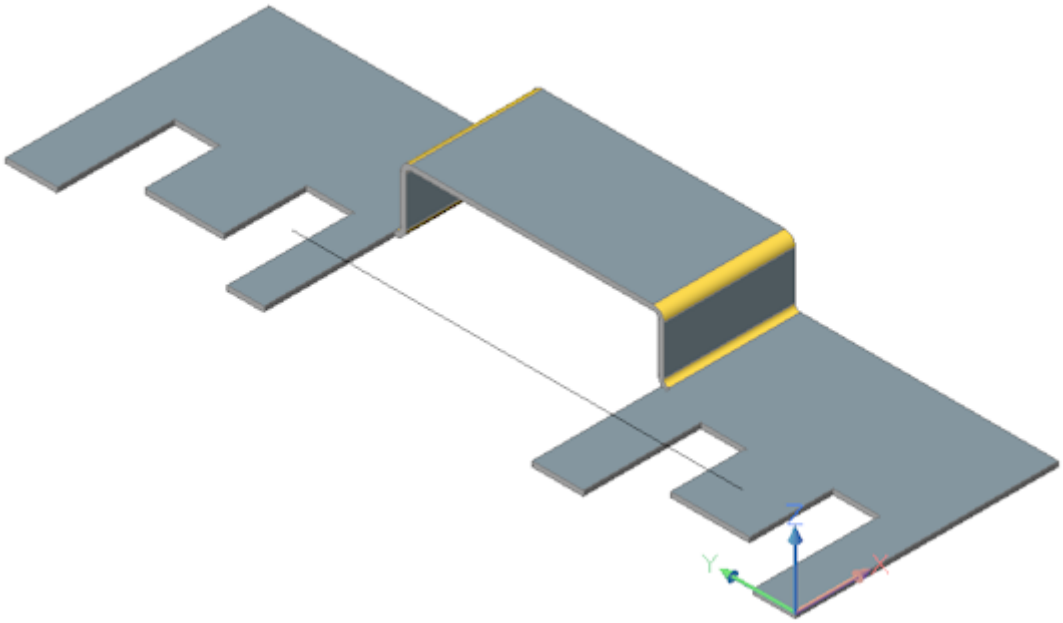
2 Seleziona la flangia che si desideri piegare.

   Viene richiesto: Selezionare linea o spigolo per piegare il solido o disegnare una [Nuova linea/opzioni di selezione (?)] <Nuova linea>:

3 Eseguire una delle seguenti operazioni:
   - Selezionare una linea che si sovrappone alle flange da piegare.
   - Scegliere l'opzione **Nuova linea**.

     Viene richiesto di tracciare una linea sopra la flangia o le flange da piegare.

4 Viene richiesto: Specificare un punto sul lato che verrà



spostato:

Scegliere un punto sul lato che desideri spostare.

La parte selezionata viene piegata dinamicamente.

Viene richiesto: Selezionare la posizione della flangia [Angolo/Raggio/Commuta lato/COmmuta estensioni piegatura/ACcetta] <Accetta>:

5 (opzionale) Digitare un valore per definire l'angolo di piegatura, utilizzare il raggio di piegatura predefinito.

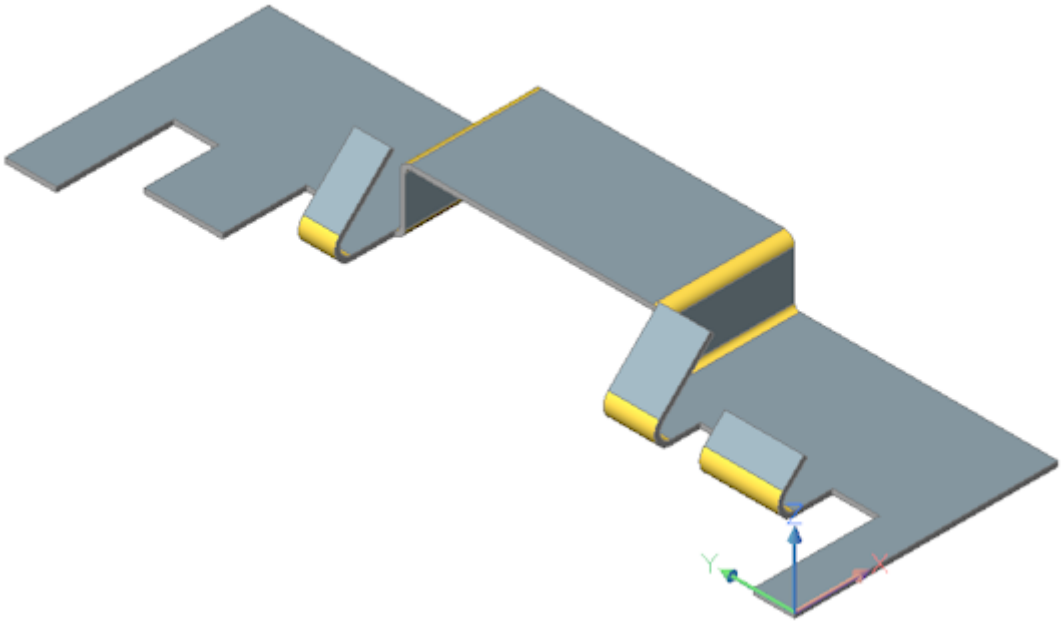
6 (opzionale) Selezionare l'opzione **Raggio**.

- Viene richiesto: Immettere il raggio di piegatura [Indietro] <Indietro>:

  Digitare un valore per ignorare il raggio di piegatura predefinito e premere **Invio**.

- VSelezionare la posizione della flangia [Angolo/Raggio/Commuta lato/COmmuta estensioni piegatura/ACcetta] <Accetta>:

  Immettere un valore per definire l'angolo di piegatura.



21.5.11.1 Collegare le flange

0

Il comando *SMFLANGECONNECT* chiude i vuoti tra 2 flange orientate arbitrariamente.

1 Avviare il comando *SMFLANGECONNECT*.

Viene richiesto: Selezionare facce dello spessore planari di due flange [opzioni di selezione (?)]:

La faccia della flangia sotto il cursore si evidenzia.

2 Selezionare la faccia dello spessore della prima flangia.

Viene richiesto:

Entità/Sottoentità nel gruppo: 1

Selezionare le facce dello spessore planare di due flange:

3 Selezionare la faccia dello spessore della seconda flangia.

Le facce selezionate verranno collegate.

Nota: Dopo aver connesso 2 flange, è possibile convertire la caratteristica giunzione in una piegatura.

21.5.12 Lavorare con Piegature e Giunzioni

Due flange adiacenti di parti in lamiera sono collegate da una piega o da una giunzione. Diversamente da molti altri sistemi CAD meccanici, in BricsCAD le piegature e le giunzioni sono caratteristiche separate. Una piegatura può essere commutata in una giunzione e viceversa, consentendo di sistemare o ottimizzare facilmente la rappresentazione spiegata di una parte in lamiera.

21.5.12.1 Conversione di Spigoli Vivi in Piegature o Giunzioni

Se le flange adiacenti di una parte in lamiera non sono collegate da una piegatura o giunzione, la parte non può essere spiegata o esportata in un sistema CAM. Pertanto, tali spigoli vivi devono essere convertiti in piegature o giunzioni.

Conversione di spigoli vivi in piegature

1 Eseguire una delle seguenti operazioni:
   - Passare con il mouse su uno spigolo vivo e fare clic sullo strumento **Crea Piegatura** ( ) nel menu **Quad**
   - Cliccare sul pulsante degli strumenti **Crea Piegatura** ( ) nella scheda **Lamiera** della barra multifunzione
   - Scegliere **Crea Piegatura** nel menu **Lamiera**
   - Digitare *SMBENDCREATE* nella barra dei comandi.

     Viene richiesto:

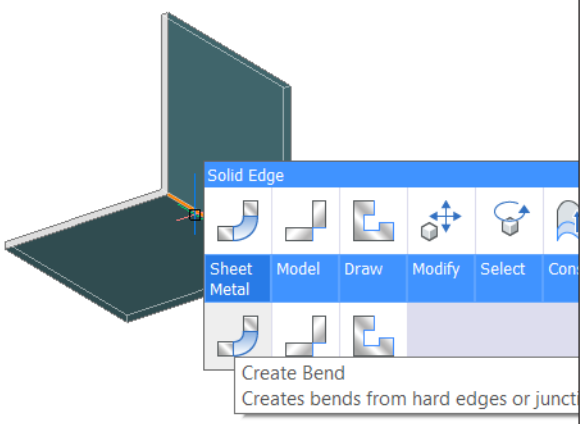
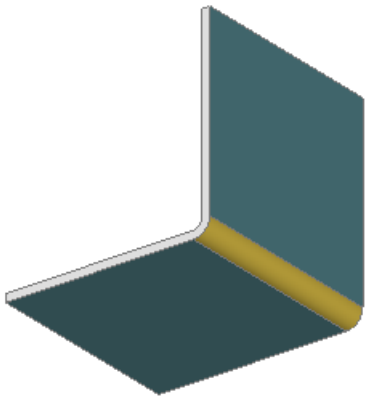
     Selezionare gli spigoli vivi o giunzioni o flange o solidi 3D o [intero Modello/opzioni di selezione (?)] <Intero modello>:

2 Selezionare una faccia di flangia o uno spigolo vivo.

   Viene richiesto:
   Entità nel gruppo di selezione: 1

   Selezionare gli spigoli vivi o giunzioni o flange o solidi 3D o [intero Modello/opzioni di selezione (?)] <Intero modello>:

3 Eseguire una delle seguenti operazioni:
   - Selezionare altre facce o spigoli vivi.
   - Cliccare il pulsante destro del mouse o premere **Invio** per creare le piegature o le giunzioni sugli elementi selezionati.

	
Spigolo vivo selezionato	Viene creata una piegatura

Convertire gli Spigoli Vivi in Giunzioni

1 Eseguire una delle seguenti operazioni:

- Passare con il mouse su uno spigolo vivo e cliccare sullo strumento **Crea Giunzione** ( ) nel menu **Quad**.
- Cliccare sul pulsante dello strumento **Crea Giunzione** ( ) nella barra multifunzione **Lamiera**.
- Scegliere **Crea Giunzione** nel menu **Lamiera**.
- Digitare *SMJUNCTIONCREATE* nella barra dei comandi.

  Selezionare spigoli vivi o piegature o flange o solidi 3D [intero Modello/opzioni di selezione (?)] <Intero modello>:

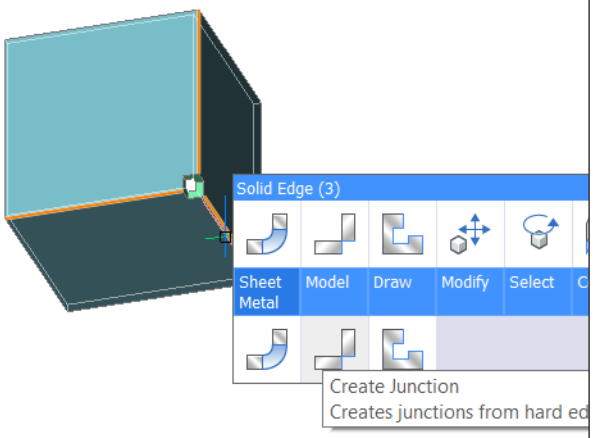
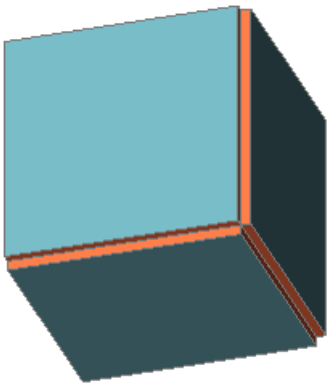
2 Selezionare una faccia di flangia o uno spigolo vivo.

Viene richiesto:
Entità nel gruppo di selezione: 1

Selezionare gli spigoli vivi o giunzioni o flange o solidi 3D o [intero Modello/opzioni di selezione (?)] <Intero modello>:

3 Eseguire una delle seguenti operazioni:

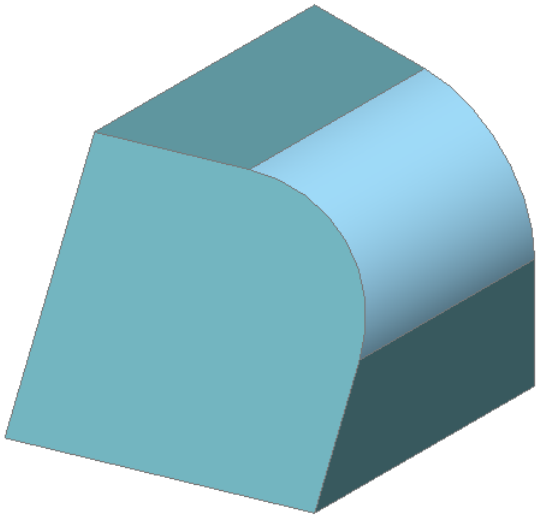
- Selezionare altre facce o spigoli vivi.
- Cliccare il pulsante destro del mouse o premere **Invio** per creare le piegature o le giunzioni sugli elementi selezionati.

	
Spigoli vivi selezionati. Si noti che è necessario creare prima uno scarico d'angolo.	Giunzioni create. Si noti che i tagli di materiale non necessari sono stati rimossi a causa delle **Caratteristiche Intelligenti**.

21.5.12.2 Creazione di Giunzioni Curve

Diamo un'occhiata a uno scenario più complesso in cui una giunzione deve essere tagliata su uno spigolo non lineare. Per rendere la parte illeggibile, dobbiamo:

- Tagliare la gola di scarico su uno degli spigoli vivi lineari.
- Tagliare le giunzioni sui rimanenti spigoli lineari e curvi.



- Crea una piegatura sul restante spigolo vivo.

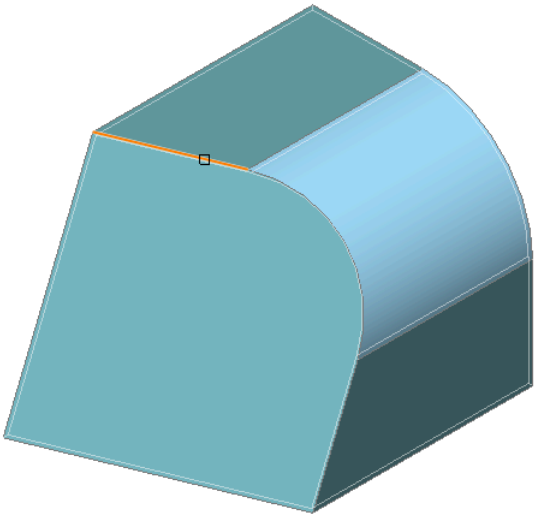
1 Creare una gola di scarico.

Eseguire una delle seguenti operazioni:

- Cliccare sullo strumento **Crea Scarico** ( ) nella scheda **Lamiera** della barra multifunzione.
- Scegliere **Crea Scarico** nel menu **Lamiera**.
- Digitare *SMRELIEFCREATE* nella barra dei comandi.

Viene richiesto:

Selezionare uno spigolo vivo o faccia della piegatura, faccia della flangia, solido 3D [intero Modello/opzioni di selezione (?)]:



2 Selezionare uno spigolo vivo.

Viene richiesto:

Selezionare un altro spigolo vivo o faccia della piegatura per lo scarico d'angolo [opzioni di selezione (?)]:

3 Premere **Invio** o cliccare il tasto destro del mouse.

Viene richiesto:

Immettere la dimensione della gola di scarico da rapporto del raggio di piegatura o [forza Gole di scarico/Auto] <Auto>:

4 Premere **Invio** o cliccare il tasto destro del mouse.

2 gole di scarico sono state create per lo spigolo:

Abbiamo creato 2 gole di scarico perché un lato dello spigolo vivo con un vertice "libero" necessita di uno scarico per rendere la faccia dello spessore ortogonale

.

Abbiamo creato una gola di scarico sul lato adiacente alla piegatura loft perché questa caratteristica è diversa da una piegatura normale: la creazione di uno scarico d'angolo sarebbe troppo grande

.

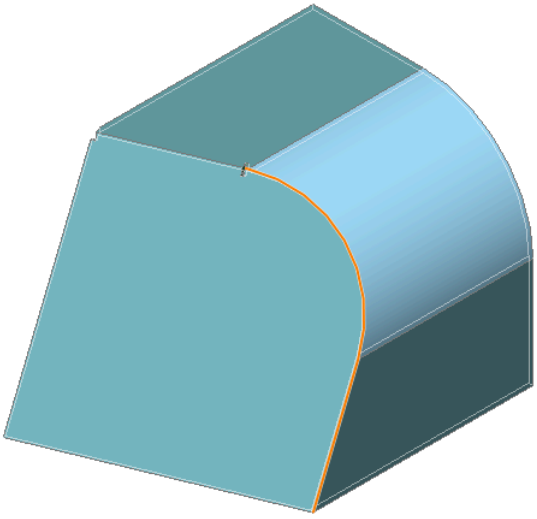
5 Creare 2 giunzioni.

Eseguire una delle operazioni seguenti:

- Cliccare sul pulsante dello strumento **Crea Giunzione** ( ) nella barra multifunzione **Lamiera**;
- Scegliere **Crea Giunzione** nel menu **Lamiera**;
- Digitare *SMJUNCTIONCREATE* nella barra dei comandi.

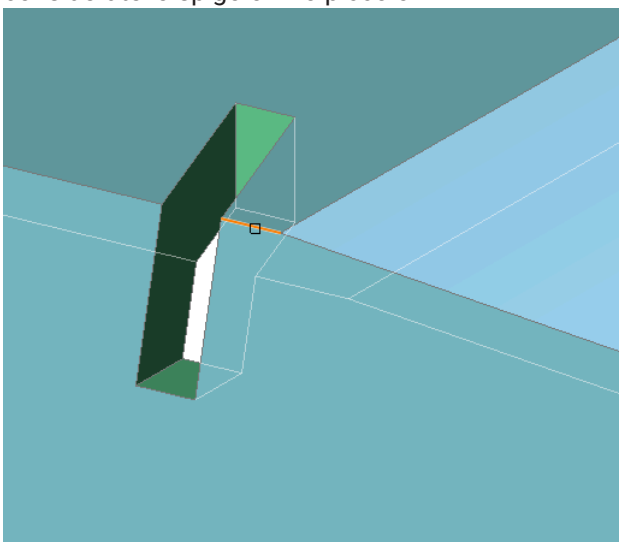
Verrà richiesto:

Selezionare spigoli vivi o piegature o flange o solidi 3D [intero Modello/opzioni di selezione (?)] <Intero modello>:



6 Selezionare i seguenti 3 spigoli vivi:

considerate lo spigolo vivo piccolo:



Il taglio è stato creato. Le parti che corrispondevano allo spigolo lineare ottengono normali caratteristiche giunzione.

7 Crea una piegatura sullo spigolo vivo. Poiché è l'unico spigolo vivo rimasto sul modello, applicare il comando all'intero modello.

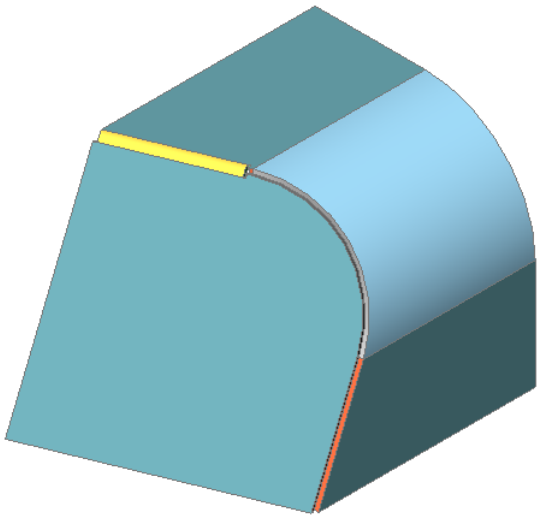
- Fare clic sul pulsante **Crea Piegatura** ( ) nella scheda della barra multifunzione **Lamiera**.
- Scegliere **Crea Piegatura** nel menu **Lamiera**
- Digitare *SMBENDCREATE* nella barra dei comandi.

  Viene richiesto:

Selezionare gli spigoli vivi o giunzioni o flange o solidi 3D o [intero Modello/opzioni di selezione (?)] <Intero modello>:

8 Premere **Invio**.

La piegatura è stata creata.



9 Spiegare la parte.

- Fare clic sul pulsante degli strumenti **Spiegamento del Corpo** ( ) nella scheda **Lamiera** della barra multifunzione.
- Scegliere **Spiegamento del Corpo** nel menu **Lamiera**.
- Digitare *SMUNFOLD* nella barra dei comandi.

  Viene richiesto:

  Selezionare una flangia o faccia di una piega con loft per iniziare lo spiegamento [Associativo/opzioni di selezione (?)]:

10 Selezionare una faccia della flangia.

Viene richiesto:

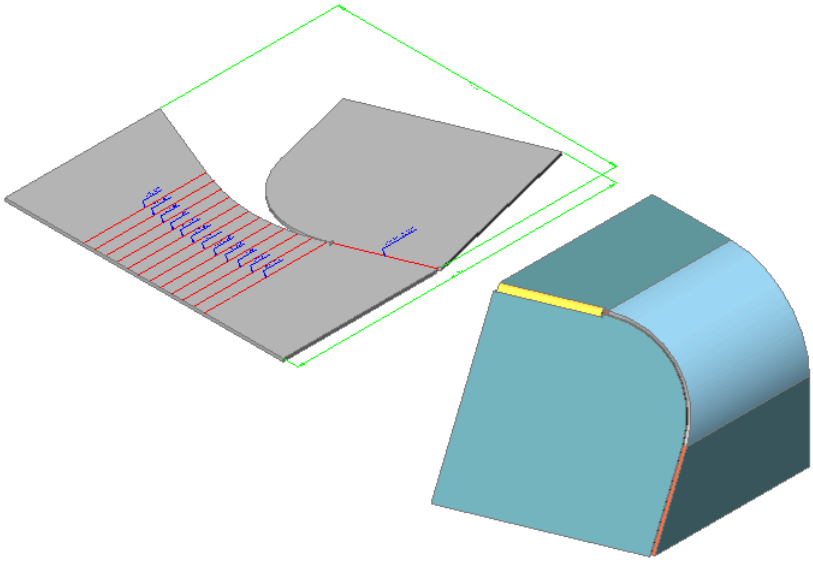
Seleziona la posizione del corpo spiegato:

11 Posizionare la parte spiegata.

Viene richiesto:

Convalidare il corpo spiegato e selezionare un'opzione [salva geometria 2D/salva geometria 3D/Posizionare la vista sul layout/Ottimizzare le annotazioni delle piegature/Mantieni] <Mantieni>:

12 Premere **Invio** per mantenere il



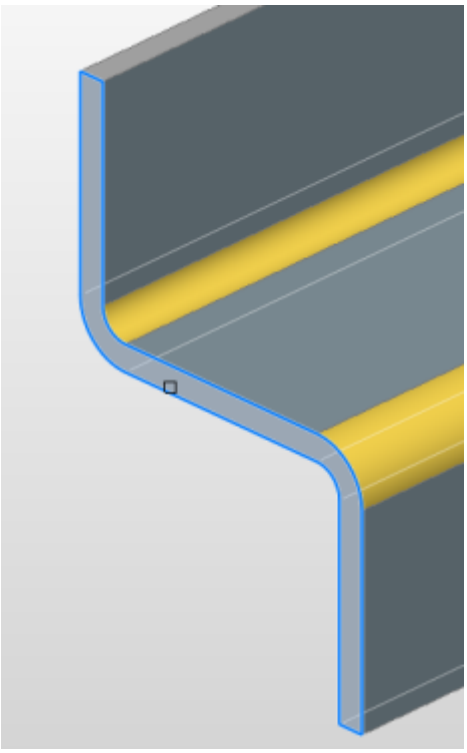
risultato.

21.5.12.3 Dividere una Faccia dello Spessore

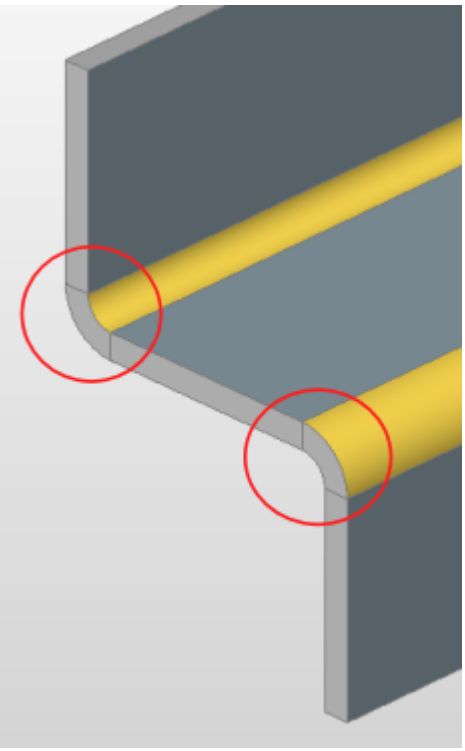
Per estendere una flangia che condivide la stessa faccia di spessore con altre flange, la faccia di spessore deve essere divisa.

1 Avviare il comando *SMIMPRINT* ( ).

Viene richiesto: Selezionare le facce di spessore:

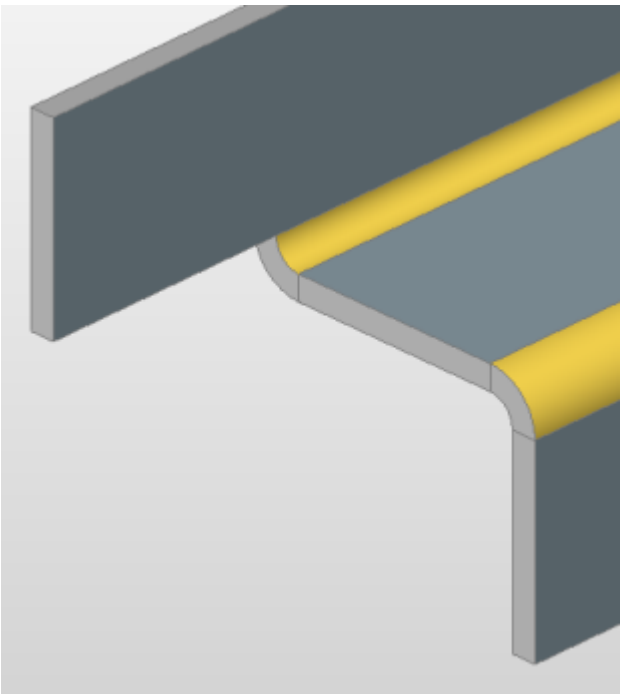


2 Selezionare la faccia dello spessore condiviso.



3 Cliccare il tasto destro o premere **Invio** per dividere la faccia.

4 Le flange ora possono essere estese separatamente:



5 Eseguire il comando *DMSIMPLIFY* per rimuovere le suddivisioni non necessarie, unificando così le facce dello spessore complanari.

Esercitazione: Suddivisione delle facce dello spessore in regioni

21.5.12.4 Modificare il raggio di piegatura

Modificare il raggio di piegatura complessivo:

1 Selezionare il nodo principale nel **Navigatore Meccanico**.

2 Trovare il gruppo **Raggio** nella parte inferiore del **Navigatore Meccanico**.

3 Eseguire una delle seguenti operazioni:
- Impostare **Tipo** su **Valore assoluto**, quindi digitare il valore del raggio nel campo **Valore**.

  Il campo **Raggio** cambia di conseguenza.
- Impostare **Tipo** su **Rapporto spessore**, quindi digitare un valore nel campo **Valore**.

  Il campo **Raggio** viene calcolato come prodotto dello **Spessore** e del **Rapporto spessore**.

Modificare il raggio di piegatura di una particolare piegatura:

1 Selezionare il nodo piegatura nel **Navigatore Maccanico**. La piegatura selezionata viene evidenziata nel modello. Le Proprietà di Piegatura vengono visualizzate nella parte inferiore del **Navigatore Meccanico**.

2 Eseguire una delle seguenti operazioni:
- Impostare il **Tipo** su **Usa valore globale**.

  Il raggio globale corrente della piegatura viene applicato alla piegatura selezionata.
- Impostare il **Tipo** su **Valore assoluto**, quindi digitare il valore del raggio nel campo **Valore**.

  Il campo **Raggio di curva** cambia di conseguenza.
- Impostare il **Tipo** su **Rapporto spessore** e digitare un valore nel campo valore.

  Il campo **Raggio** viene calcolato come prodotto dello **Spessore** e del **Rapporto spessore**.

21.5.12.5 Modificare il Vuoto della Giunzione

Per modificare il vuoto complessivo della giunzione:

1 Selezionare il nodo principale nel **Navigatore Meccanico**.

2 Trovare il gruppo **Vuoto giunzione** nella parte inferiore del **Navigatore Meccanico**.

3 Eseguire una delle seguenti operazioni:
- Impostare **Tipo** su **Valore assoluto**, quindi digitare il valore del vuoto nel campo **Valore**.

  Il campo del **Vuoto giunzione** cambia di conseguenza.
- Impostare **Tipo** su **Rapporto spessore**, quindi digitare un valore nel campo **Valore**.

  Il campo **Vuoto giunzione** viene calcolato come prodotto dello **Spessore** e del **Rapporto spessore**.

Per modificare il vuoto della giunzione per una particolare giunzione:

1 Selezionare il nodo piegatura nel **Navigatore Maccanico**. La piegatura selezionata viene evidenziata nel modello. Le Proprietà di Piegatura vengono visualizzate nella parte inferiore del **Navigatore Meccanico**.

2 Eseguire una delle seguenti operazioni:
- Impostare il **Tipo** su **Usa Valore globale**.

L'attuale vuoto globale di giunzione viene applicato alla giunzione selezionata.

- Impostare **Tipo** su **Valore assoluto**, quindi digitare il valore del vuoto nel campo **Valore**.

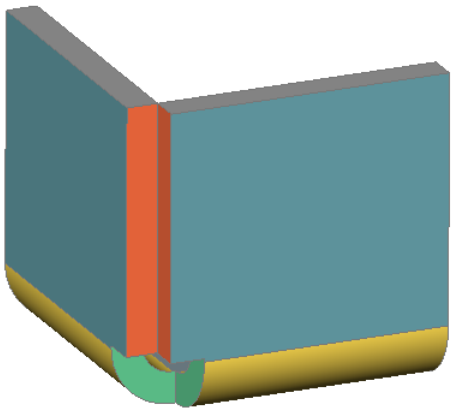
  Il campo **Vuoto giunzione** cambia di conseguenza.

- Impostare **Tipo** su **Rapporto spessore** e digitare un valore nel campo **Valore**.

  Il campo **Vuoto giunzione** viene calcolato come prodotto dello **Spessore** e del **Rapporto spessore**.

21.5.12.6 Modificare il Tipo Giunzione

Per impostazione predefinita, BricsCAD crea giunzioni simmetriche, come illustrato nell'immagine sottostante:

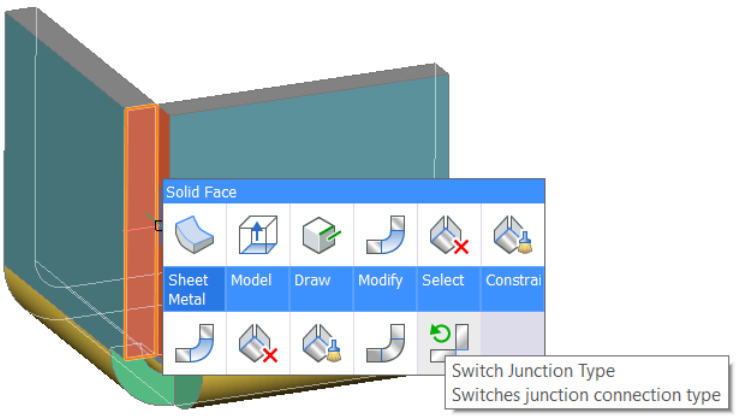


Il comando *SMJUNCTIONSWITCH* consente di modificare una caratteristica giunzione simmetrica in una con facce sovrapposte.

Per cambiare il tipo di giunzione utilizzando il menu Quad:

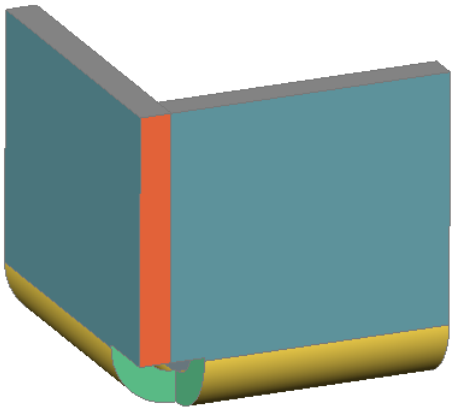
1 Posizionare il cursore su una faccia dello spessore di questa giunzione.

2 Selezionare **Cambia Tipo di Giunzione** nel gruppo di comando **Lamiera** nel menu



Quad.

La faccia di spessore selezionata viene allineata con la faccia flangia.



Modificare il tipo di giunzione nel Navigatore Meccanico

Metodo 1:

1 Selezionare la giunzione corrispondente nell'albero.

Le proprietà della giunzione vengono visualizzate nella parte inferiore del **Navigatore Meccanico**.

2 Selezionare la proprietà **Tipo di connessione**.

3 Scegliere il tipo di connessione desiderato.

Le opzioni sono: **Simmetrico**, **Allineato a Flange_X** o **Allineato a Flange_Y**.

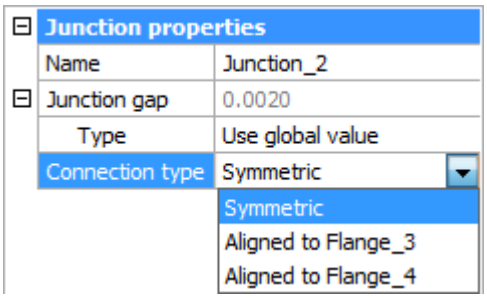


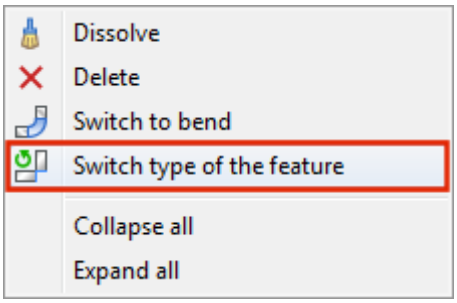
Figura 133: Proprietà giunzione

La giunzione viene aggiornata di conseguenza.

Metodo 2:

1 Selezionare la giunzione corrispondente nell'albero.

Viene visualizzato un menu contestuale:



2 Cliccare il pulsante destro del mouse e scegliere **Commuta il tipo di caratteristica** nel menu

contestuale.

Nota: Le giunzioni sia simmetriche sia allineate mantengono lo stesso valore di vuoto tra le flange collegate.

21.5.13 Lavorare con gli Orli

Una caratteristica orlo viene prodotta quando lo spigolo di una flangia della lamiera viene ripiegato su se stesso. *SMHEMCREATE* semplifica la creazione degli orli nella lamiera. In BricsCAD gli orli hanno il proprio gruppo di proprietà che dipende dal tipo di geometria, "flangia" e "piegatura". Le caratteristiche dell'orlo vengono riconosciute dal comando *SMCONVERT* e hanno un simbolo speciale se spiegate con *SMUNFOLD*.

21.5.13.1 Creare un Orlo

1 Eseguire una delle seguenti operazioni:
   - Selezionare più spigoli lineari sulla flangia
   - Passare il cursore su uno spigolo lineare della flangia.

2 Eseguire una delle seguenti operazioni:
   - Selezionare lo strumento **Crea un orlo chiuso** () nel **Quad** .
   - Cliccare sullo strumento **Crea orlo chiuso** () nella scheda **Lamiera** della barra multifunzione.
   - Scegliere **Creare Orlo Chiuso** nel menu **Lamiera**.
   - Digitare *SMHEMCREATE* nella barra dei comandi.

3 Posizionare il cursore per regolare la lunghezza dell'orlo:

4 Utilizzare il widget **Ctrl** o l'opzione **Attiva/disattiva rastremazione** per capovolgere la regolazione di un orlo sullo spigolo:

5 Al termine, premere **Invio**:

Verranno creati gli orli ed altre caratteristiche ausiliarie (scarichi e tagli ad angolo).

21.5.13.2 Modificare i Parametri dell'Orlo

Per modificare i parametri dell'Orlo, selezionare la caratteristica e utilizzare il **Navigatore Meccanico**. Sono disponibili diversi parametri, a seconda del tipo di orlo.

Tabella 43: Parametri Orlo

Tipo di orlo	Parametri disponibili
Chiusa	Raggio Lunghezza
Apri	Raggio Lunghezza

Tabella 43: Parametri Orlo

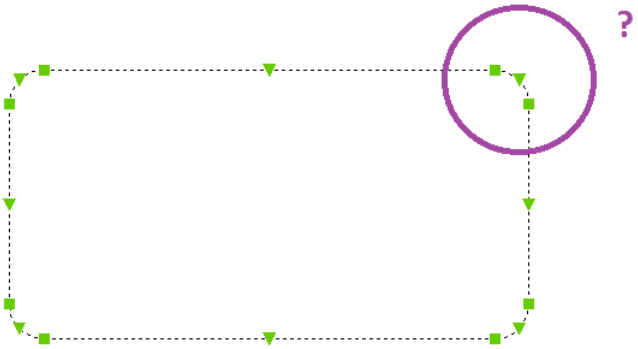
Tipo di orlo	Parametri disponibili
Orlo a Goccia	Raggio - modifica l'ango dell'orlo Lunghezza - modifica la lunghezza dell'orlo
Asolato	Raggio - cambia la parte "planare" dell'orlo

In caso di qualsiasi modifica parametrica, il tipo di orlo viene preservato e la geometria viene spostata all'interno, per mantenere le dimensioni esterne.

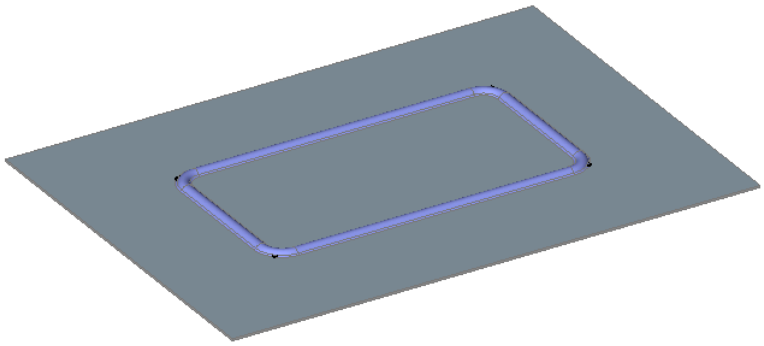
21.5.13.3 Cancellare un Orlo

Quando una caratteristica orlo viene cancellata, BricsCAD ripristina in modo intelligente le dimensioni e gli scarichi della geometria.

La parte della flangia in basso ha dimensioni esterne pari a 100.

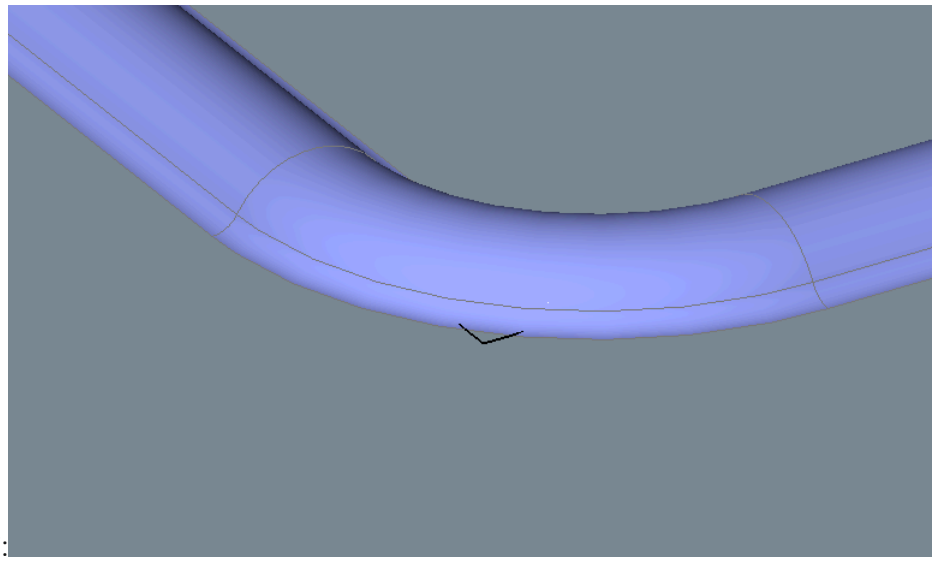


1 Utilizzare l'opzione Larghezza del comando *SMHEMCREATE* per creare un orlo parziale su un singolo

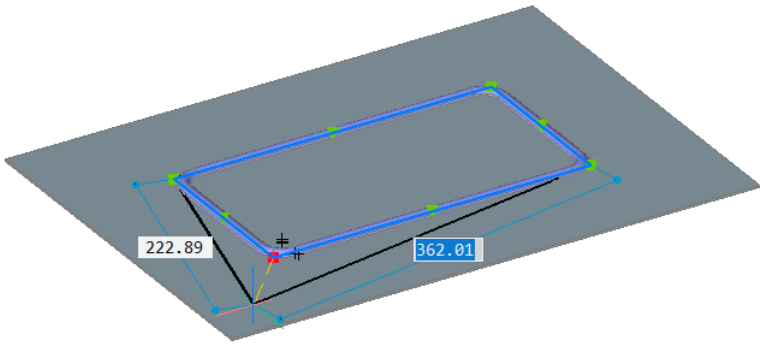


spigolo:

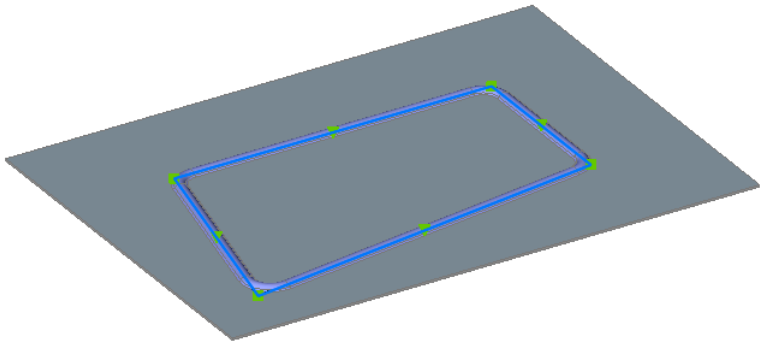
2 Creare orli su altri 2

spigoli: 

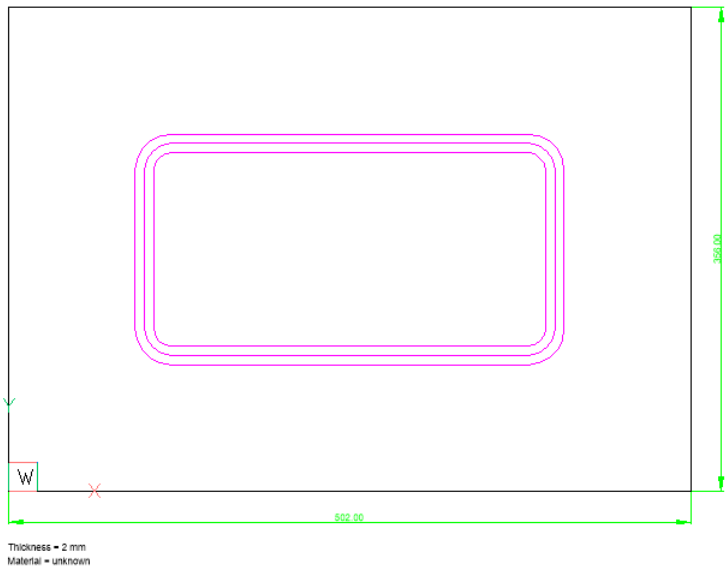
Queste operazioni portano all'aspetto di una complessa geometria dello scarico:



Nota: Le dimensioni esterne della parte vengono preservate.



3 Selezionare le caratteristiche dell'orlo ed



eliminale.

Alcune imperfezioni vengono lasciate sulla caratteristica scarico d'angolo e lo spigolo impresso:

4 Eliminare la caratteristica scarico d'angolo e utilizzare *DMSIMPLIFY* per ottenere la parte originale con le dimensioni originali.

21.5.13.4 Comandi Correlati

Il comando *MECHANICALBROWSEROPEN* visualizza il Navigatore Meccanico.

Il comando *SMDELETE* ripristina uno spigolo vivo tra 2 flange per rimuovere una piegatura o una giunzione - rimuove una flangia e tutte le piegature adiacenti. Le flange adiacenti vengono estese fino ad una configurazione della giunzione con la flangia da eliminare. Ripristina la geometria rimossa dal taglio ad angolo.

Il comando *SMHEMCREATE* crea uno o più orli per una parte in lamiera tirando uno o più spigoli di una flangia esistente.

21.5.14 Lavorare con le Linguette

Le caratteristiche linguetta vengono utilizzate per creare connessioni semplici e robuste tra le parti in lamiera.

Le caratteristiche linguetta vengono create dalle giunzioni.

21.5.14.1 Creazione di una linguetta Singola

Per impostazione predefinita, dal comando *SMTABCREATE* viene creata una singola linguetta. Questa caratteristica ha solo un unica fessura.

1 Eseguire una delle seguenti operazioni:
   - Passare il mouse sopra una faccia della giunzione e fare clic sullo strumento **Crea caratteristica linguetta singola** ( ) del **Quad**.
   - Fare clic sullo strumento **Crea Linguetta Singola** ( ) nella scheda **Lamiera** della barra

multifunzione.

- Scegliere **Crea Linguetta Singola** nel menu **Lamiera**.
- Digitare *SMTABCREATE* nella barra dei comandi.

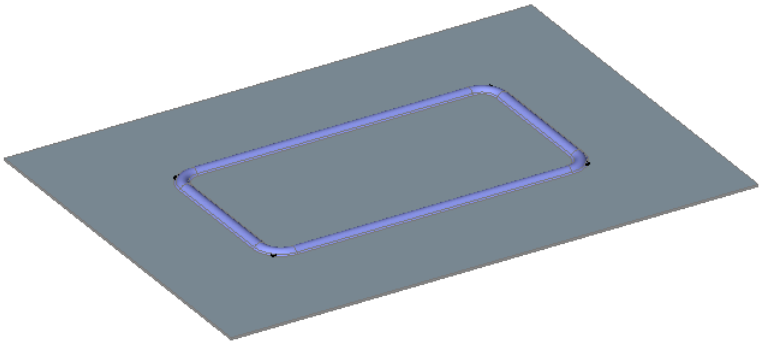
  Viene richiesto: Selezionare la giunzione per creare una linguetta singola o [Serie linguette/ opzioni di selezione (?)]:

2 Seleziona la faccia di una giunzione.

Viene richiesto: Inserire la lunghezza della linguetta o [punto Centrale]:

3 Definire la lunghezza della linguetta.

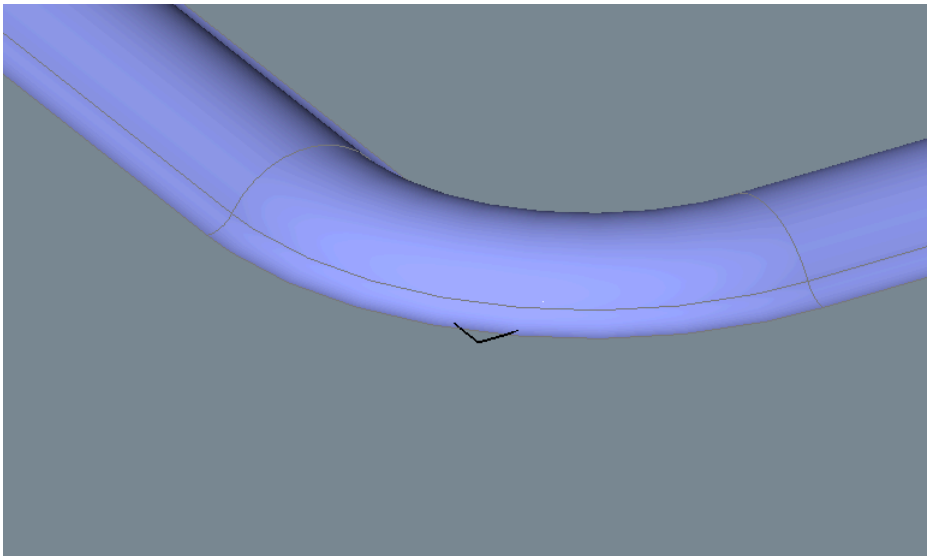
Una linguetta viene posizionata simmetricamente rispetto a un punto centrale. Digitare i valori nei campi dinamici di immissione della lunghezza per definire la lunghezza della linguetta.



4 Selezionare l'opzione **Punto centrale** per posizionare la linguetta.

Viene richiesto: Selezionare il punto centrale o [Lunghezza]:

2 dimensioni dinamiche controllate dal tasto Tab vengono utilizzate per creare una linguetta relativa a una giunzione:



5 Selezionare l'opzione **Lunghezza** per specificare la lunghezza oppure premere **Invio** per completare il comando.

21.5.14.2 Creazione di una Serie di Linguette

A differenza delle linguette Singole, una caratteristica Serie di linguette possiede più fessure.

1 Per creare una serie di linguette, effettuare una delle seguenti operazioni:
   - Passare il mouse sopra una faccia della giunzione e fare clic sullo strumento **Crea caratteristica serie di linguette** ( ) del **Quad**.
   - Fare clic sullo strumento **Crea Serie Linguette** ( ) nella scheda **Lamiera** della barra multifunzione.
   - Scegliere **Crea Serie Linguette** nel menu **Lamiera**.
   - Digitare *SMTABCREATE* nella barra dei comandi.

     Viene richiesto: Selezionare la giunzione per creare una linguetta singola o [Serie linguette/opzioni di selezione (?)]:

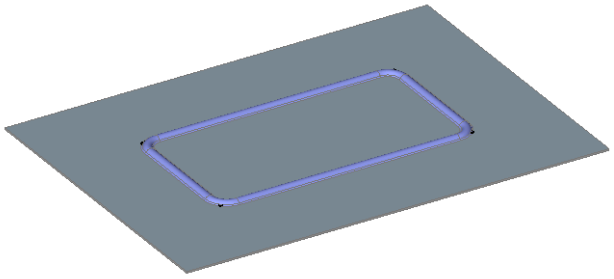
2 Seleziona l'opzione **Serie**.

   Viene richiesto: Selezionare la giunzione o lo spigolo vivo curvo per creare la serie di linguette o [linguetta curva Invertita/linguetta Singola/opzioni di selezione (?)]:

3 Selezionare una faccia della giunzione.

   Viene richiesto: Accettare la configurazione della linguetta o [Inverti linguetta/Numero di fessure/Distanza/Lunghezza/raggio CIRcolare/distanza CIma/Accetta] <Accetta>:

4 È possibile utilizzare le opzioni per modificare l'aspetto di una serie di linguette (vedere l'esempio seguente).
   - **Numero Fessure = 3** - nota che l'immagine non è simmetrica. Se fai clic sull'opzione **Inverti**, sull'altro lato della giunzione verranno visualizzati 3 slot.
   - **Lunghezza = 8** - la lunghezza della fessura
   - **Distanza = 40** - l'interasse tra i centri delle

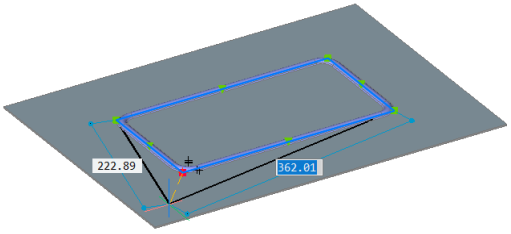
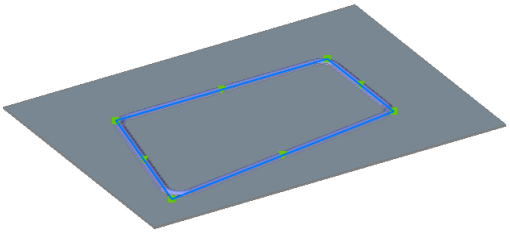


   fessure.

5 Modificare il **Raggio circolare** e la **Distanza Cima** per definire i raccordi e gli smussi per gli spigoli.

   Suggerimento: Se si impostano i raccordi o le cimature per errore, utilizzare il valore zero per disattivarli.

6 Una volta impostate tutte le opzioni, scegliere **Accetta** o premere **Invio** per creare la linguetta e terminare il comando.

Raccordi	Cima
	

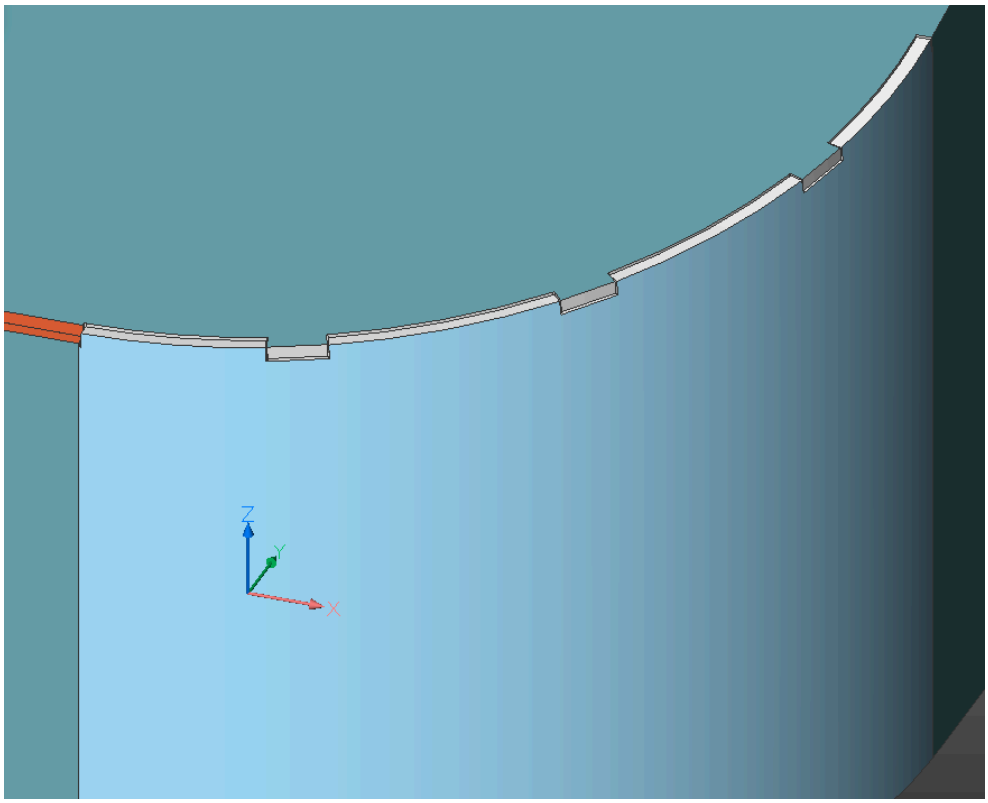
21.5.14.3 Creazione di una Serie di Linguette Curve

Questo caso è simile alla creazione di una una serie dritta di linguette, ma le linguette vengono posizionate lungo un percorso curvo.

1 Per creare una serie di linguette curve, effettuare una delle seguenti operazioni:
   - Passare il puntatore del mouse su uno spigolo vivo curvo e fare clic sullo strumento **Crea caratteristica serie di linguette** ( ) nel menu **Quad**.
   - Fare clic sullo strumento **Crea Serie Linguette** ( ) nella scheda **Lamiera** della barra multifunzione.
   - Scegliere **Crea Serie Linguette** nel menu **Lamiera**.
   - Digitare *SMTABCREATE* nella barra dei comandi.

     Viene richiesto: Selezionare la giunzione per creare una linguetta singola o [Serie linguette/ opzioni di selezione (?)]:

2 Selezionare uno spigolo vivo curvo.

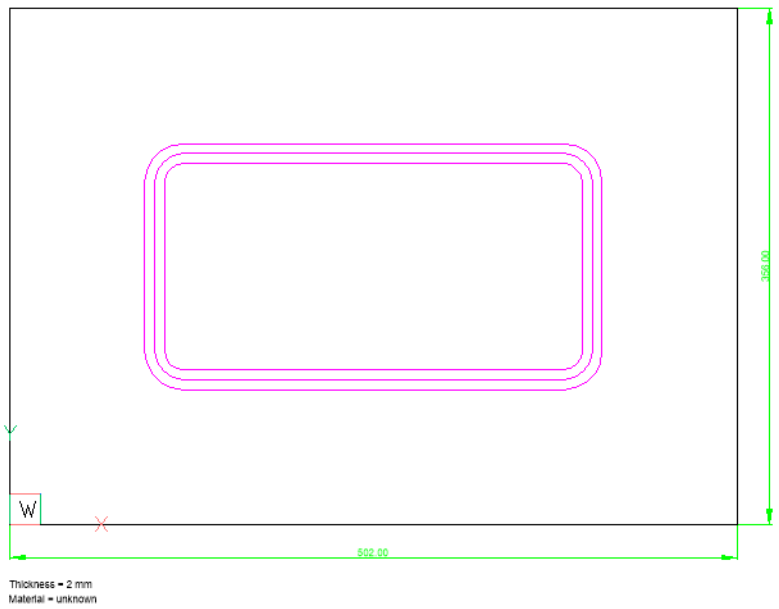


21.5.14.4 Modifica delle Linguette

Le serie di linguette o le linguette singole hanno diverse proprietà modificabili, disponibili nel **Navigatore Meccanico**.

Cliccare sul nome della caratteristica della linguetta da modificare.

Una linguetta singola possiede parametri **Lunghezza**, **Spazio libero**, **Raggio di raccordo** e **Distanza di cimatura**. **Spazio libero della linguetta** definisce il divario tra la linguetta e la fessura.



Una serie di linguette possiede i parametri **Numero di fessure**, **Distanza**, **Lunguezza**,**Spazio libero**, **Raggio di raccordo** e **Distanza di cimatura**.

Nota: Le Serie di Linguette Curve non possono essere modificate.

21.5.14.5 Eliminazione delle Linguette

Utilizzare il comando *SMDELETE* per eliminare le linguette (non applicabile per le curve). Viene creata una caratteristica giunzione al posto delle linguette.

21.5.14.6 Esplosione delle Linguette

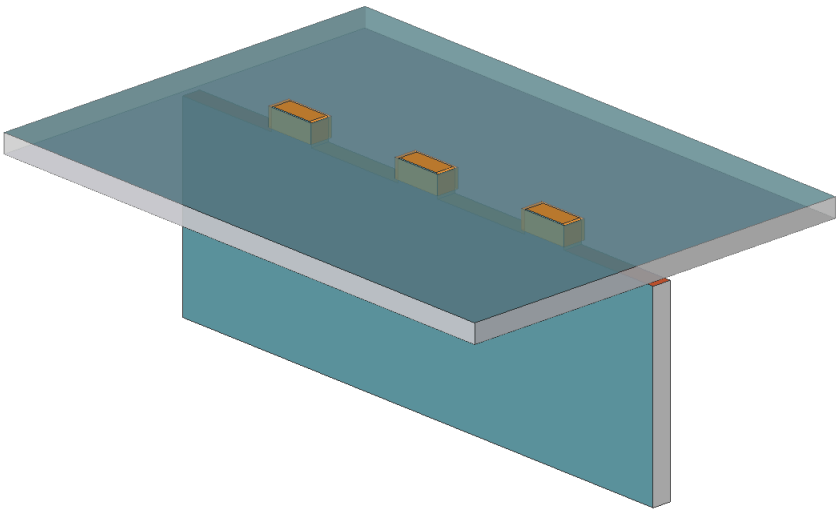
Utilizzare il comando *SMEXPLODE* per esplodere una giunzione a linguette con un gruppo di caratteristiche linguetta. La geometria della linguetta rimane invariata e le sue facce vengono distribuite tra diverse caratteristiche di giunzione.

Caratteristica linguetta originale	Risultato della cancellazione	Risultato dell'esplosione

21.5.15 Lavorare con linguette a T e giunzioni a T

21.5.15.1 Panoramica

È possibile utilizzare le caratteristiche linguette a T per creare una connessione con un posizionamento robusto tra 2 flange in lamiera. A differenza delle linguette normali, le linguette a T formano una configurazione a T, quando una flange è collegata ad un'altra.



Le linguette a T possono essere create dalle giunzioni a T dal comando *SmTabCreate*. Le caratteristiche giunzione a T sono spesso utilizzate come fase intermedia per la creazione delle linguette a T.

21.5.15.2 Creazione di una caratteristica a T

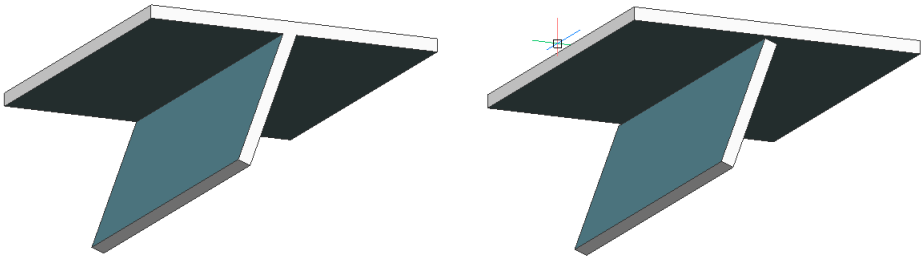
È possibile creare una caratteristica giunziona a T su flange separate esistenti nel seguente modo:

1 Selezionare una faccia dello spessore sulla flangia.

2 Eseguire una delle seguenti operazioni:
- Cliccare sul pulsante dello strumento **Connetti Flange** ( ) nella scheda **Lamiera | Giunzione** .
- Scegliere **Connetti Flange** nel menu **Lamiera | Giunzione**.
- Digitare *SMFLANGECONNECT* nella barra dei comandi.

3 Premere **Invio**.

In alternativa, è possibile creare una caratteristica giunzione a T al posto di uno spigolo rigido simile a un T tra 2 flange (immagine a sinistra):



1 Passare il puntatore del mouse su uno spigolo vivo tra le flange e fare clic sullo strumento **Crea Giunzione** ( ) nel menu **Quad** oppure eseguire una delle seguenti operazioni:
- Fare clic sul pulsante dello strumento **Crea Giunzione** ( ) nella barra multifunzione **Lamiera**.
- Scegliere **Crea giunzione** dal menu **Lamiera | Giunzione**.

- Digitare *SMJUNCTIONCREATE* nella barra dei comandi.

2 Selezionare uno spigolo vivo.

21.5.15.3 Creazione di una Linguetta Singola

Il comando *SMTABCREATE* crea una singola linguetta a T per impostazione predefinita. Questa caratteristica possiede solo un'unica fessura.

Per creare una singola linguetta a T:

1 Passare il mouse su una faccia della giunzione a T e fare clic sullo strumento **Crea caratteristica linguetta singola** ( ) nel menu **Quad** oppure esegui una delle seguenti operazioni:

- Fare clic sullo strumento **Crea Linguetta Singola** ( ) nella scheda **Lamiera** della barra multifunzione.
- Scegliere **Crea Linguetta Singola** dal menu **Lamiera | Linguetta**
- Digitare *SMTABCREATE* nella barra dei comandi.

2 Selezionare una faccia della giunzione a T.

21.5.15.4 Creazione di una Serie di Linguette

Differentemente dalle linguette a T singole, la caratteristica serie di linguette a T possiedono più fessure.

Per creare una serie di linguette a T:

1 Passare il puntatore del mouse su una faccia della giunzione a T e fare clic sullo strumento **Crea caratteristica serie di linguette** ( ) nel menu **Quad** oppure eseguire una delle seguenti operazioni:

- Fare clic sullo strumento **Crea Serie Linguette** ( ) nella scheda **Lamiera** della barra multifunzione.
- Scegliere **Crea Serie Linguette** dal menu **Lamiera | Linguetta**.
- Digitare *SMTABCREATE* nella barra dei comandi.

  Viene richiesto: Selezionare la giunzione per creare una linguetta singola o [Serie linguette/ opzioni di selezione (?)]: selezionare l'opzione **Serie di linguette**.

2 Selezionare una faccia della giunzione a T.

21.5.15.5 Modifica dei parametri delle giunzione a T e linguetta a T

Una giunzione a T possiede solo il parametro **Vuoto giunzione**.

Le serie di linguette a T e le linguette a T singole possiedono le seguenti proprietà: **Distanza**, **Lunghezza**, **Spazio libero**, **Raggio di raccordo** e **distanza di cimatura**. Le serie di linguette a T possiedono anche il parametro **numero di fessure**.

È possibile impostare il valore di tutti i parametri nel pannello **Navigatore Meccanico**.

21.5.15.6 Eliminazione di una giunzione a T o di una linguetta a T

Quando viene eliminata una caratteristica linguetta a T, BricsCAD ripristina una caratteristica di giunzione a T con la geometria corrispondente. L'eliminazione di una giunzione a T unisce le flange e crea uno spigolo vivo a T tra di esse.

21.5.15.7 Comandi Correlati

Il comando *MECHANICALBROWSEROPEN* visualizza il Navigatore Meccanico.

Il comando *SMDELETE* elimina una caratteristica.

21.5.16 Lavorare con gli Smussi

21.5.16.1 Panoramica

Le caratteristiche Smusso rappresentano un'operazione di cianfrinatura effettuata sulle parti in lamiera. A livello di BricsCAD V21 ogni smusso dovrebbe trovarsi esattamente su una singola flangia. Non esistono operazioni specifiche per creare uno smusso, quindi deve essere utilizzato il comando *DMCHAMFER* su uno spigolo di una flangia e quindi il comando *DMROTATE* per regolare l'angolo dello smusso. Il comando *SMCONVERT* riconosce gli smussi che quindi possono essere processati dal comando *SMUNFOLD*. Le caratteristiche smusso sono supportate anche dai comandi *SMDELETE* e *SMREPAIR*.

Questa procedura si applica a 2 scenari:

- Lavorando con una geometria importata, inclusa l'esportazione in batch da parte di *SMASSEMBLYEXPORT*.
- Modificando le parti.

Distinguiamo tra i seguenti tipi di smussatura:

Tabella 44: Tipo di Smusso

	Smusso su un lato

Tabella 44: Tipo di Smusso

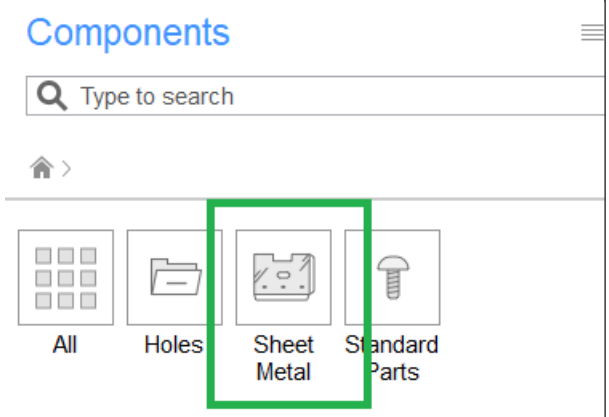
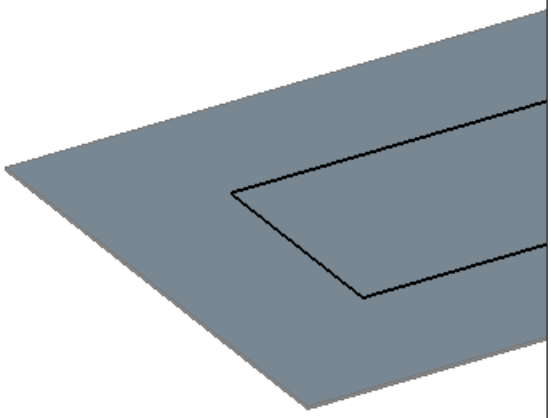
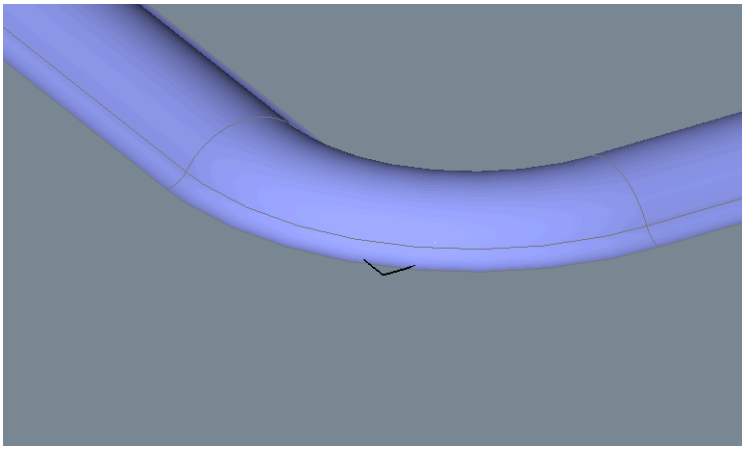
	Smusso a 45°
	Smusso a K

Tabella 44: Tipo di Smusso

	Smusso a X

Nel **Navigatore Meccanico**, le proprietà di ogni smusso sono disponibili separatamente. Queste proprietà sono:



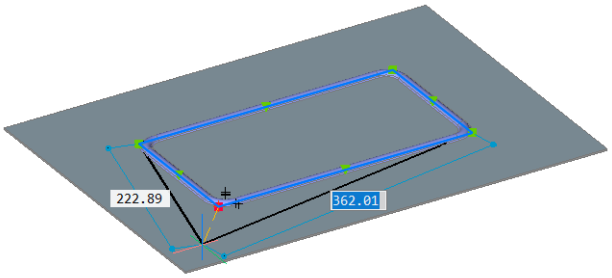
Profondità (H): la profondità dell'operazione di cianfrinatura, come eseguita dal comando *DMCHAMFER*.

Angolo (A): l'inclinazione della faccia smussata.

Nota: Nelle proprietà del Navigatore Meccanico, Alto o Basso non vengono utilizzati per descrivere le caratteristiche dello smusso.

21.5.16.2 Riconoscere gli smussi

Iniziamo con una parte in lamiera con geometria smussata (spigoli della flangia smussati).



1 Cliccare sul pulsante dello strumento **Converti in Lamiera** ( ) nel pannello **Lamiera / Crea** della barra multifunzione.

   Viene richiesto: Selezionare solidi 3D o facce seme o [Modello intero/opzioni di selezione (?)] <Intero modello>:

2 Premere **Invio** per convertire il solido 3D in una parte in lamiera.

Le caratteristiche degli smussi vengono riconosciute e appaiono in **Navigatore Meccanico** e visualizzate sulla parte:

21.5.16.3 Elimina e Ripara

Il seguente percorso di andata e ritorno non modifica le dimensioni complessive della parte: Creare la geometria e la caratteristica smusso utilizzando i comandi *DMCHAMFER*/*SMCONVERT*, quindi eliminare lo smusso utilizzando *SMDELETE* e riavviare la procedura.

Quando si utilizza *SMDELETE* per eliminare uno smusso, il volume viene **sempre aggiunto** alla parte perché le dimensioni della parte non possono essere ridotte.

Anche il supporto nel comando *SMREPAIR* è importante, poiché questo comando fa parte del flusso di lavoro Importa e Rielabora.

Nota: Se si applica il comando *SMREPAIR* a una parte con smussi ma senza caratteristiche smusso, gli smussi andranno persi e le dimensioni cambieranno, a seconda del lato selezionato della parte.

21.5.16.4 Spiegamento

Le caratteristiche smusso vengono spiegate in base all'impostazione **Modalità spiegamento** nel **Navigatore Meccanico**.

Le opzioni sono:

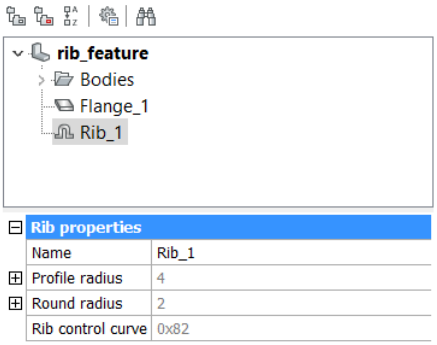
Mantieni: non fa nulla con la geometria smussata, risultando in un solido 3D con smussi. Queste facce smussate produrranno probabilmente degli artefatti nei file .DXF 2D.

Rimuovi: stesso risultato dell'applicazione di *SMDELETE* su tutte le caratteristiche smusso: tutti i tagli vengono riempiti.

Annotazione: ogni caratteristica smusso viene rimossa e lo spigolo viene annotato. Questa è la modalità consigliata.

Questo è il risultato per la parte nell'esempio:

- Ogni spigolo smussato è stato annotato con un testo rosa sul layer **Annotazioni smussi**.
- SOPRA e SOTTO sono rispetto al lato selezionato per lo spiegamento. Quando si sceglie il lato opposto, tutte le annotazioni SOPRA diventano SOTTO e viceversa.
- Per gli smussi **su un lato** e **a 45°** è necessaria una sola operazione di cianfrinatura: viene creata una singola annotazione SOPRA o SOTTO.
- Per gli smussi a **K** e a **X** sono necessarie 2 operazioni di cianfrinatura: vengono create annotazioni SOPRA e SOTTO.



21.5.17 Creare e Dividere Piegature con Loft

BricsCAD consente di creare o riconoscere le parti in lamiera loft con geometria curvilinea, che può essere fabbricata utilizzando piegature multiple.

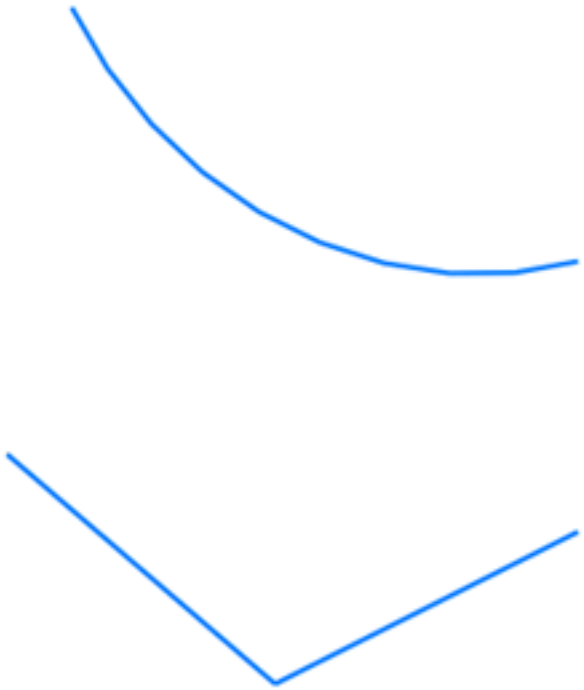
21.5.17.1 Procedura Generale per Creare una Parte di Lamiera con una Piegatura con Loft

1 Avviare il comando *SMLOFT*.

2 Eseguire una delle seguenti operazioni:

- Fare clic sullo strumento **Crea Piegatura con Loft** ( ) nella barra degli strumenti **Lamiera**.
- Fare clic sullo strumento **Lamiera Loft** ( ) nella barra multifunzione **Lamiera**.
- Scegliere **Create Lamiera Loft** nel menu **Lamiera**.

  Viene richiesto: Selezionare due sezioni trasversali [opzioni di selezione (?)]:



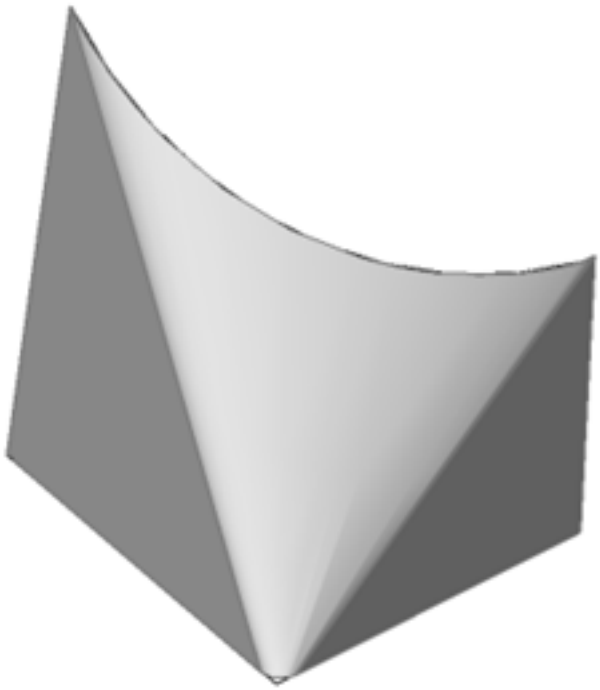
3 Selezionare 2 entità di profilo.

Viene richiesto: Imposta [Raggio dei raccordi/Spessore/piegatura SIngola/Flange con piegature] <Flange con piegature>:

4 Premere **Invio** per accettare le impostazioni predefinite.

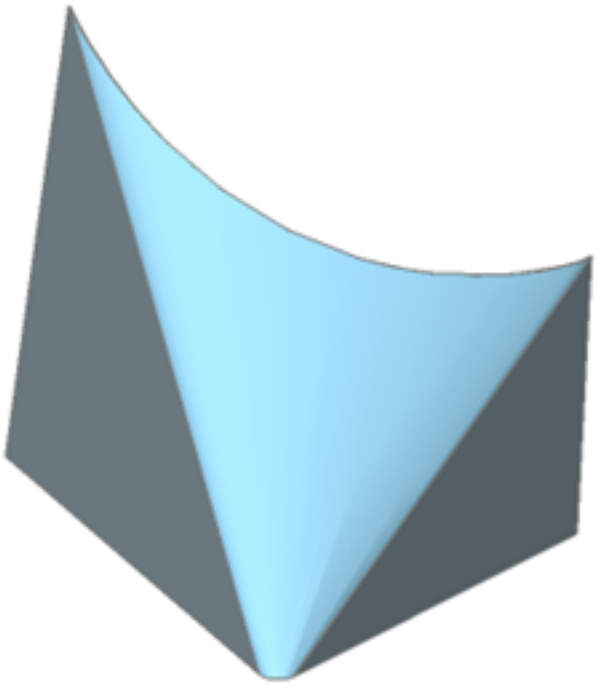
Viene creata un'anteprima della lamiera.

Viene richiesto:Imposta [Raggio dei raccordi/Spessore/piegatura SIngola/Flange con piegature/Ispessire interno/Entrambi i lati/ispessire ESterno] <Ispessire interno>:



5 Premere il tasto **CTRL** per scegliere la posizione dello spessore della parte in lamiera rispetto ai profili di origine: **Sinistra**, **Centro** o **Destra**.

Se **TIPS** è ON, viene visualizzato il widget suggerimenti.



6 Premere **Invio** per creare la parte in lamiera.

Se **FEATURECOLORS** è ON, è possibile distinguere tra le 2 flange ( ) e la piegatura loft ( ).

Le proprietà della piegatura loft vengono visualizzate nel **Navigatore Meccanico**:

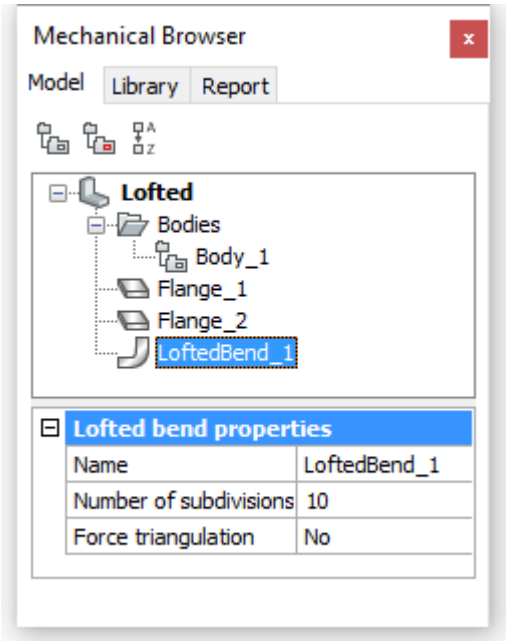


Figura 134: Proprietà delle piegature con loft

Facoltativamente, selezionare la caratteristica piegatura loft per modificarne le proprietà:

- **Numero di suddivisioni**: il numero di regioni piatte nella rappresentazione spiegata in 2d di una caratteristica di piegatura loft (maggiore è il numero, più linee di piegatura verranno aggiunte alla vista spiegata in 2d).

- **Triangolazione forzata**: la maggior parte delle piegature loft può essere sviluppata in 2D utilizzando un algoritmo intelligente di tassellazione quadrupla. Tuttavia a volte potrebbe produrre risultati non ottimali. In questi casi, impostare questa proprietà su *Sì* per passare ad un algoritmo più diretto che crea triangoli simili a quadrati.

Esercitazione: Piegature con Loft in BricsCAD Sheet Metal

21.5.17.2 Dividere una Piegatura con Loft

Le piegature con loft possono essere divise lungo una linea sulla superficie loft in un punto arbitrario:

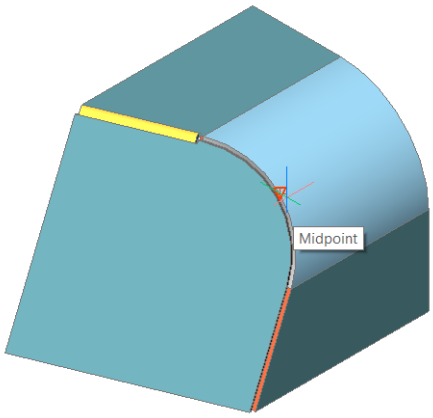
1 Eseguire una delle seguenti operazioni:
   - Cliccare sullo strumento **Dividi** ( ) sulla scheda **Lamiera** della barra multifunzionale.
   - Scegliere **Dividi** nel menu **Lamiera**.
   - Digitare **smsplit** nella barra dei comandi.

   Viene richiesto: Selezionare flangia e faccia della piegatura con loft:

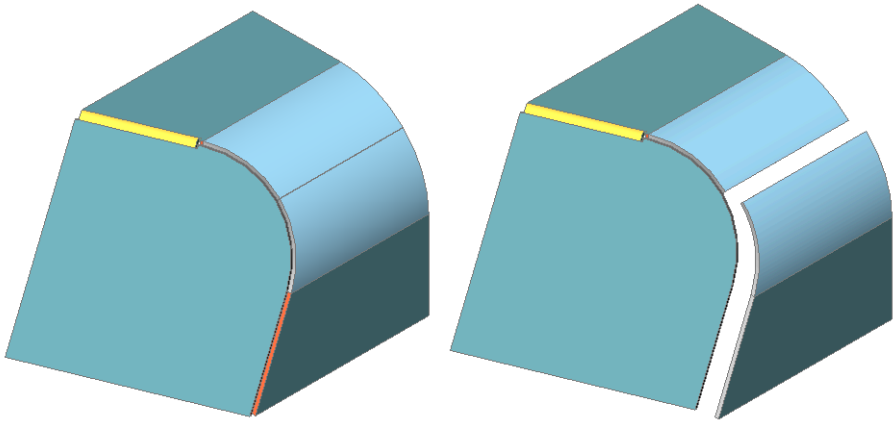
2 Selezionare una faccia di una piegatura con loft.

   Viene richiesto: Selezionare il punto sulla faccia di piegatura loft:

3 Selezionare un punto sulla superficie della piegatura con loft.



La piegatura loft è stata divisa. Se la giunzione della connessione è dissolta, è possibile spostare liberamente la parte separata.



21.5.18 Operazioni con le Gole di Scarico e gli Scarichi d'Angolo

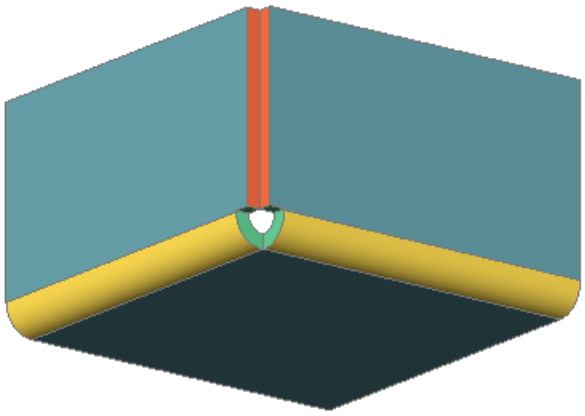
21.5.18.1 Panoramica

Utilizzare i comandi *SMFLANGEEDGE*, *SMRELIEFCREATE*, *SMJUNCTIONSWITCH* e *SMRELIEFSWITCH*.

Gli scarichi sono necessari nelle parti in lamiera per evitare che il materiale si strappi durante la piegatura. Fondamentalmente non devi preoccuparti degli scarichi, perché BricsCAD crea automaticamente tutti gli scarichi necessari quando crei una piegatura (implicitamente o esplicitamente). Tuttavia, se si rielaborano parti in lamiera povere, potrebbe essere utile creare gole e scarichi di piegatura con gli strumenti dedicati.

21.5.18.2 Scarichi d'Angolo e Giunzioni

Quando si crea una flangia utilizzando il comando *SMFLANGEEDGE*, adiacente a uno spigolo di piegatura, viene creato automaticamente uno scarico d'angolo nell'angolo in cui si incontrano le 3 flange. Inoltre, viene creata una giunzione tra le 2 flange, che non sono collegate con una curva.



Esercitazione: Progettare da Zero la Lamiera in BricsCAD

21.5.18.3 Creazione di Scarichi Mancanti

1 Eseguire una delle seguenti operazioni:

- Fare clic sullo strumento **Crea Scarico** ( ) nella scheda **Lamiera** della barra multifunzione.
- Scegliere **Crea Scarico** nel menu **Lamiera**.
- Digitare *SMRELIEFCREATE* nella barra dei comandi.

  Viene richiesto: Selezionare uno spigolo vivo o faccia della piegatura, faccia della flangia, solido 3D [intero Modello/opzioni di selezione (?)]:

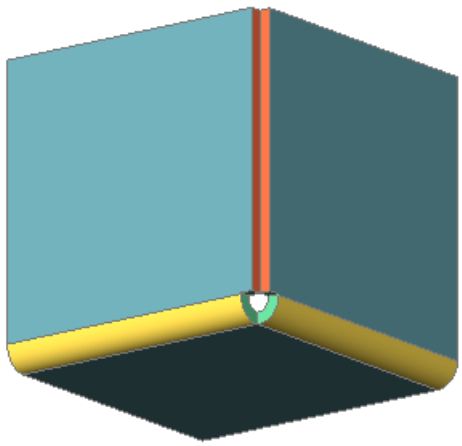
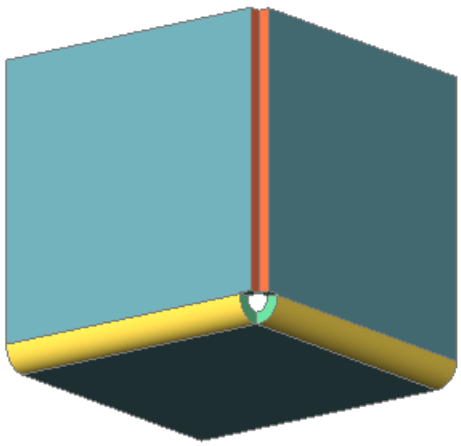
2 Selezionare un elemento.

Immettere la dimensione della gola di scarico da rapporto del raggio di piegatura o [forza Gole di scarico/Auto] <Auto>:

3 Eseguire una delle seguenti operazioni:

- Premere **Invio** o cliccare il pulsante destro del mouse per accettare il rapporto raggio di curvatura 1.0.
- Digitare un valore nella barra dei comandi, quindi premere **Invio**.

  Lo scarico è stato creato.

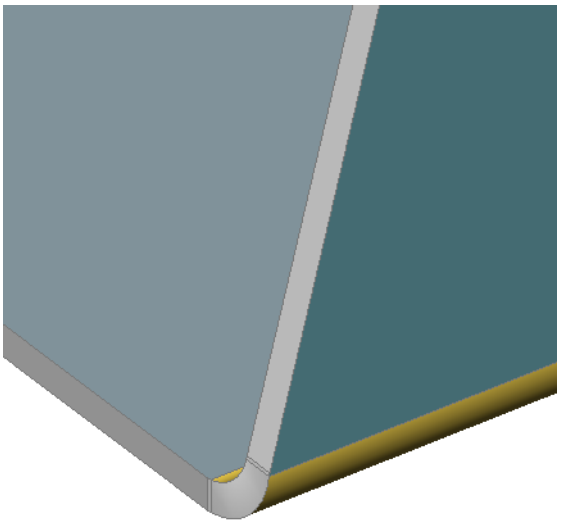
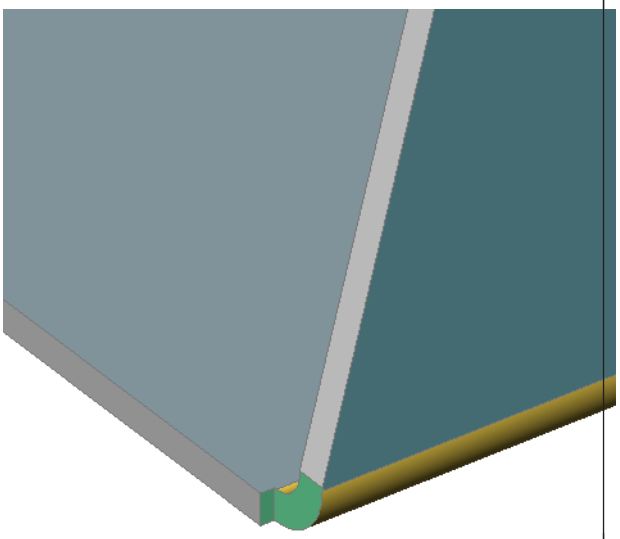
	
Facce di piegatura selezionate	Scarichi creati

Nota:

- Gli elementi selezionati possono essere un solido con caratteristiche di lamiera, una faccia o uno spigolo vivo del solido 3D.
- Per creare uno scarico localmente, è possibile selezionare 1 o 2 spigoli vivi, 1 o 2 piegature o una piegatura ed uno spigolo vivo.
- Se viene selezionato un solido 3D il comando riconosce i punti dove è necessario inserire gole di scarico e scarichi d'angolo e li crea automaticamente.

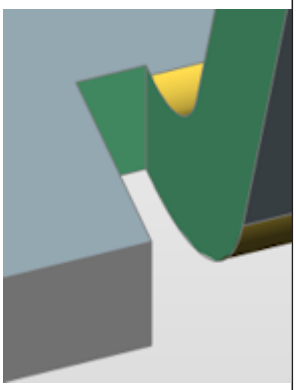
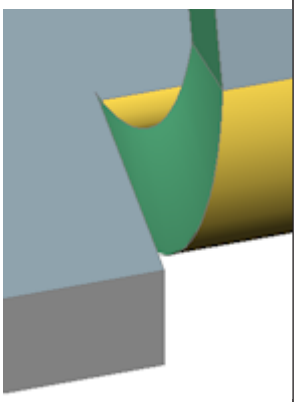
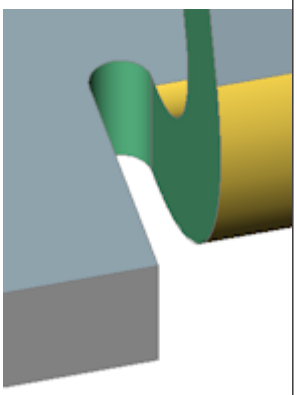
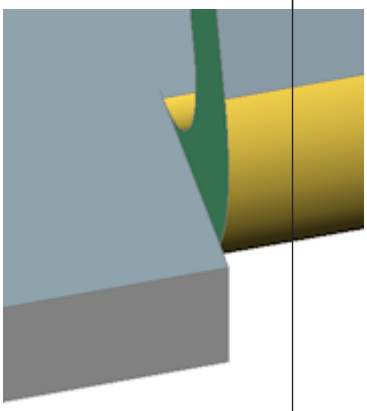
- Se vengono selezionati 2 spigoli vivi o 2 piegature o una piegatura ed un spigolo vivo in un angolo, il comando tenta di creare uno scarico d'angolo.

L'opzione **forza Gole di scarico** forza la creazione di una gola di scarico della piegatura anche se non è necessario creare uno scarico di piegatura. Come mostrato nell'immagine sottostante, non è necessario creare uno scarico per la piegatura, ma la geometria è diversa dal taglio rettangolare.

	
Uno scarico di piegatura non è necessario a causa di una transizione graduale della geometria.	Uno scarico di piegatura rettangolare viene creato solo se l'opzione **forza Gole di scarico** è abilitata.

21.5.18.4 Commutazione dei Tipo di Gola di Scarico

Il tipo di gola di scarico può essere commutato tra quattro tipi:

			
Rettangolo	Uniforme	Asolato	Lacerato

Nota:

- Un scarico **Lacerato** ha una larghezza molto piccola di 0,001 * spessore. Se necessario rimuovere completamente la gola di scarico, modificandone la larghezza a 0. Questa operazione rimuove sia la geometria, sia la caratteristica della gola di scarico.
- I tipi di gole di scarico **Uniforme**, **Asolato** e **Lacerato** possono essere applicati solo se il tipo di gola di scarico corrente è **Rettangolare**.

Utilizzare la barra multifunzione, il menu o la barra degli strumenti

1 Per scegliere il tipo di scarico desiderato. Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante a discesa **Scarico** nel pannello **Lamiera / Modifica** sulla barra multifunzione e selezionare **Commuta a Scarico Rettangolare**, **Commuta a Scarico Uniforme**, **Commuta a Scarico Asolato** o **Commuta a Scarico Lacerato**.
   - Cliccare sul pulsante dello strumento sulla barra degli strumenti **SM Scarico**: **Commuta a Scarico Rettangolare** (), **Passa a Scarico Uniforme** (), **Commuta a Scarico Asolato** () o **Commuta a Scarico Lacerato** ().
   - Scegliere il tipo di scarico nel menu **Lamiera / Scarico**.

     Viene richiesto: Selezionare facce, solidi 3d per commutare scarichi in tipo-V [RETtangolare/CIrcolare/tipo-V/Uniforme/Lacerato/Asolato/modello Intero/opzioni di selezione (?)] <Modello intero>:
2 Eseguire una delle seguenti operazioni:
   - Premere **Invio** per applicare il tipo di scarico selezionato all'intero modello.
   - Fare clic su una flangia o piegatura adiacente o su una faccia della gola di scarico.

     La gola di scarico verrà convertita.
3 Se il comando è stato chiamato dalla barra multifunzione, il comando ripete la richiesta principale fino a quando non si preme il tasto **ESC** per interromperlo.

Utilizzare il Quad

1 Passare il mouse sulla flangia o piegatura adiacente o su una faccia della gola di scarico.

   Se l'opzione **Seleziona Facce** della variabile di sistema **SELECTIONMODES** è selezionata, la flangia o la faccia di evidenzia; altrimenti tenere premuto il tasto **CTRL** fino a quando la faccia non si evidenzia.
2 Selezionare il tipo di scarico desiderato nella scheda **Lamiera** nel menu **Quad**.

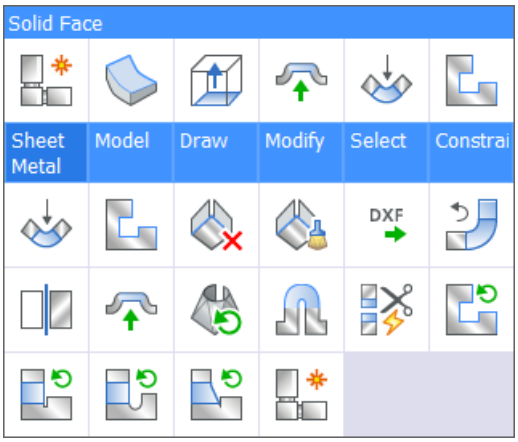
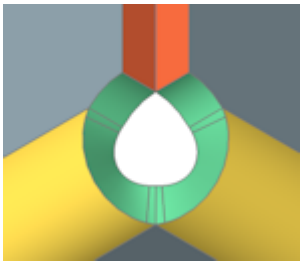
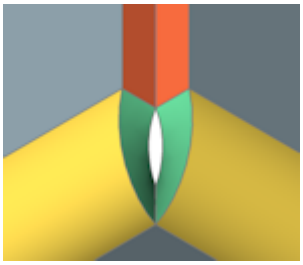


Figura 135: Lamiera

21.5.18.5 Commutazione del tipo di Scarico d'Angolo

Il tipo di scarico d'angolo può essere commutato tra 3 tipi:

		
Rettangolo	Circolare	Tipo-V

Utilizzare la barra multifunzione, il menu o la barra degli strumenti:

1 Per scegliere il tipo di scarico desiderato. Eseguire una delle seguenti operazioni:
   - Fare clic sul pulsante a discesa **Scarico** del pannello **Lamiera / Modifica** della barra multifunzione e selezionare un tipo di scarico.
   - Cliccare su un pulsante della barra degli strumenti **SM Scarico**.
   - Scegliere il tipo di scarico nel menu **Lamiera / Scarico**.

     Viene richiesto: Selezionare facce, solidi 3d per commutare scarichi in tipo-V [RETtangolare/CIrcolare/tipo-V/Uniforme/Lacerato/Asolato/modello Intero/opzioni di selezione (?)] <Modello intero>:

2 Eseguire una delle seguenti operazioni:
   - Premere **Invio** per applicare il tipo di scarico selezionato all'intero modello.
   - Fare clic su una flangia o piegatura adiacenti o una faccia dello scarico d'angolo.

     Lo scarico d'angolo verrà convertito.

3 Se il comando è stato chiamato dalla barra multifunzione, il comando ripete la richiesta principale fino

a quando non si preme il tasto **ESC** per interromperlo.

Utilizzare il Quad:

1 Assicurarsi che sia selezionata l'opzione **Seleziona Facce** della variabile di sistema **SELECTIONMODES**.

2 Passare il mouse sopra la flangia o piegatura vicina oppure una faccia dello scarico d'angolo.

3 Selezionare il tipo di scarico desiderato nel gruppo di comandi **Lamiera** nel menu **Quad**.

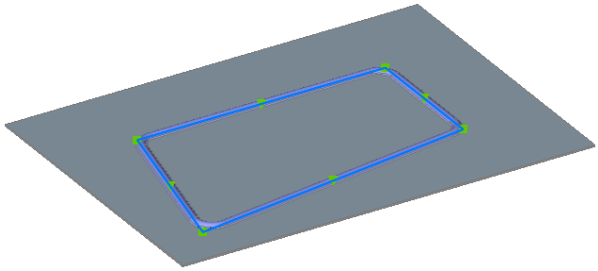


Figura 136: Lamiera

21.5.19 Lavorare con le Caratteristiche di Forma e Nervatura

21.5.19.1 Panoramica

Le Caratteristiche Forma e Nervatura vengono create da uno strumento di formatura. Le caratteristiche nervatura sono prodotte imprimendo una ruota che rotola sulla flangia seguendo una traiettoria arbitraria. Useremo il termine 'forma' per fare riferimento ad entrambe le caratteristiche di forma e nervatura. Le caratteristiche di forma possono essere riconosciute nella geometria importata o inserite da librerie incorporate o definite dall'utente. Le caratteristiche di forma possono essere modificate direttamente o in modo parametrico. Le caratteristiche di forma sono elencate nel **Navigatore Meccanico** con i relativi parametri. I parametri delle caratteristiche di forma vengono visualizzati e possono essere modificati nel **Pannello Proprietà** quando selezionate.

I seguenti comandi si applicano alle caratteristiche forma:

Icona	Comando	Descrizione
	SMFORM	Converti un gruppo selezionato di facce in una caratteristica forma. Un file di disegno con una caratteristica forma definita dall'utente può essere salvato e quindi inserito con il comando *BMINSERT*.
	SMCONVERT	Riconosce le caratteristiche di forma nella geometria importata.
	SMREPLACE	Sostituisce le caratteristiche forma (comprese quelle riconosciute) presenti nelle parti in lamiera, con le caratteristiche forma della libreria utente o incorporata.

Icona	Comando	Descrizione
	SMRIBCREATE	Crea caratteristiche forma a nervatura associative su parti in lamiera da profili 2D. La caratteristica viene aggiornata automaticamente quando il profilo di definizione viene modificato.
	BMINSERT	Inserisce un componente meccanico esistente o una caratteristica forma di lamiera nel disegno corrente.
	SMSELECT	Seleziona caratteristiche forma comparabili. Il comando funziona in 2 modalità: • **Simili**: caratteristiche che hanno lo stesso insieme di facce, ma eventualmente dimensioni diverse. • **Identiche**: caratteristiche identiche.

Una libreria di caratteristiche di forma parametrizzate è disponibile nella scheda Caratteristica Forma nel pannello **Tavolozze degli Strumenti**. La libreria contiene i tipi di caratteristiche di forma più utilizzati. Per inserire una caratteristiche di forma in un solido 3D di lamiera, trascinare l'icona della caratteristiche di forma dal pannello **Tavolozze degli Strumento** o dal pannello **Componenti** su una flangia del solido.

La variabile di sistema *SMFORMFEATUREUNFOLDMODE* controlla l'aspetto delle caratteristiche forma nelle rappresentazioni del modello spiegato in 2D e 3D.

21.5.19.2 Inserire una Caratteristiche Forma

Le caratteristiche di forma vengono aggiunte a una parte in lamiera mediante l'inserimento nelle flange esistenti. Le caratteristiche di forma possono essere inserite mediante il comando *BMINSERT* o trascinandole dalla scheda **Caratteristiche Forma** del pannello **Tavolozze degli Strumenti**.

1 Assicurati che le **Quote Dinamiche** (DYN) e l'**UCS Dinamico** (UCSD) siano attivi.

2 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Inserisci Caratteristica Forma** ( ) sulla barra degli strumenti **Lamiera**.
- Cliccare sul pulsante dello strumento **Inserisci Caratteristica Forma** ( ) sulla barra multifunzione **Lamiera**.
- Scegliere **Inserisci Caratteristica Forma** nel menu **Lamiera**.
- Digitare *BMINSERT* nella barra dei comandi, quindi premere **Invio**.

  La finestra di dialogo *Seleziona il file da inserire* visualizza la cartella **Form Features** [cartella di installazione di BricsCAD]\Support\DesignLibrary\SheetMetal\FormFeatures:

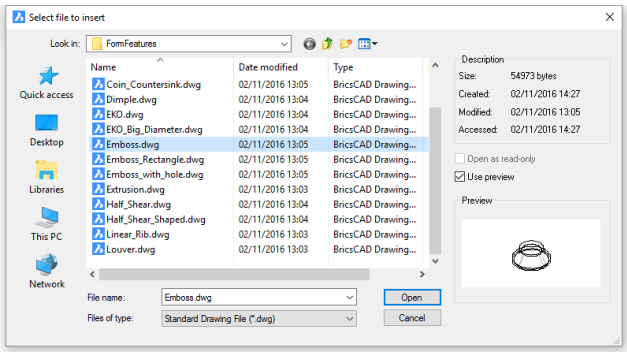


Figura 137: Seleziona il file da inserire

Selezionare la caratteristica di forma desiderata, quindi fare doppio clic sulla caratteristica o fare clic su **Apri**.

- Selezionare la caratteristica forma dalla scheda Caratteristiche Forma del pannello **Libreria**:

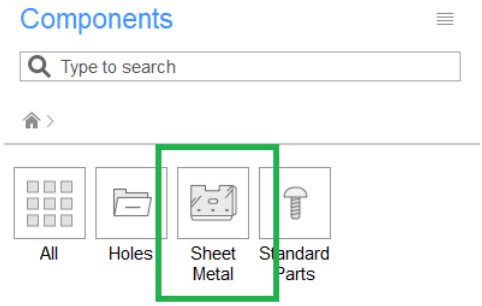


Figura 138: Componenti

- Selezionare la caratteristica forma nella scheda **Caratteristica Forma** nel pannello **Tavolozze degli Strumenti**:

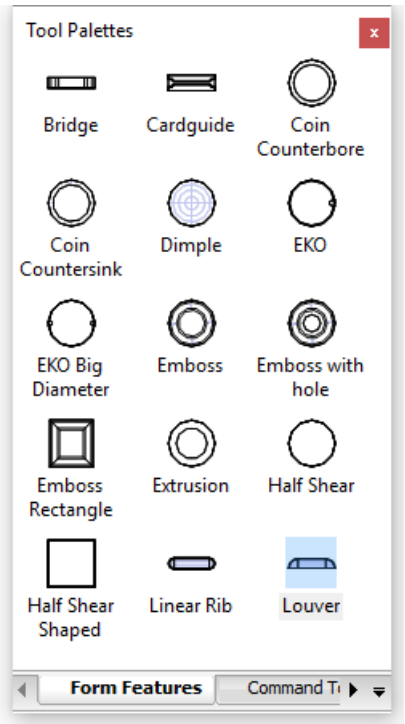


Figura 139: Tavolozze degli Strumenti

3 Spostare il cursore sulla faccia della flangia in cui si desidera inserire la caratteristica.

L'UCS dinamico si allinea alla faccia selezionata.

L'orientamento dell'UCS dipende dallo spigolo in cui il cursore entra nella faccia.

Le quote dinamiche vengono visualizzate dall'origine dell'UCS al punto di inserimento della caratteristica di forma.

Viene richiesto:

Selezionare punto di inserimento o [Edita entità inserita/Ruota componente/impostare punto Base/Nome/Tipo di inserimento/INverti/SErie/Cambia i solidi 3d di destinazione]:

4 (opzionale) Scegliere un'opzione (vedi il comando *BMINSERT* per maggiori dettagli):

- **Edita Entità Inserita**: Modifica i parametri dell'elemento selezionato.
- **Ruota Componente**: Definisce l'angolo di rotazione del componente.
- **Imposta Punto Base**: Specifica un punto base diverso per inserire il componente.
- **Nome**: Definisce il nome dell'elemento del componente.
- **Tipo di inserimento**: Specifica il tipo di inserimento: locale o esterno.
- **Inverti**: Inserisce la caratteristica di forma nella faccia opposta della flangia.
- **Multiplo**: inserisce più elementi dello stesso componente o creare una serie.

5 Per inserire una caratteristica forma, effettuare una delle seguenti operazioni:

- Specificare un punto.
- Digitare una distanza nel campo della dimensione dinamica. Premere il tasto **TAB** per andare avanti e indietro tra i campi di immissione dinamica.

6 Ripetere il passaggio 5 per inserire un'altra copia o premere **Invio** per interrompere.

Al fine di avere Caratteristiche di Forma nel pannello **Componenti** è necessario creare o utilizzare il **Profilo Mechanical** nella finestra di dialogo **Avvio di BricsCAD**.

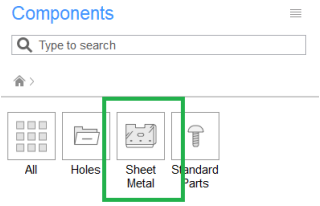


Figura 140: Componenti

21.5.19.3 Modificare i Parametri di una Caratteristica Forma della Libreria

Le caratteristiche forma della libreria corrispondono a componenti parametrici. Diversi parametri controllano le loro dimensioni. I parametri per controllare Lunghezza, Larghezza, Altezza e Raggio della caratteristica forma sono denominati corrispondentemente L, W, H e R. Lo spessore della caratteristica forma viene impostato automaticamente come lo spessore della parte in lamiera in cui viene inserita.

I parametri di una singola caratteristica forma possono essere modificati nel **Pannello delle Proprietà** o nel **Navigatore Meccanico**. I parametri di più caratteristiche forma possono essere modificati solo nel **Pannello delle Proprietà**.

Metodi per selezionare le caratteristiche forma

- Selezionare l'opzione **Seleziona Facce** della variabile di sistema *SELECTIONMODES*, quindi cliccare sulle caratteristiche forma nel modello.
- Selezionare la(e) caratteristica(he) nel **Navigatore Meccanico**, quindi premere la **barra spaziatrice** o cliccare il pulsante destro del mouse e scegliere **Seleziona** nel menu contestuale. Tenere premuto il tasto **CTRL** per selezionare più caratteristiche forma.
- Una parte in lamiera può avere molte caratteristiche forma diverse, che potrebbero complicare operazioni come l'eliminazione o la sostituzione, specialmente se deve essere elaborata solo una selezione di caratteristiche, che soddisfano determinate condizioni. Per selezionare più facilmente le caratteristiche forma, il comando **SmSelect** consente di selezionare caratteristiche forma identiche a una caratteristica selezionata o simile.

Modificare i parametri nelle Proprietà

1 Selezionare le caratteristiche forma.

2 Selezionare un parametro nel nodo **Parametri** delle **Proprietà**.

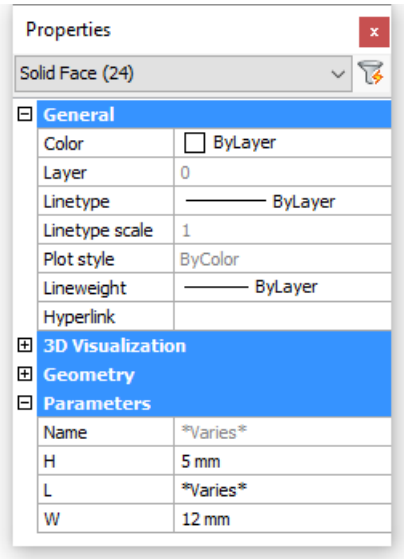


Figura 141: Proprietà

Modificare i parametri nel Navigatore Meccanico

1 Selezionare la caratteristica forma nel nodo **Forma** dell'albero della struttura del componente.

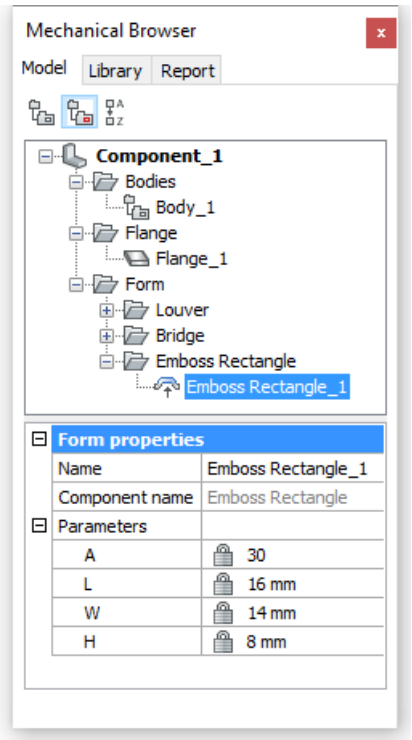
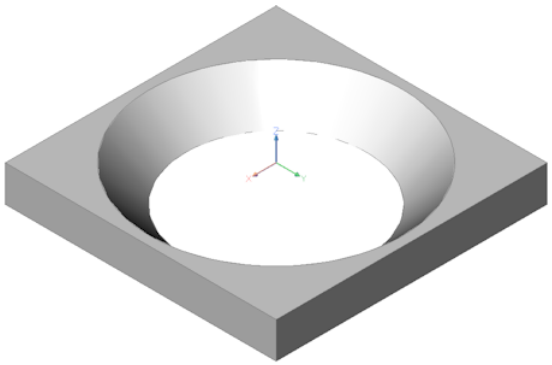


Figura 142: Navigatore Meccanico

2 Selezionare un parametro nel pannello inferiore del **Navigatore Meccanico** e digitare un nuovo valore.

21.5.19.4 Creare una caratteristica forma personalizzata

1 Aprire un nuovo disegno utilizzando il template **Mechanical-metric.dwt**.



2 Creare la geometria 3D della caratteristica forma.

3 Eseguire il comando *SMCONVERT* ( ).

4 Eseguire il comando *BMMECH* ( ).

Viene richiesto: Nome del componente meccanico< Component_2>:

5 Digitare un nome per la caratteristica forma.

Viene richiesto: Convertire blocchi e riferimenti esterni in componenti meccanici o [Sì - per convertire tutti/No - per saltare la conversione/opzioni di selezione (?)] <Sì - per convertire tutti>:

6 Selezionare l'opzione **No**.

7 Nel campo **Tipo estensione** in **Navigatore Meccanico**, selezionare **Caratteristica di Forma Lamiera**.

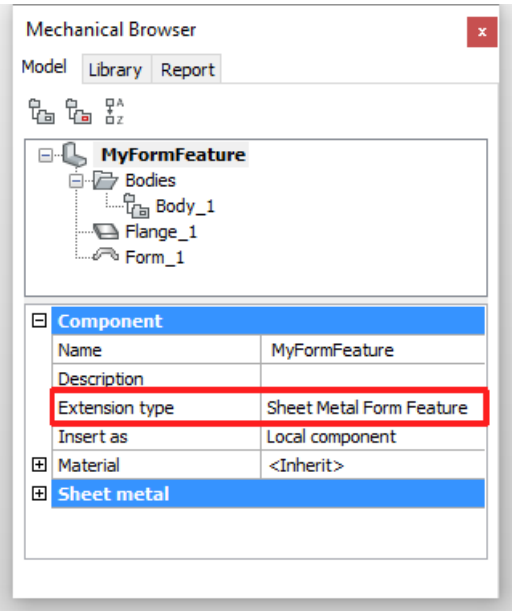


Figura 143: Navigatore Meccanico

8 Selezionare le caratteristica Flangia_ 1 e Forma_1 nel nodo **Corpi**, quindi fare cliccare il tasto destro e selezionare **Disabilita** nel menu contestuale.

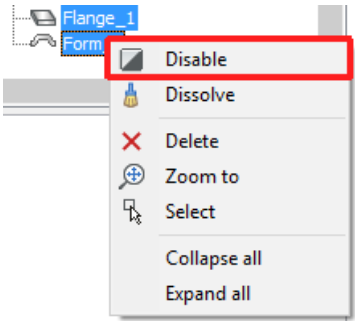
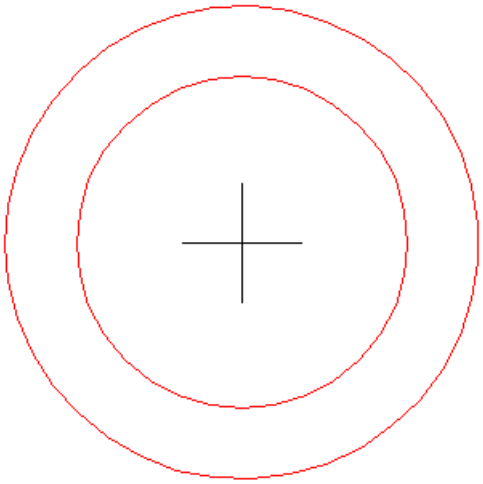


Figura 144: Disabilita

9 Creare 2 nuovi layer: UNFOLD_SYMBOL_UP e UNFOLD_SYMBOL_DOWN.

10 Spegnere il layer 0 e UNFOLD_SYMBOL_DOWN e impostare come **Corrente** il layer UNFOLD_SYMBOL_UP.

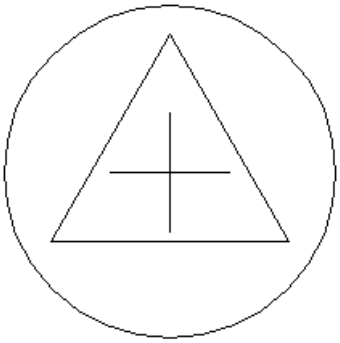
11 Creare il simbolo per lo spiegamento che verrà posizionato sulla faccia opposta a quella in cui è



posizionata la caratteristica forma.

12 Spegnere il layer UNFOLD_SYMBOL_UP e accendere il layer UNFOLD_SYMBOL_DOWN. Impostare il layer UNFOLD_SYMBOL_DOWN come **Corrente**.

13 Creare il simbolo per lo spiegamento che verrà posizionato sulla faccia opposta a quella su cui è



posizionata la caratteristica forma.

14 Accendere il layer 0 e spegnere i layers UNFOLD_SYMBOL_UP e UNFOLD_SYMBOL_DOWN. Impostare

il layer 0 come **Corrente**.

15 Salvare il disegno.

21.5.19.5 Creare una caratteristica forma parametrica personalizzata

Ora parametrizzeremo la caratteristica forma personalizzata che abbiamo creato con la precedente procedura.

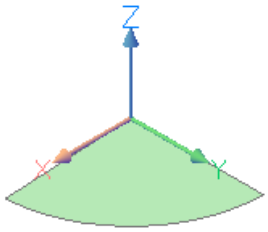
Sono necessari 2 parametri:

- **D** per il diametro dello spigolo maggiore del foro conico.
- **A** per l'angolo della faccia del foro conico.

Nella Lamiera, il parametro predefinito **_SM_THICKNESS** controlla lo spessore della parte in lamiera.

1 Creare un corpo ausiliario per il vincolo su un nuovo layer CONSTRUCTION.

Rendere corrente questo layer e disattivare tutti gli altri layer.



2 Creare una Regione sul layer CONSTRUCTION nel piano XY del WCS.

3 Applicare un vincolo *DMFISSO3D* ( ) alla regione.

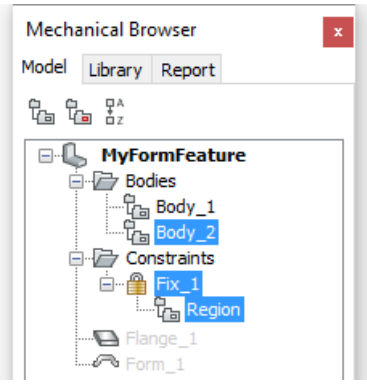
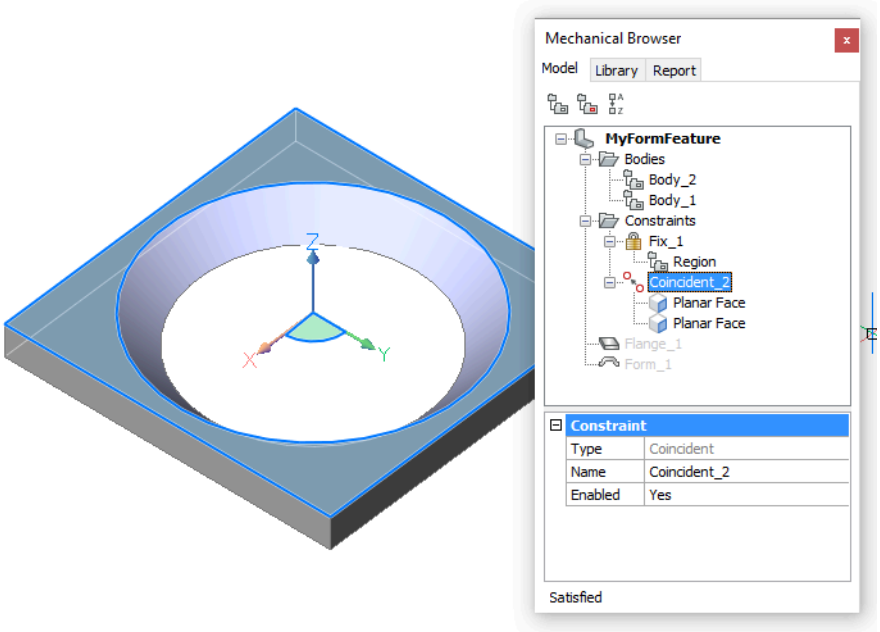


Figura 145: Navigatore Meccanico

4 Accendere il layer 0.

5 Applicare un vincolo *DMCOINCIDENTE3D* ( ) tra la faccia della regione e la faccia superiore della



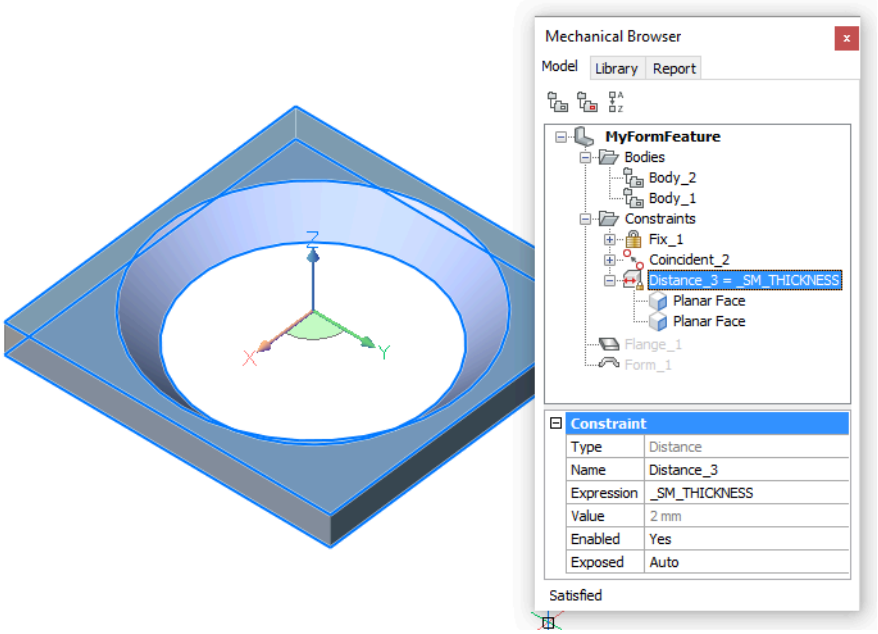
flangia.

6 Applicare un vincolo *DMDISTANZA3D* ( ) tra la faccia superiore e inferiore della flangia.

Selezionare innanzitutto la faccia superiore, quindi premere il tasto **TAB** per selezionare la faccia inferiore.

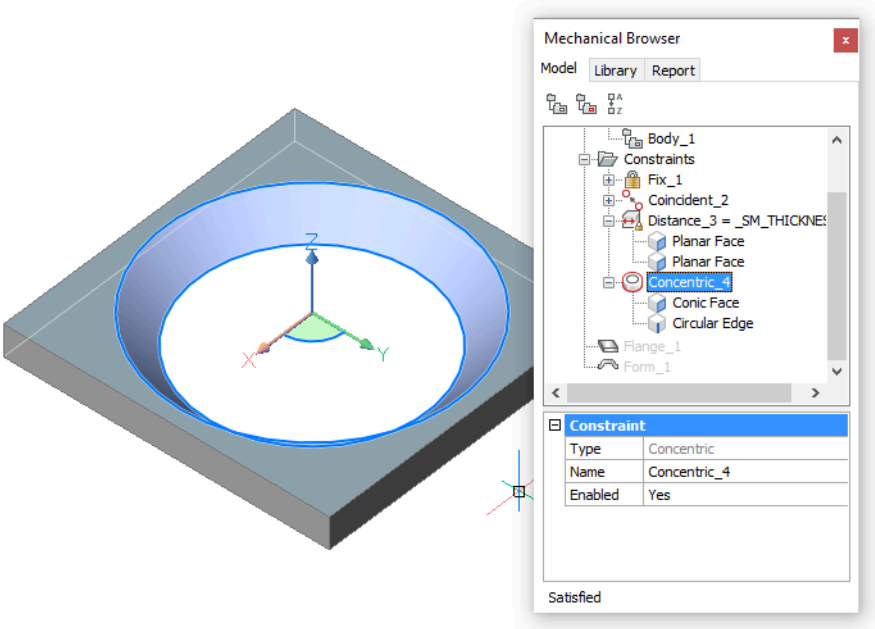
Accettare il valore predefinito nella dimensione dinamica.

7 Digitare _SM_THICKNESS nel campo **Espressione** del vincolo



Distanza_3.

8 Poiché la nostra Caratteristica Forma è conica, possiamo modificare la flangia da parallelepipedo a



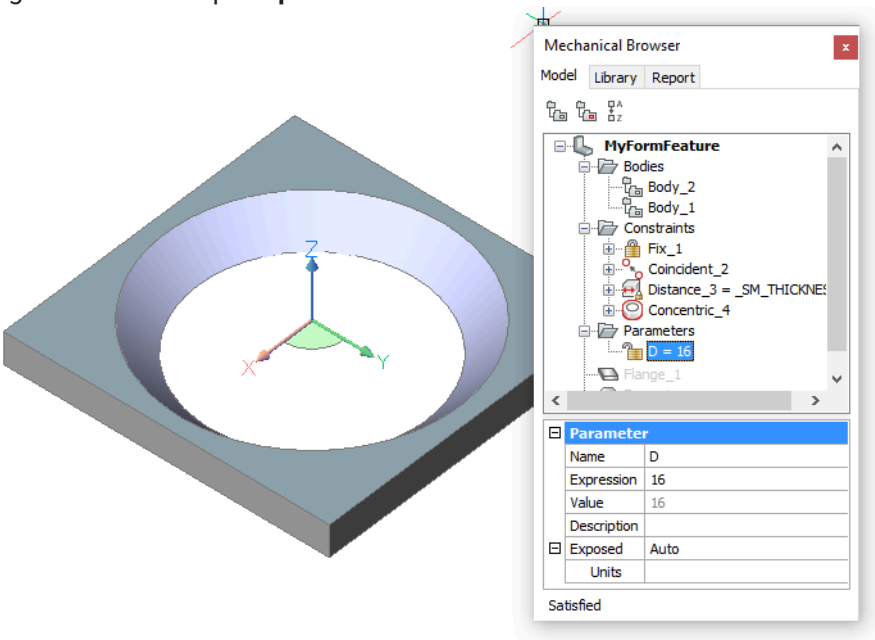
cilindro.

9 Nel **Navigatore Meccanico** cliccare il pulsante destro del mouse sul nome della caratteristica e selezionare **Aggiungi nuovo parametro** nel menu di scelta rapida.

Rinominare il nuovo parametro in **D**.

Selezionare mm nel campo **Unità**.

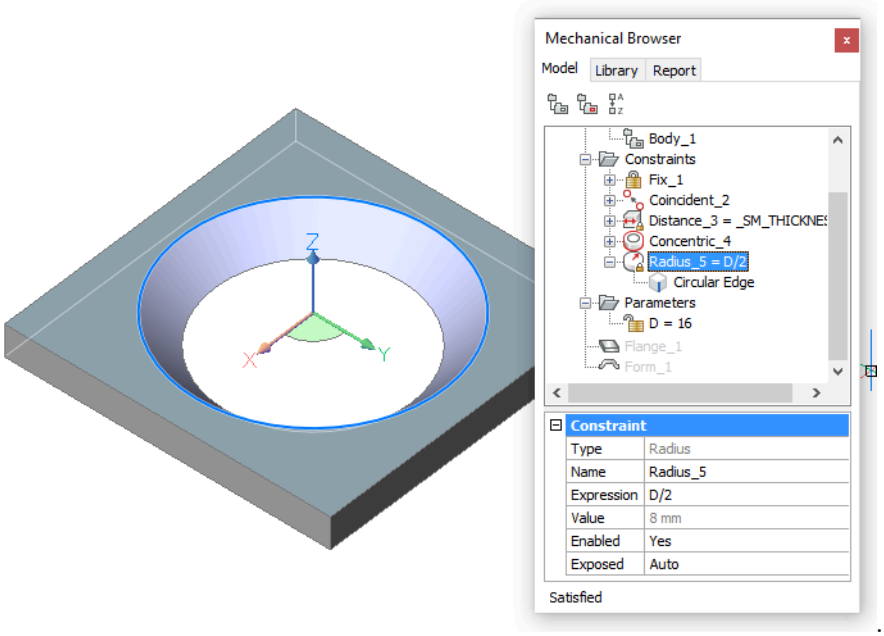
Digitare 16 nel campo **Espressione**.



10 Applicare il vincolo *DMRAGGI3D* ( ) allo spigolo superiore della faccia conica.

Accettare il valore predefinito nella dimensione dinamica.

11 Digitare D/2 nel campo **Espressione** del vincolo



Raggio_5 .

12 Nel **Navigatore Meccanico** cliccare il pulsante destro del mouse sul nome della caratteristica e selezionare **Aggiungi nuovo parametro** nel menu di scelta rapida.

Rinominare il nuovo parametro in **A**.
Lasciare vuoto il campo **Unità**.

Digitare 45 nel campo **Espressione**.

13 Effettuare una delle seguenti operazioni per creare un **Vincolo Angolo del Cono** sulla faccia conica:

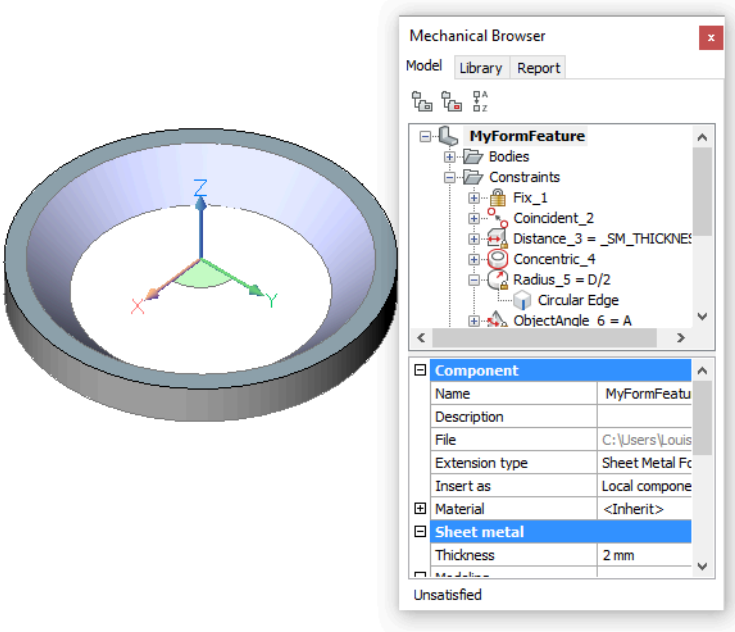
- Avviare il comando *DMANGOLO3D* (), quindi selezionare l'opzione **Vincolo Angolo del Cono**, quindi selezionare la faccia conica.
- Passare con il mouse sopra la faccia conica, quindi selezionare **Vincolo Angolo del Cono** () nel gruppo di comandi **Vincoli** nel **Quad**.

Accettare il valore predefinito.

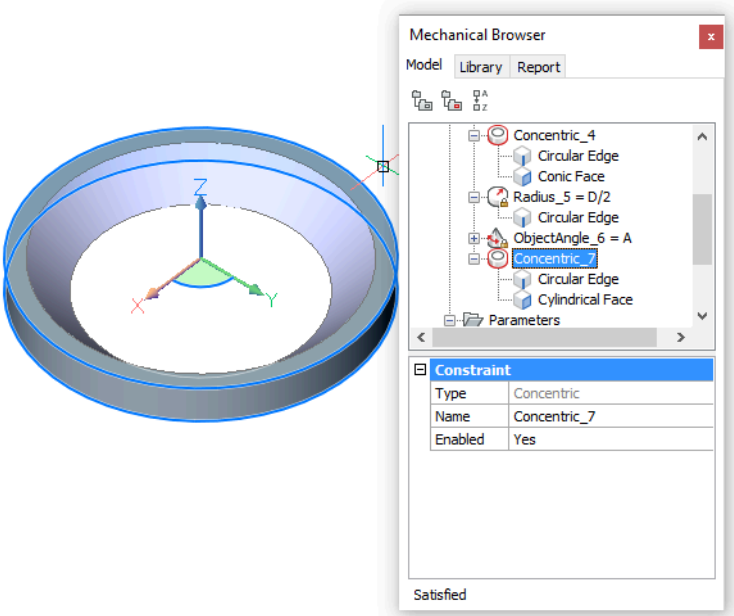
14 Digitare A nel campo **Espressione** del vincolo AngoloPiano_6.

15 Poiché la nostra caratteristica Forma è conica, modificare la flangia da parallelepipedo a cilindro:

- Disegnare un cerchio sopra la flangia. Il raggio del cerchio è di 9 mm.
- Selezionare **Attiva Rilevamento Contorno** nel pannello **Impostazioni** nella scheda **Inizio** della barra multifunzione.
- Posizionare il cursore sull'area tra il cerchio e lo spigolo quadrato della flangia, quindi selezionare **Estrudi** ( ) nel **Quad** e spostare il cursore verso il basso.



16 Applicare il vincolo *DMCONCENTRICO3D* ( ) tra lo spigolo circolare della regione e la faccia

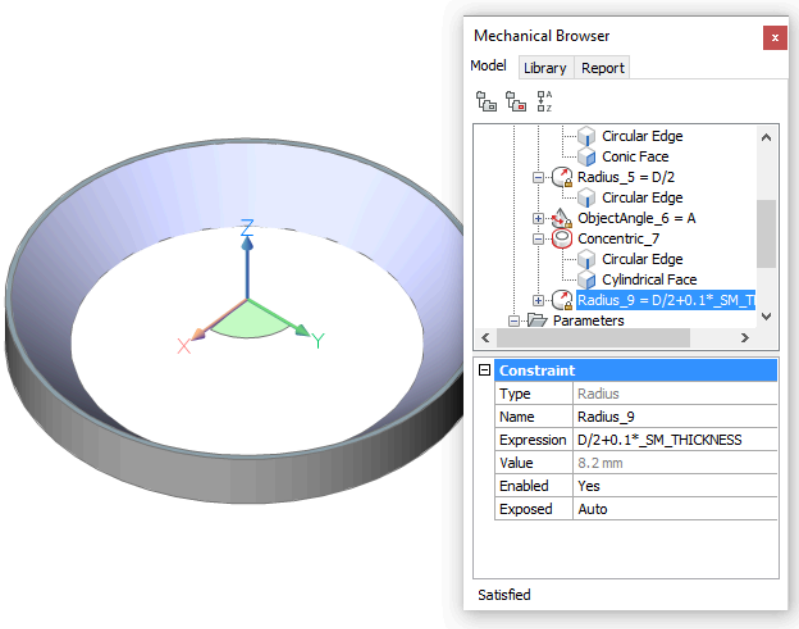


cilindrica.

17 Applicare il vincolo *DMRAGGI3D* ( ) alla faccia cilindrica.

Accettare il valore predefinito nella dimensione dinamica.

18 Digitare D/2+0.1*_SM_THICKNESS nel campo **Espressione** del vincolo



Raggio_8.

19 Impostare su **On** il campo **Esposto** di entrambi i parametri A e D.

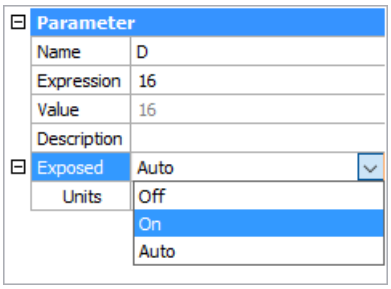


Figura 146: Parametro

Ora creare vincoli 2D per i simboli della spiegatura.

20 Rendere **Corrente** il layer UNFOLD_SYMBOL_UP e **Disattivare** tutti gli altri layer.

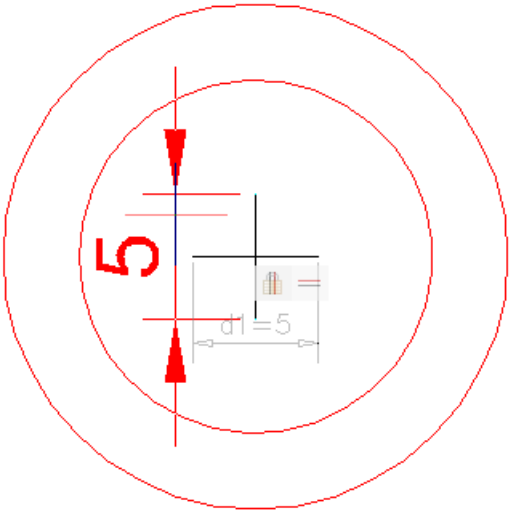
21 Applicare un vincolo *VGFISSO* ( ) ad entrambe le linee che indicano il centro dei cerchi.

22 Applicare il vincolo *VGORIZZONTALE* ( ) alla linea centrale orizzontale.

23 Applica il vincolo *VGVERTICALE* ( ) alla linea centrale verticale.

24 Applicare il vincolo *VGCONCENTRICO* ( ) tra i 2 cerchi.

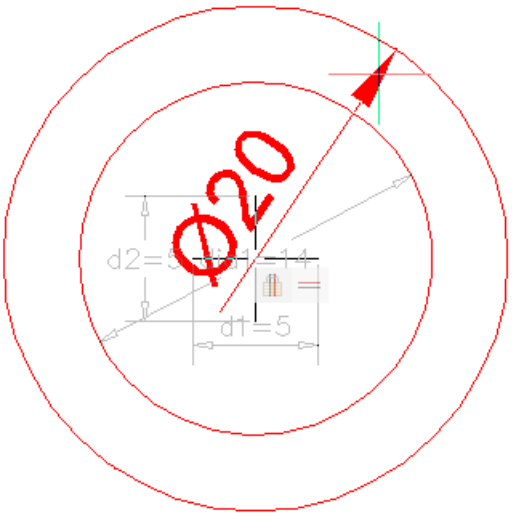
25 Applicare il vincolo *VDLINEARE* ( ) tra i punti finali delle linee centrali. Accettare il valore



predefinito.

26 Applicare il vincolo *VDDIAMETRICO* ( ) a entrambi i cerchi.

Accettare il valore predefinito per il dia1 (=cerchio interno) e dia2 (= cerchio esterno).



27 Regolare i campi **Espressione** dei vincoli 2D come segue:

- d1 = 1
- d2 = d1
- dia 1 = D - 2*_SM_THICKNESS/tan(A)
- dia 2 = D

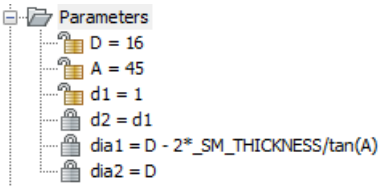
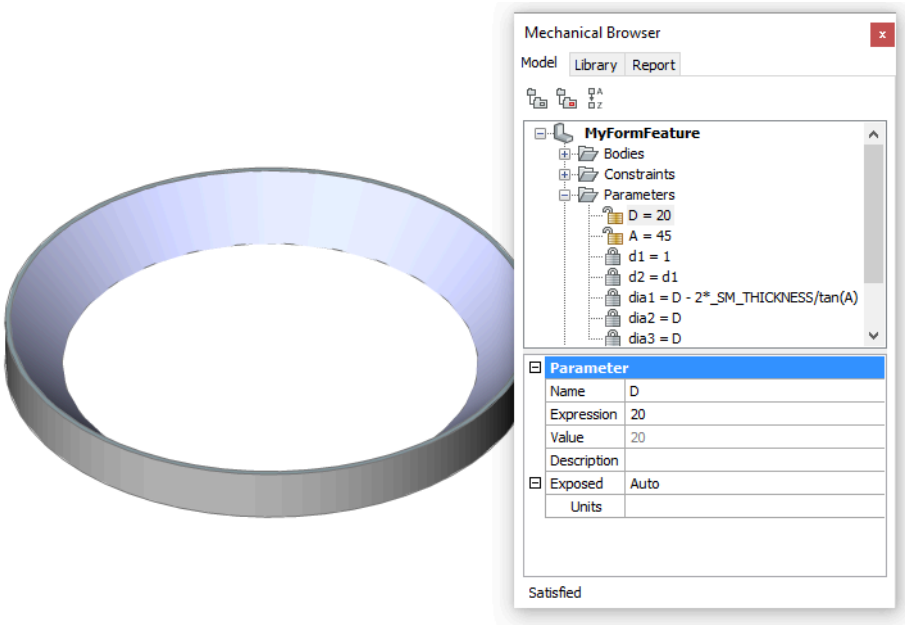


Figura 147: Parametri

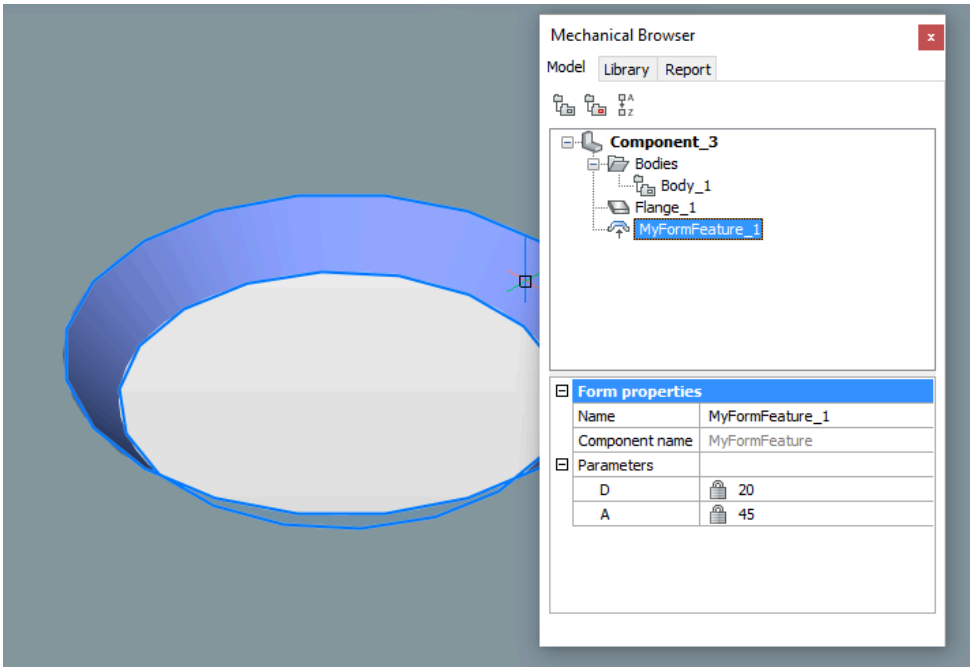
28 **Disattivare** la visualizzazione di tutti i layer, tranne il layer 0, quindi salvare il



file.

Ora è possibile utilizzare la caratteristica forma per creare aperture parametriche coniche in una flangia della lamiera:

Utilizzare lo strumento **Inserisci Caratteristica Forma** (), per avviare il comando *BMINSERT*.



Facoltativamente: Aggiungere la caratteristica forma come componente su una tavolozza degli strumenti.

21.5.19.6 Creare una caratteristica nervatura

Le caratteristiche Nervatura vengono create con il comando *SMRIBCREATE* da una curva 2D sulla faccia della flangia.

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante **Crea Caratteristica Nervatura** ( ) sulla barra degli strumenti **Lamiera**.
   - Cliccare sul pulsante dello strumento **Crea Caratteristica Nervatura** ( ) sul pannello **Modifica** della barra multifunzionale **Lamiera**.
   - Digitare *SMRIBCREATE* nella barra dei comandi, quindi premere **Invio**.

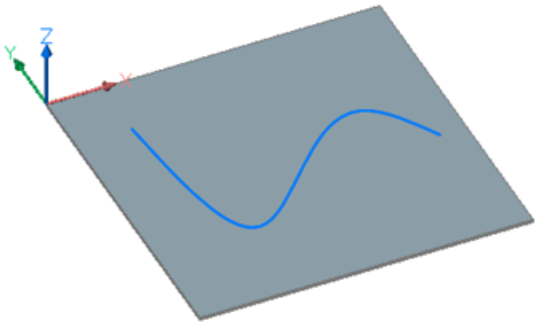
     Viene richiesto: Selezionare faccia della flangia [opzioni di selezione (?)]:

2 Selezionare la faccia della flangia.

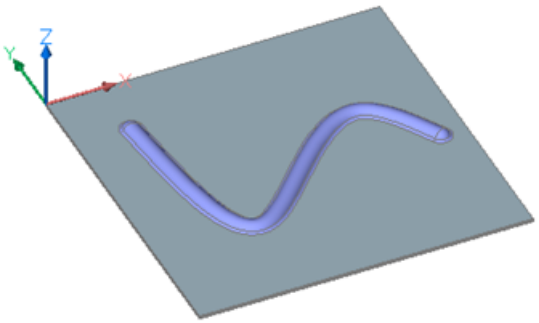
   Viene richiesto: Selezionare curva di controllo [raggio del Profilo/Raggio di raccordo/opzioni di selezione (?)]:

3 Selezionare una curva sulla faccia della flangia selezionata.

   Linee, polilinee, cerchi, archi, ellissi, archi ellittici e spline sono accettati. Una curva 2D può essere aperta o chiusa, ma non può auto-intersecarsi o intersecarsi con altre caratteristiche forma.



   Dopo aver selezionato la curva, viene creata una caratteristica nervatura:



21.5.19.7 Modificare una caratteristica nervatura

Modificare una caratteristica nervatura

- Modificare la curva generatrice usando i suoi grips.

- Modificare il **Profilo** e il raggio di **Raccordo** nel **Navigatore Meccanico**.

Modificare la curva generatrice

1 Selezionare la curva.

Poiché la curva è nascosta dalla caratteristica nervatura, premi il tasto **TAB** per selezionare entità oscurate o seleziona la curva nel pannello **Struttura**.

2 Modificare la curva mediante grip.

La caratteristica nervatura si aggiorna in base alla modifica della curva
.

3 Dopo aver selezionato la faccia della flangia, il comando chiede:

Selezionare curva di controllo [raggio del Profilo/Raggio di raccordo/opzioni di selezione (?)]:

4 Seleziona la curva sul lato della flangia selezionato.

Modificare le proprietà nel Navigatore Meccanico

1 Selezionare la caratteristica nervatura nel **Navigatore Meccanico**.

La caratteristica nervatura si evidenzia nel modello.

2 Selezionare la proprietà nella griglia **Proprietà Nervatura**.

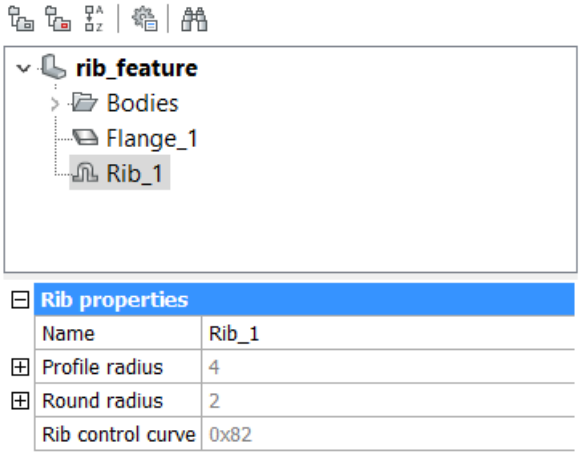


Figura 148: Proprietà nervatura

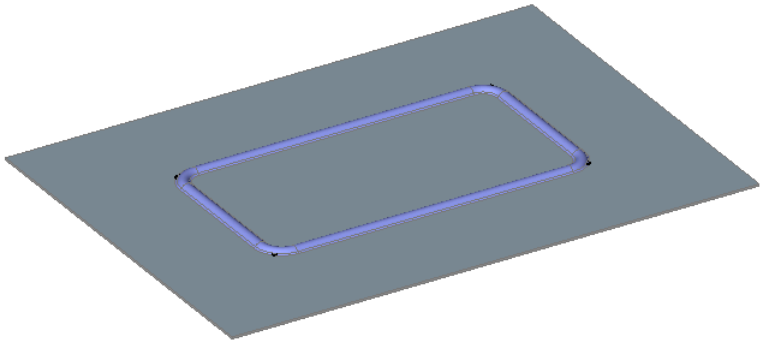
3 Digitare un valore.

La caratteristica nervatura selezionata viene aggiornata.

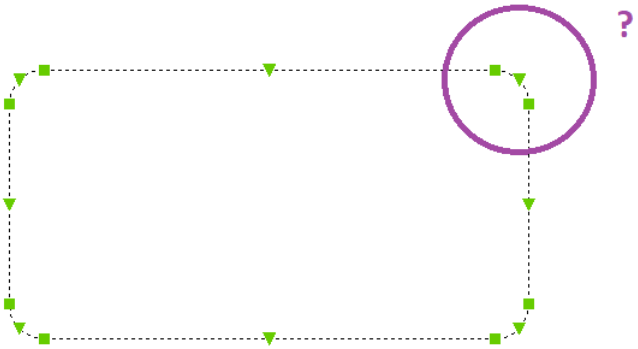
21.5.19.8 Comprendere la differenza tra le curve di Controllo e di Traiettoria

Quando creiamo una caratteristica nervatura rettangolare, disegniamo prima un rettangolo che viene poi utilizzato per creare una caratteristica nervatura mediante il comando *SMRIBCREATE*. Quando questa curva di controllo viene modificata, la caratteristica nervatura viene aggiornata di conseguenza.

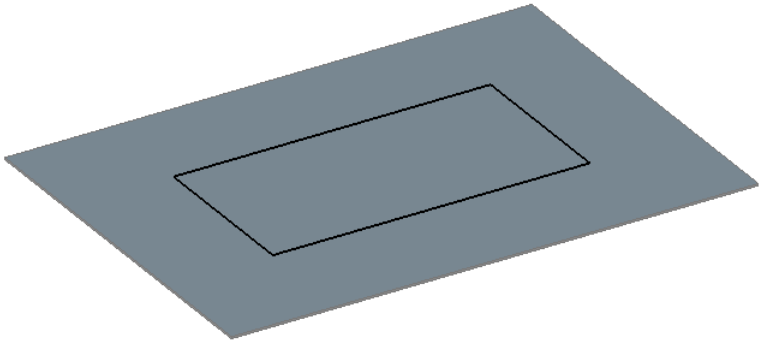
Dal punto di vista della produzione, la curva di controllo rettangolare è sostituita da una curva di traiettoria raccordata, in base ai limiti fisici dello strumento ruota:



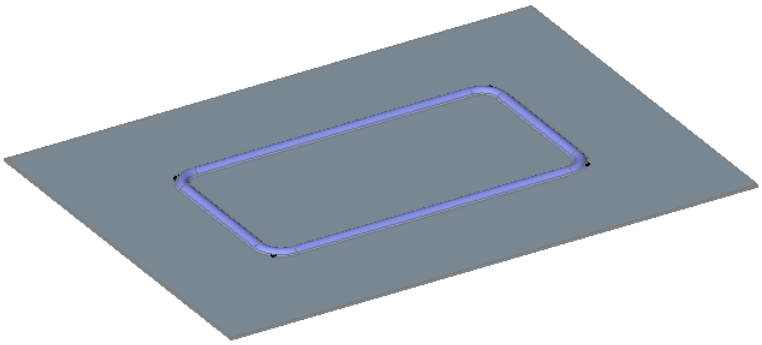
La curva traiettoria, comunque corretta, è difficile da manipolare, ad esempio quando si desidera spostare un angolo.



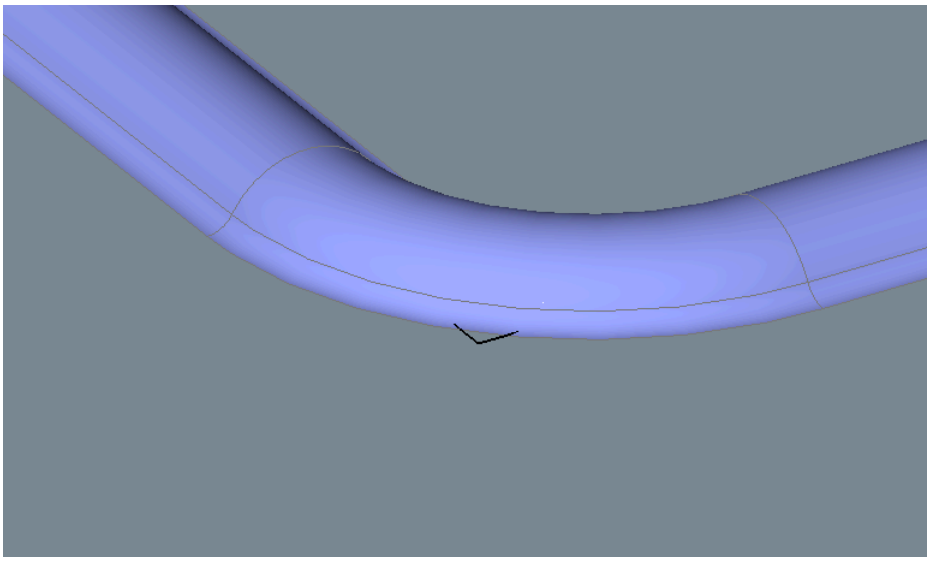
Nella fase di progettazione utilizziamo la **Curva di Controllo**:



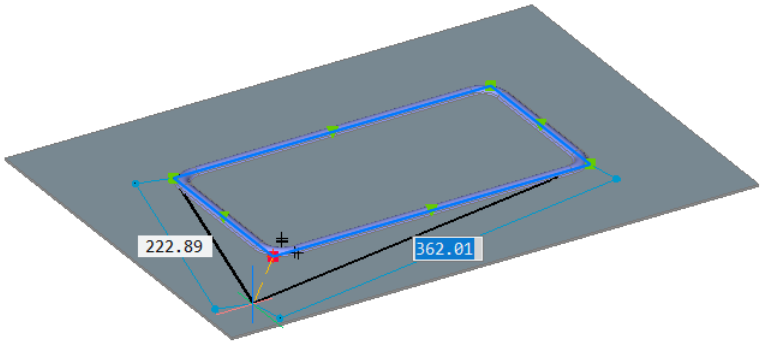
Durante la creazione della caratteristica nervatura, *SMRIBCREATE* aggiunge i raccordi in base al valore del parametro **Raggio di raccordo della nervatura** nel **Navigatore Meccanico**. Questo parametro non è controllato dalla singola caratteristica, in quanto si riferisce alle proprietà dello strumento fisico:



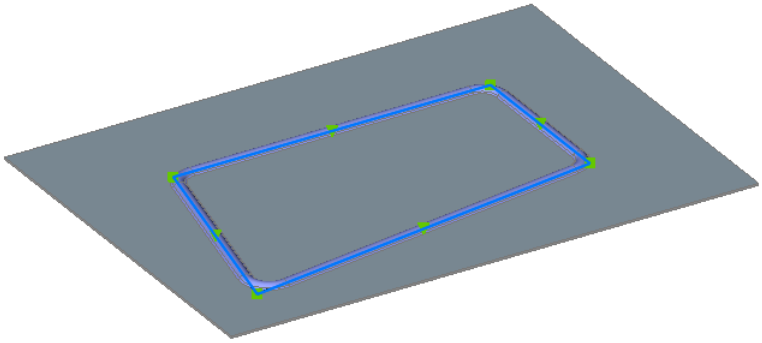
Quando si esegue uno zoom ravvicinato, si nota la differenza tra la geometria 3D della caratteristica nervatura e la curva di controllo:



Quando la Polilinea è in fase di modifica con grip...

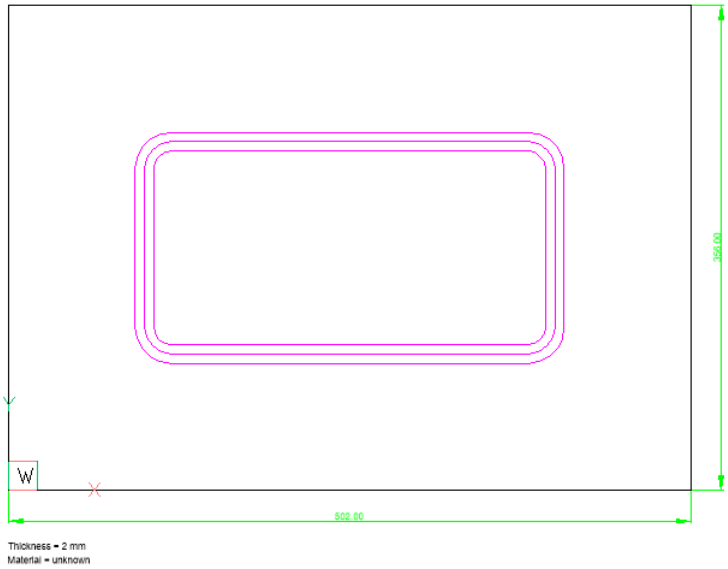


... la caratteristica nervatura viene regolata di conseguenza:



Questo illustra l'associatività tra la curva 2D e la caratteristica nervatura 3D. Il riconoscimento delle caratteristiche delle nervature in *SMCONVERT* è controllato dalle preferenze dell'utente *SmConvertRecognizeRibControlCurves*.

Quando la parte viene spiegata dal comando *SMUNFOLD*, le curve seguono il progetto 3D e dispongono possiedono i raccordi. La curva centrale è chiamata **Curva Traiettoria**:

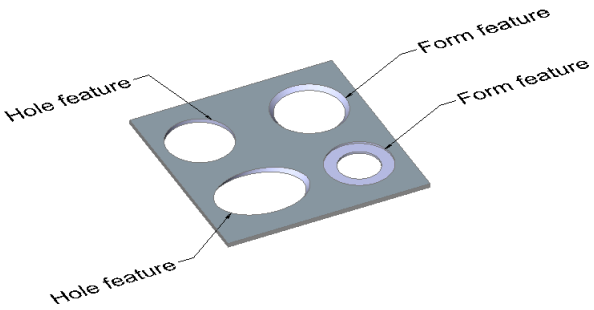


21.5.20 Lavorare con le Caratteristiche Foro

21.5.20.1 Panoramica

Le caratteristiche foro sono una versione semplificata delle Caratteristiche Forma. Nell'immagine seguente è possibile visualizzare caratteristiche diverse, che hanno solo facce spesse (il comando *SMCONVERT* rispetta queste regole):

- I fori ortogonali sono caratteristiche foro.
- Semplici fori non ortogonali sono caratteristiche foro
- I fori ortogonali smussati sono caratteristiche forma
- Se la faccia dello spessore del foro è composta da 2 diverse regioni, è una caratteristica forma.



Le caratteristiche foro vengono riconosciute dal comando *SMCONVERT* se la preferenza utente *SMCONVERTRECOGNIZEHOLES* è ON.

Con le caratteristiche foro sono consentite le seguenti operazioni di base:

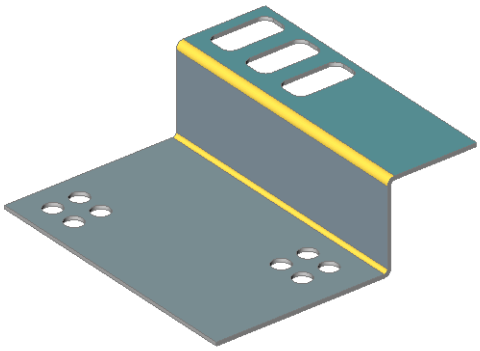
- *SMDELETE*, *SMDISSOLVE*, disabilita e *SMSELECT* utilizzando i relativi comandi o nel **Navigatore Meccanico per la Lamiera**.
- **Modalità Caratteristica Forma** nel comando *SMUNFOLD*, inclusa la modalità **Simbolo**.
- Associare la tua geometria 2D in un layout spiegato.
- Le caratteristiche foro sono mantenute dal comando *SMREPAIR*. Le facce dei fori semplici diventano ortogonali. Una caratteristica foro della stessa geometria non è influenzata dal comando *SMREPAIR*.

21.5.20.2 Riconoscimento delle Serie di Caratteristiche Foro

Quando esistono caratteristiche foro in un corpo, il comando *SMPARAMETRIZE* rileva serie rettangolari di fori sulle flange, secondo le seguenti regole:

- I fori appartengono alla stessa flangia.
- i fori si trovano su una griglia rettangolare
- Non ci sono spazi vuoti (elementi mancanti) nella matrice
- I fori sono orientati allo stesso modo
- La dimensione minima della serie è 1x3 o 2x2

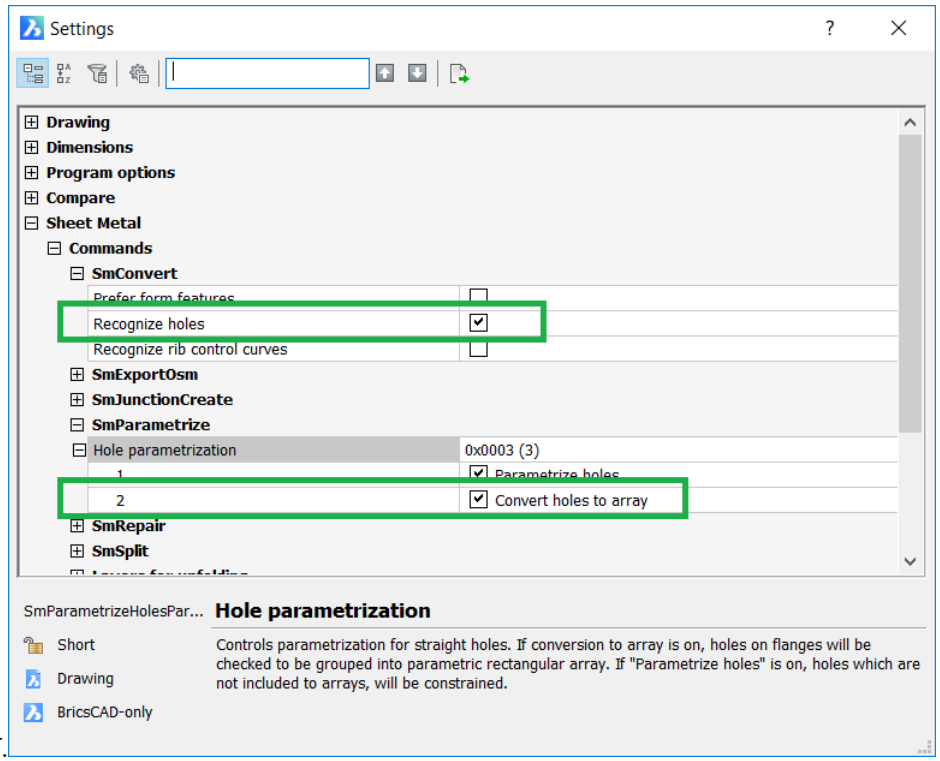
Consentici di illustrare il flusso di lavoro su una parte con le caratteristiche Flangia e Piegatura.



1 Nella finestra di dialogo *Impostazioni*, andare su **Lamiera/Comandi**.

2 Abilitare l'opzione **Riconosci fori** per il comando *SMCONVERT*.

3 Abilitare le opzioni **Converti fori in serie** e **Parametrizza fori** per il comando



SMPARAMETRIZE.

4 Eseguire il comando *SMCONVERT*.

5 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Converti in Lamiera** ( ) della barra degli strumenti **Lamiera**.
- Fare clic sul pulsante strumento **Converti in Lamiera** ( ) della barra multifunzione **Lamiera**.
- Scegliere **Converti in Lamiera** nel menu **Lamiera**.

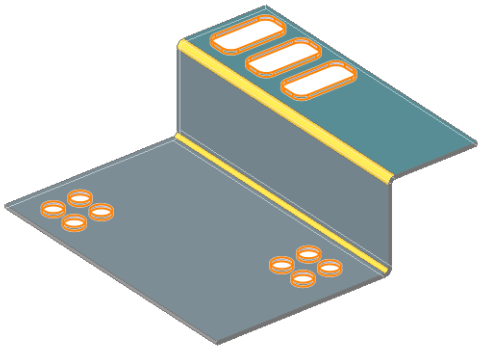
  Viene richiesto: Selezionare solidi 3D o facce seme o [Modello intero/opzioni di selezione (?)] <Intero modello>:

6 Selezionare il solido 3D da convertire.

Viene richiesto: Selezionare solidi 3D o facce seme o [Modello intero/opzioni di selezione (?)]:

7 Premere **Invio** per convertire il solido 3D in una parte di lamiera.

Sulla parte sono stati riconosciuti 11 fori.



8 Eseguire il comando *SMPARAMETRIZE*.

9 Eseguire una delle seguenti operazioni:

- Cliccare sul pulsante dello strumento **Parametrizza** ( ) sulla barra degli strumenti **Lamiera**.
- Cliccare sul pulsante dello strumento **Parametrizza** ( ) sulla barra multifunzione **Lamiera**.
- Scegliere **Parametrizza** nel menu **Lamiera**.

  Viene richiesto: SSelezionare solidi 3D da parametrizzare o [intero Modello/opzioni di selezione (?)] <Intero modello>:

10 Premere **Invio** per elaborare la parte in lamiera.

I report dei comandi devono essere simili ai seguenti:

Creato 4 vincoli di distanza

Creato 3 vincoli fisso

Creato 0 vincoli coincidenti

Creato 0 vincoli tangente

Creato 0 vincoli gruppo rigido

Creato matrice rettangolare 2x2 basata sulla caratteristica Hole_3

Creato matrice rettangolare 2x2 basata sulla caratteristica Hole_7

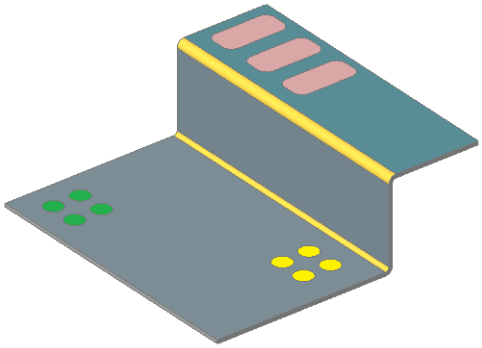
Creata matrice rettangolare 3x1 basata sulla caratteristica Hole_9

Totale: 7 vincoli e 3 serie create

Nota: I nomi delle caratteristiche e la distribuzione dei vincoli possono variare a seconda della versione di BricsCAD.

Per esaminare le serie è possibile sbloccare temporaneamente il layer **BC_SUBTRACT**.

Nell'immagine sottostante le serie sono state colorate manualmente.

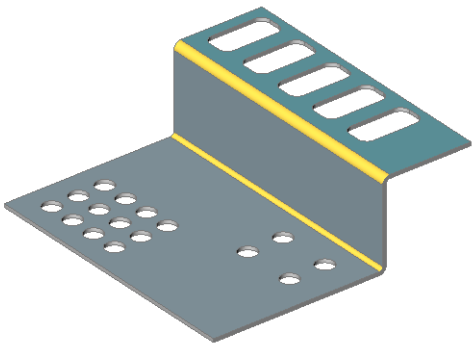


Nota: Le serie verdi e gialle non sono state unificate. Altrimenti la serie avrebbe dei fori mancanti.

11 Modificare le **Proprietà della Serie** nel **Navigatore Meccanico**.

Nell'immagine sottostante è stato modificato quanto segue:

- Serie verde: numero di righe e colonne da 2x2 a 3x4
- Serie gialla: spaziatura X e Y.
- Serie rossa: numero di righe e colonne da 3x1 a 5x1.



21.5.21 Convertire in Lamiera e Riparare

Usare i comandi *SMCONVERT* e *SMREPAIR*.

21.5.21.1 Convertire la geometria 3D in una valida parte di Lamiera

È possibile creare una parte di lamiera da qualsiasi geometria solida 3D esistente, creata in BricsCAD od importata da un altro sistema CAD. Tuttavia, la geometria potrebbe essere errata e quindi non congruente con un modello di lamiera: piegature e scarichi mancanti, o facce dello spessore che non sono ortogonali alle facce della flangia. BricsCAD Sheet Metal contiene un gruppo di strumenti dedicati per convertire tale geometria in una valida parte di lamiera ed aggiungere tutte le caratteristiche della lamiera mancanti.

1 Eseguire una delle seguenti operazioni:
   - Cliccare sul pulsante dello strumento **Converti in Lamiera** ( ) della barra degli strumenti **Lamiera**.
   - Fare clic sul pulsante strumento **Converti in Lamiera** ( ) della barra multifunzione **Lamiera**.
   - Scegliere **Converti in Lamiera** nel menu **Lamiera**.

     Viene richiesto: Selezionare solidi 3D o facce seme o [Modello intero/opzioni di selezione (?)] <Intero modello>:
2 Selezionare il solido 3D da convertire.

   Viene richiesto: Selezionare solidi 3D o facce seme o [Modello intero/opzioni di selezione (?)]:
3 Premere **Invio** per convertire il solido 3D in una parte in lamiera.

Le proprietà della parte in lamiera vengono visualizzate nel **Navigatore Meccanico** della Lamiera.

Nota: I corpi convertiti possono essere spiegati o esportati in formati di file .DXF o .OSM per l'elaborazione da parte di un sistema CAM.

In alcuni casi potrebbe essere necessario correggere i corpi convertiti prima dello spiegamento/esportazione:

- Creare gli scarichi mancanti: comando *SMRELIEFCREATE*.
- Convertire gli spigoli vivi in in piegature o giunzioni: comandi *SMBENDCREATE* e *SMJUNCTIONCREATE*.
- Dividere flange e piegature con lofted: comando *SMSPLIT*.
- Riparare i corpi in lamiera: comando *SMREPAIR*.

21.5.21.2 Riparare una parte di Lamiera

Una parte in lamiera può presentare imprecisioni nella geometria e nella topologia o problemi di progettazione della lamiera. Questo capita spesso con parti importate progettate con kernel geometrici diversi da ACIS.

Esempi di tali problemi sono:

- Facce di spessore, che non sono perpendicolari alle facce della flangia/piegatura.
- Le piegature loft che non sono tangenti alle flange/piegature adiacenti.
- Piegature errate.
- Problemi generali ACIS.

Tali problemi possono impedire l'esecuzione corretta delle operazioni di modifica.

Il flusso di lavoro consigliato per risolvere tali problemi è:

- Eseguire il comando *DMAUDIT* per risolvere i problemi generali di modellazione.
- Eseguire il comando *SMREPAIR*.

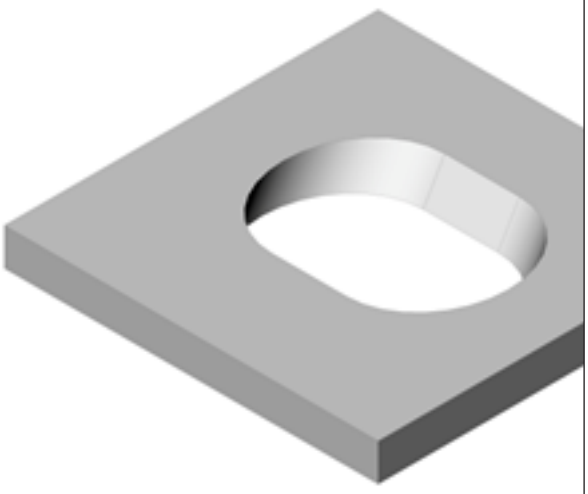
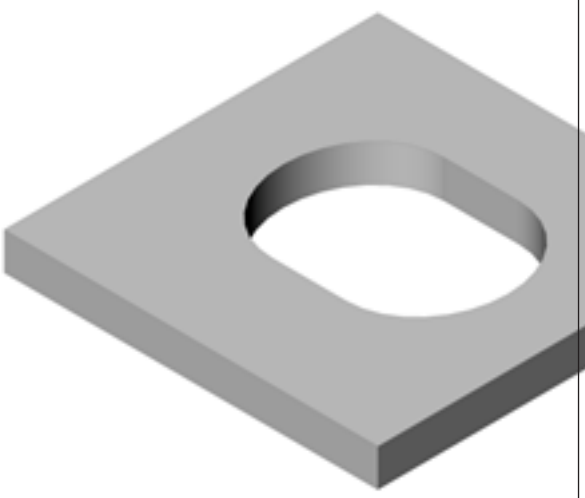
21.5.21.3 Riparare un corpo di Lamiera

I corpi di lamiera con facce di spessore, che non sono perpendicolari alle facce della flangia/piegatura, non possono essere prodotti correttamente utilizzando i macchinari tradizionali.

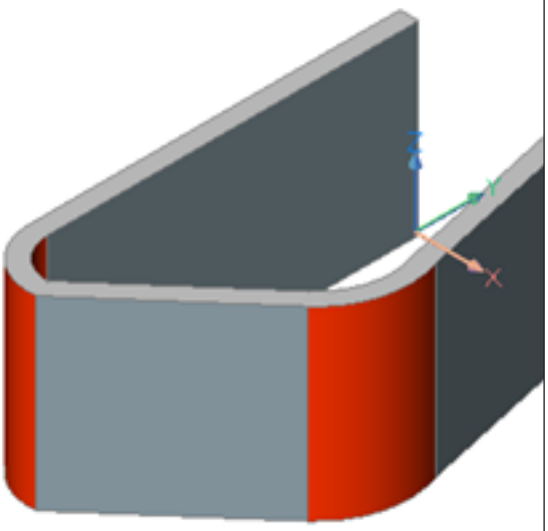
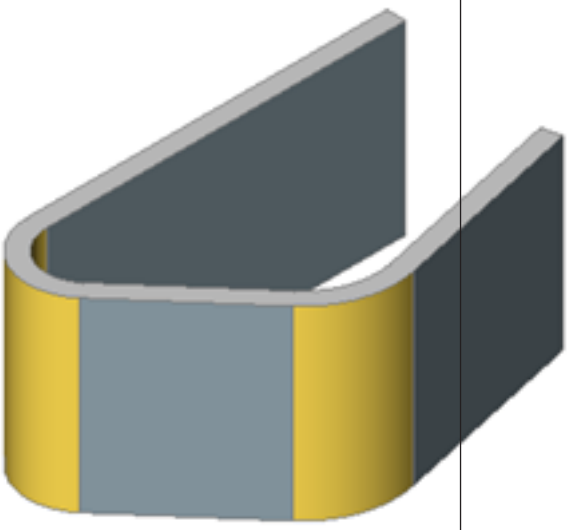
1 Eseguire una delle seguenti operazioni:
   - Cliccare su **Ripara Lamiera** ( ) sul pannello Ripara nella barra multifunzione Lamiera.
   - Cliccare su **Ripara Lamiera** ( ) sulla barra degli strumenti **Lamiera**.
   - Scegliere **Ripara Lamiera** nel menu **Lamiera**
   - Digitare *SMREPAIR* nella barra dei comandi.

     Viene richiesto: Selezionare faccia di una flangia o faccia della piegatura con loft da riparare o [passare sulla riparazione Locale/Scansione per dati persi/opzioni di selezione (?)]:

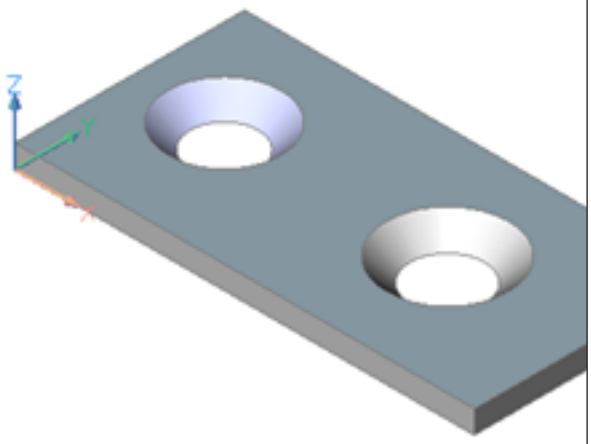
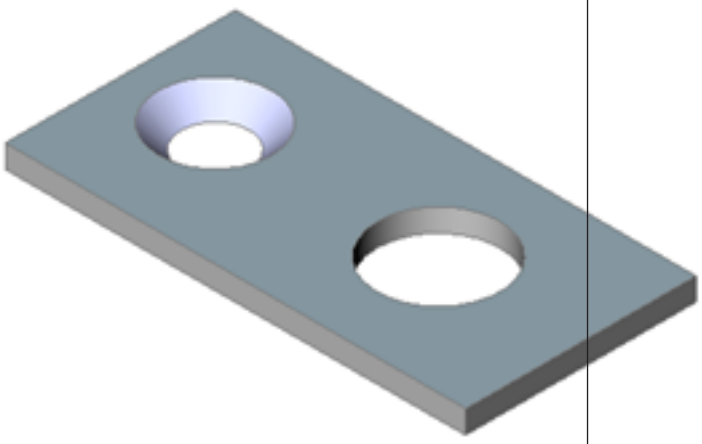
2 Selezionare la faccia da riparare.

	
Lo stesso modello prima...	... e dopo aver applicato un'operazione di riparazione:

Quando il modello presenta piegature errate, *SMREPAIR* le trasforma in piegature corrette:

	
Piegature errate, che hanno una connessione non tangenziale con le flange.	Piegature corrette dopo la riparazione.

Le caratteristiche di forma vengono tradotte nel nuovo modello riparato:

	
Modello prima della riparazione.	La caratteristica di forma non è cambiata mentre il foro, che non è una caratteristica di forma, è stato reso ortogonale.

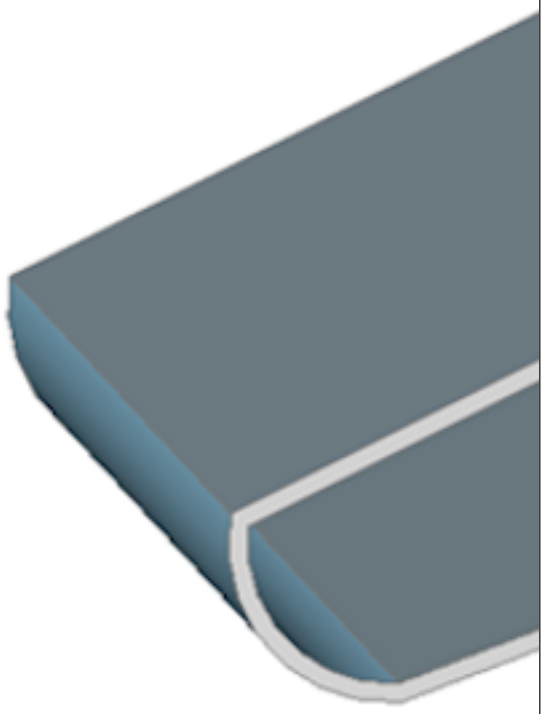
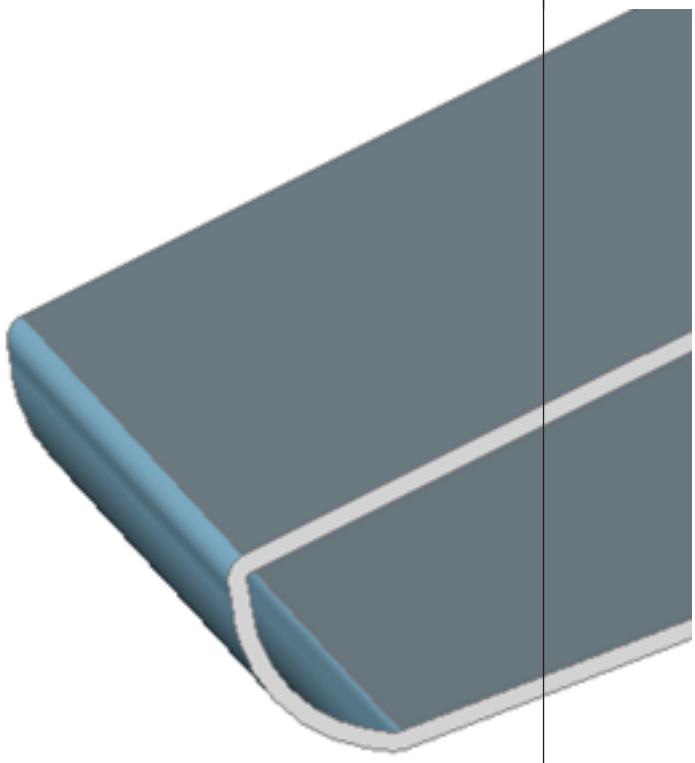
21.5.21.4 Riparare un corpo in Lamiera con piegature lofted

1 Eseguire una delle seguenti operazioni:

- Cliccare su **Ripara Lamiera** ( ) sul pannello **Ripara** nella barra multifunzione **Lamiera**.
- Cliccare su **Ripara Lamiera** ( ) sulla barra degli strumenti **Lamiera**.
- Scegliere **Ripara Lamiera** nel menu **Lamiera**.
- Digitare *SMREPAIR* nella barra dei comandi.

  Viene richiesto: Selezionare faccia di una flangia o faccia della piegatura con loft da riparare o [passare sulla riparazione Locale/Scansione per dati persi/opzioni di selezione (?)]:

2 Selezionare una faccia di flangia o di piegatura loft. La faccia selezionata fornisce la superficie da ispessire.

Prima	Dopo
	
La piegatura con loft non è tangente alla flangia. Lo spiegamento è impossibile.	Notare la differenza nella connessione della piegatura con loft blu con la flangia superiore, che ora è più omogenea. Lo spiegamento è possibile.

Opzione: se la preferenza utente **SmRepairLoftedBendMergela** è ON, le piegature loft adiacenti con una connessione tangenziale vengono unite in un'unica piegatura loft.

21.5.22 Spiegamento delle Parti in Lamiera ed Esportazione in CAM

21.5.22.1 Panoramica

Usare il comando *SMUNFOLD*.

Le parti in lamiera possono essere progettate in molti modi. Per realizzare la parte in lamiera è necessario un modello piatto con annotazioni sulle piegature, che viene successivamente utilizzato in un sistema CAM per programmare macchine CNC, come taglierine laser e piegatrici. Questa operazione di appiattimento viene chiamata "spiegamento" in 2D.

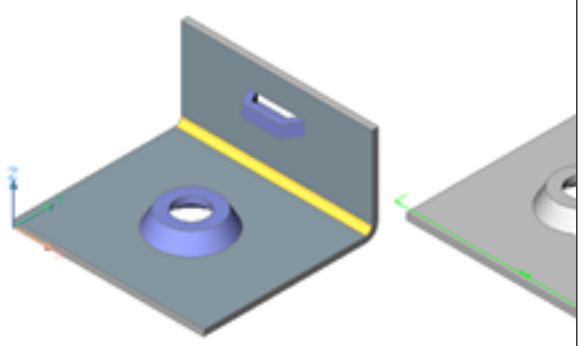
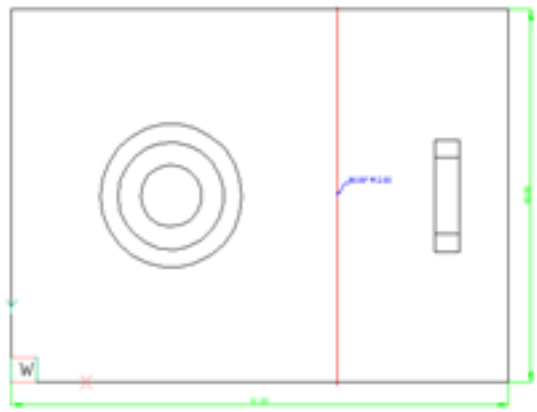
21.5.22.2 Spiegamento di un corpo in lamiera

Il comando *SMUNFOLD* esegue lo spiegamento in 2D delle parti della lamiera. BricsCAD creerà automaticamente un corpo solido corrispondente ad un foglio di lamiera necessario per la fabbricazione della parte di lamiera utilizzando le tecniche di piegatura. Questo foglio è posizionato sul piano XY ed è orientato lungo gli assi delle coordinate allo stesso modo dell'orientamento del corpo iniziale nello spazio. Per modificare l'orientamento dello spiegamento nell'asse Z, eseguire nuovamente il comando *SMUNFOLD* e selezionare la faccia opposta del corpo iniziale.

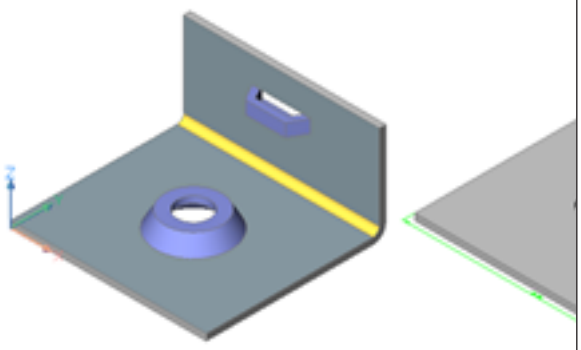
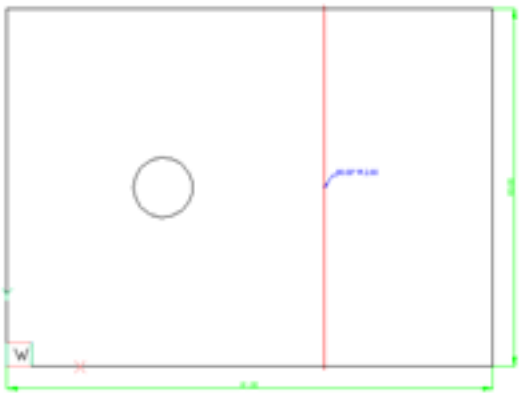
Le caratteristiche forma vengono spiegate in base all'impostazione **Modalità spiegamento della caratteristica forma** nel **Navigatore Meccanico**.

Le opzioni sono:

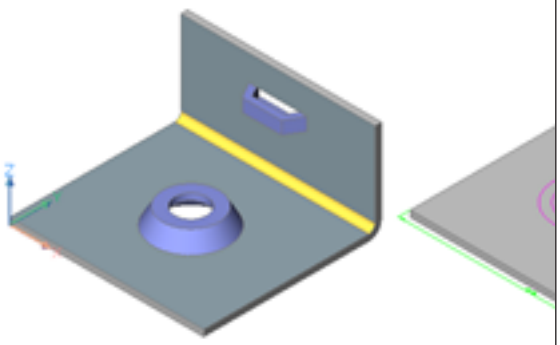
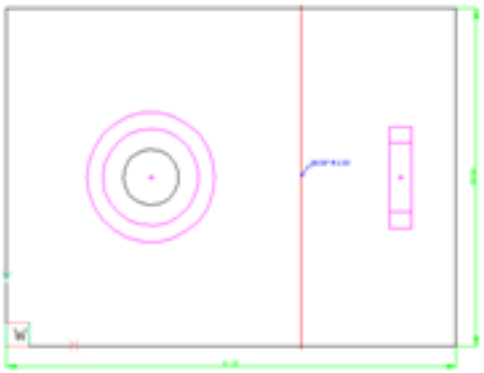
- **Mantieni** (0): consente di mantenere la geometria delle caratteristica forma sulla rappresentazione spiegata in 2D.

Modello 3D e corpo spiegato	2D
	

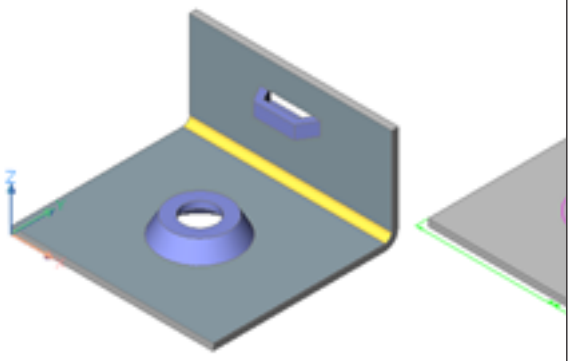
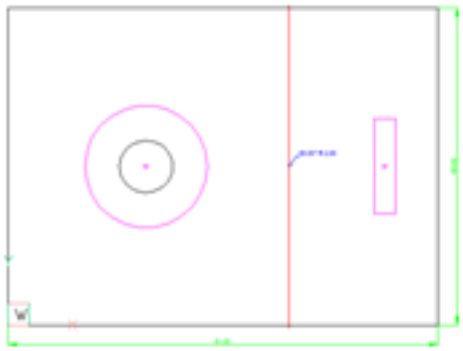
- **Rimuovi** (1): rimuove la geometria della caratteristica forma dalla rappresentazione spiegata in 2D.

Modello 3D e corpo spiegato	2D
	

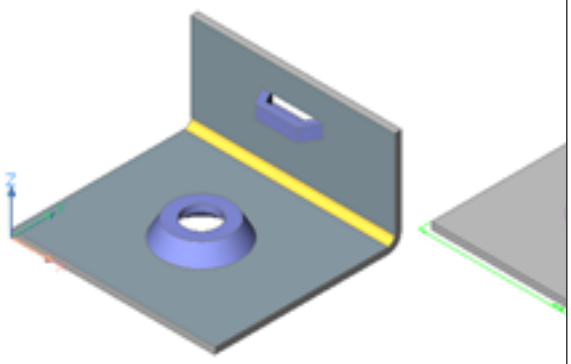
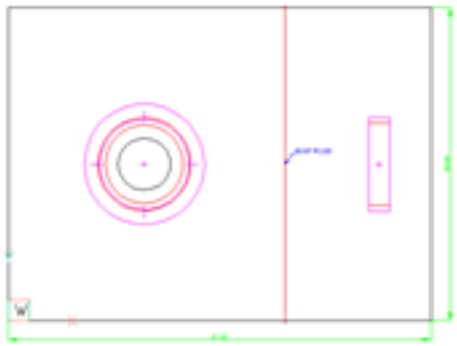
- **Proietta** (2): genera un simbolo di caratteristica forma proiettando gli spigoli delle caratteristiche forma sul piano della flangia.

Modello 3D e corpo spiegato	2D
	

- **Contorno** (3): genera un simbolo di caratteristica forma proiettando un contorno esterno ed i fori della caratteristica forma sul piano della flangia.

Modello 3D e corpo spiegato	2D
	

- **Simbolo** (4): crea il simbolo di dispiegamento sul layer **Caratteristiche di Forma Sopra** o **Caratteristiche di Forma Sotto** a seconda della direzione della caratteristica forma.

Modello 3D e corpo spiegato	2D
	

La modalità di spiegamento 2D per la parte di lamiera corrente può essere impostata nel **Navigatore Meccanico**.

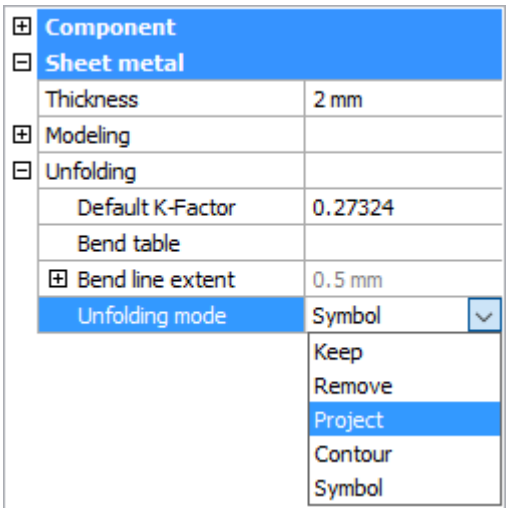


Figura 149: Modalità spiegamento

Le piegature con loft che giacciono su superfici cilindriche e coniche possono essere spiegate analiticamente: le dimensioni e la forma della piegatura con loft dispiegata saranno come se la sua superficie media fosse srotolata matematicamente sul piano. Questa forma può essere ottenuta anche con un numero elevato di campioni per la curva con loft. È necessario impostare la **Modalità di dispiegamento** singolarmente per ogni caratteristica di piegatura con loft. Se lo spiegamento analitico è consentito, l'opzione sarà presente in un elenco a discesa:

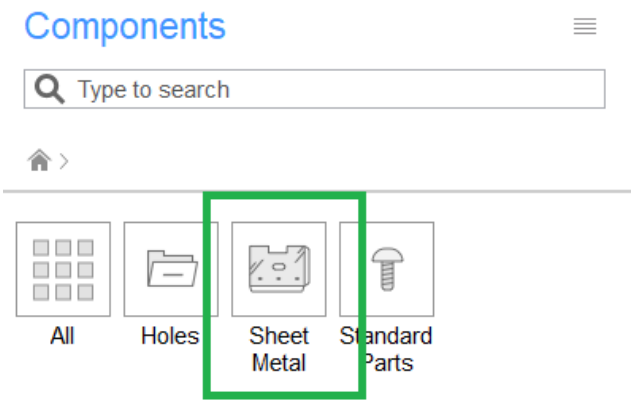


Figura 150: Analitico

21.5.22.3 Spiegare un corpo in lamiera

1 Avviare il comando *SMUNFOLD*.

Viene richiesto: Selezionare una flangia o faccia di una piega con loft per iniziare lo spiegamento [Associativo/opzioni di selezione (?)]:

2 Seleziona una faccia sulla parte in lamiera. Viene richiesto:

Selezionare la posizione del corpo spiegato:

3 Specificare un punto.

Viene richiesto: Convalidare il corpo spiegato e selezionare un'opzione [salva geometria 2D/salva geometria 3D/Posizionare la vista sul layout/Ottimizzare le annotazioni delle piegature/Mantieni] <Mantieni>:

4 Eseguire una delle seguenti operazioni:
   - Salvare il solido 3D dello sviluppo in un file di disegno separato.
   - Convertire il solido 3D spiegato in un file di disegno 2D (*.DWG o *.DXF).
   - Riorganizzare le annotazioni delle piegature.
   - Mantenere il blocco con la parte spiegata nel modello della parte in lamiera.

Il comando *SMUNFOLD* considera la deformazione del materiale della lamiera durante la piegatura. Quando una lamiera viene piegata con un macchinario di piegatura (come ad es. una pressa), il materiale viene deformato plasticamente, ovvero si comprime all'interno della curva e si estende all'esterno di essa. Quindi la lunghezza della parte misurata lungo la sua superficie è diversa negli stati di foglio di lamiera piatto rispetto a piegato. BricsCAD può calcolare automaticamente la lunghezza della parte sulla base delle proprietà di deformazione del materiale. Queste proprietà possono essere definite impostando il valore del parametro **Fattore K** o allegando una Tabella di piegatura.

Esercitazione: Rielaborazione di un modello importato in BricsCAD Sheet Metal

21.5.22.4 Gestione delle proprietà dei layer della geometria 2D

La geometria 2D generata dai comandi *SMUNFOLD*, *SMEXPORT2D* e *SMASSEMBLYEXPORT* possiede colori diversi per contorni, linee di piegatura verso l'alto, linee di piegatura verso il basso, quotatura complessive, caratteristiche forma in alto, caratteristiche forma in basso, annotazioni di piegatura e attributi.

E' possibile impostare i colori in modo diverso nel gruppo impostazioni **Lamiera/Comandi/Layers per spiegamento** della finestra di dialogo *Impostazioni*. Ogni tipo di entità viene inserito su un layer dedicato il cui il colore è controllato attraverso una serie di variabili di sistema:

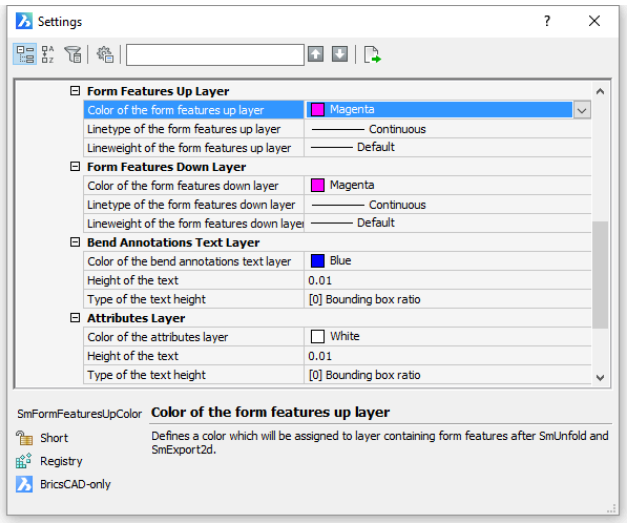


Figura 151: IMpostazioni

Se la geometria 2D di una caratteristica forma è stata presa da un componente della libreria, le sue proprietà sono controllate dalle proprietà di questo componente al momento dell'inserimento. Ci sono altre 2 **Modalità di Spiegamento** le quali portano alla generazione della geometria 2D per le caratteristiche forma: **Progetto** o **Contorno**. In queste modalità la geometria 2D sarà generata automaticamente in geometria 3D e le sue forme saranno controllate dalle impostazioni di gruppo **Layer Carattetistiche di Forma Sopra** e **Layer Carattetistiche di Forma Sotto**.

21.5.22.5 Esportare un corpo in lamiera

Parti di lamiera create o modificate in BricsCAD possono essere elaborate da diversi sistemi CAM (ad esempio JETCAM o CADMAN-B) per generare il codice NC per le tagliatrici e piegatrici.

La preferenza utente controlla l'opzione **Salva geometria 2D** del comando *SMUNFOLD* per personalizzare il file .DXF in modo da essere compatibile con l'elenco dei popolari sistemi CAM.

Il comando *SMEXPORT2D* consente di esportare la rappresentazione dello spiegamento di un corpo in lamiera come un profilo 2D in formato .*DXF /o *.DWG.

Il comando *SMEXPORTOSM* consente di esportare un solido di lamiera nel formato .OSM (Open Sheet Metal) (nativo per il sistema CAM CADMAN- B).

21.5.22.6 Esportare un assieme con parti di lamiera su .dxf

Il comando *SMASSEMBLYEXPORT* esegue la ricerca nell'albero dell'assieme di tutte le lamiere e le esporta in un file *.DXF. Di conseguenza, nella cartella di output viene creato un gruppo di file *.DXF, ordinati in base allo spessore in diverse sottocartelle. Il comando genera un report HTML che consente di controllare l'output. La maggior parte delle parti in lamiera verrà elaborata automaticamente. Le parti di lamiera scadenti saranno segnalate dal comando per essere rielaborate manualmente.

21.5.23 Rappresentazioni Associative dello Spiegamento 2D di Solidi 3D

21.5.23.1 Panoramica

BricsCAD offre associatività bidirezionale tra la parte 3D Sheet Metal e il suo aspetto spiegato. In questa modalità esistono entrambe le rappresentazioni in documenti separati: la parte Originale che è memorizzata nel documento nativo e la parte Spiegata/Appiattita che è memorizzata in un documento temporaneo. L'associatività è temporanea ed esiste solo durante l'apertura di entrambi i documenti. Per poter lavorare in modalità associatività nella prossima sessione BricsCAD o una volta chiuso il documento, è necessario ripetere nuovamente i passaggi.

21.5.23.2 Cosa significa associativo?

Spiegamento Associativo consente di eseguire tagli sulla geometria appiattita e di propagare le modifiche al modello 3D. Ciò aiuta a semplificare gli 2 scenari:

- Creazione di piegature complesse, non standard e scarichi d'angolo.
- Evitare le autointersezioni nello sviluppo 2D.

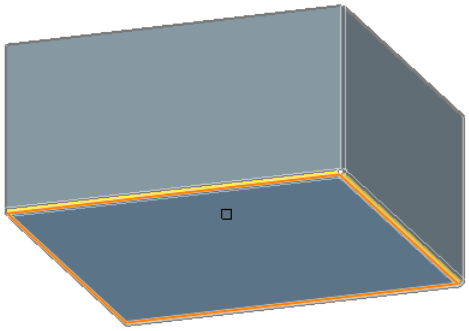
In questi 2 scenari, entrambi sono semplici da fare in 2D anziché in 3D. In modalità appiattita la parte di lamiera è rappresentata come un insieme di solidi piatti separati (per caratteristica). Utilizzare il comando *SMDELETE* per eliminare le flange solide e le piegature sui modelli piatti.

L'associazione è bidirezionale; le modifiche possono essere propagate dalle parti appiattite ai modelli 3D e viceversa. Le modifiche apportate a una delle rappresentazioni vengono propagate al documento con l'altra rappresentazione una volta che diventa attivo.

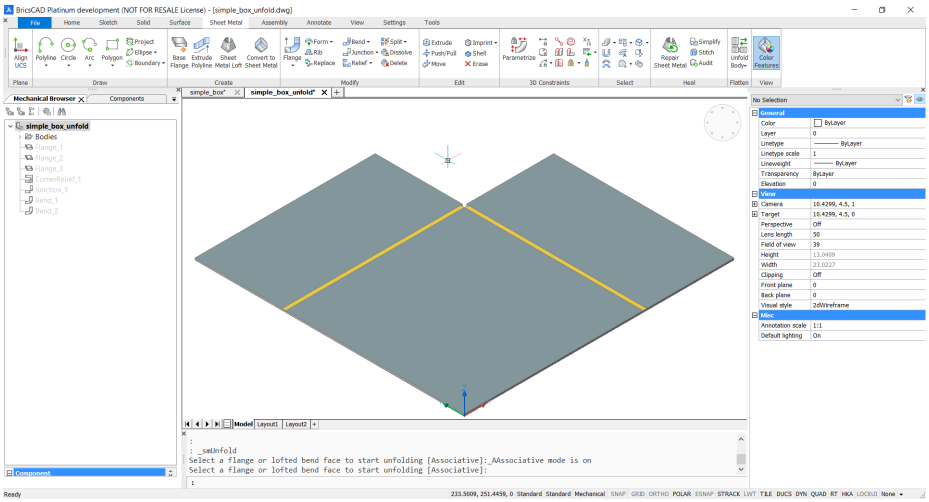
21.5.23.3 Passaggio alla Modalità Associativa

La modalità associatività viene attivata utilizzando il comando *SMUNFOLD*. Assicurarsi che non ci siano facce selezionate.

Dalla sezione **Lamiera** della barra multifunzione, selezionare l'icona **Spiegamento Associativo del Corpo** ( ) e selezionare la faccia di una **Flangia**.



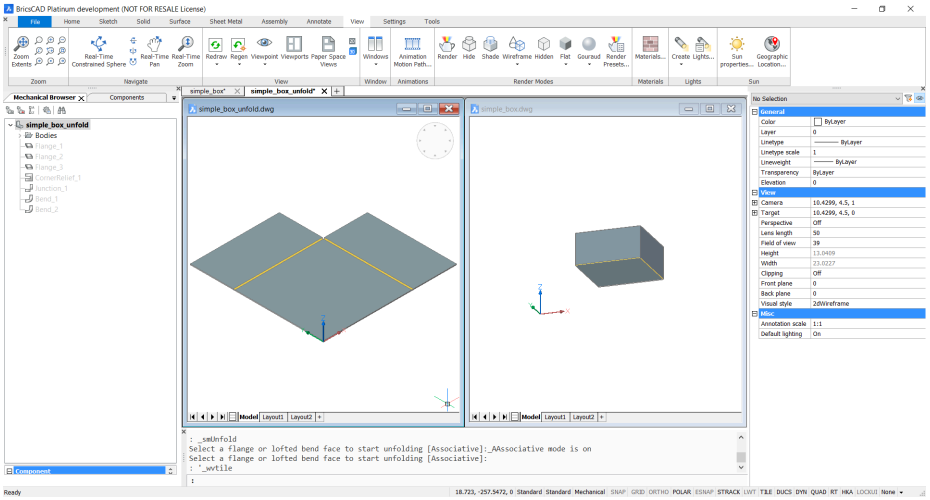
L'applicazione calcolerà lo spiegamento della parte specificata e la commuterà in modalità associativa. La condizione di successo di una determinata operazione è la stessa del normale spiegamento. Continuare a lavorare in modalità associatività solo se durante l'operazione non sono stati visualizzati errori o avvisi. Verrà creato un nuovo documento temporaneo contenente la parte spiegata.



Nota: Non è necessario salvare questo documento. Viene utilizzato per propagare le operazioni di modifica dal solido appiattito al modello 3D.

Usa **Affianca Verticalmente** per visualizzare sia il modello 3D che il documento dello spiegamento 2D associativo.

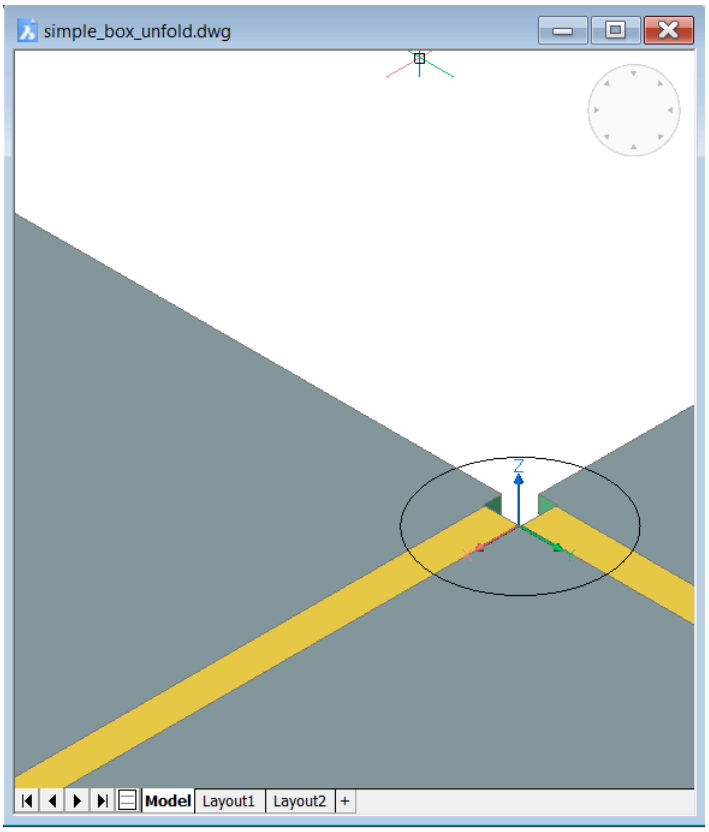
Fare clic sul pulsante **Affianca Verticalmente** sulla barra degli strumenti o sulla barra multifunzione.



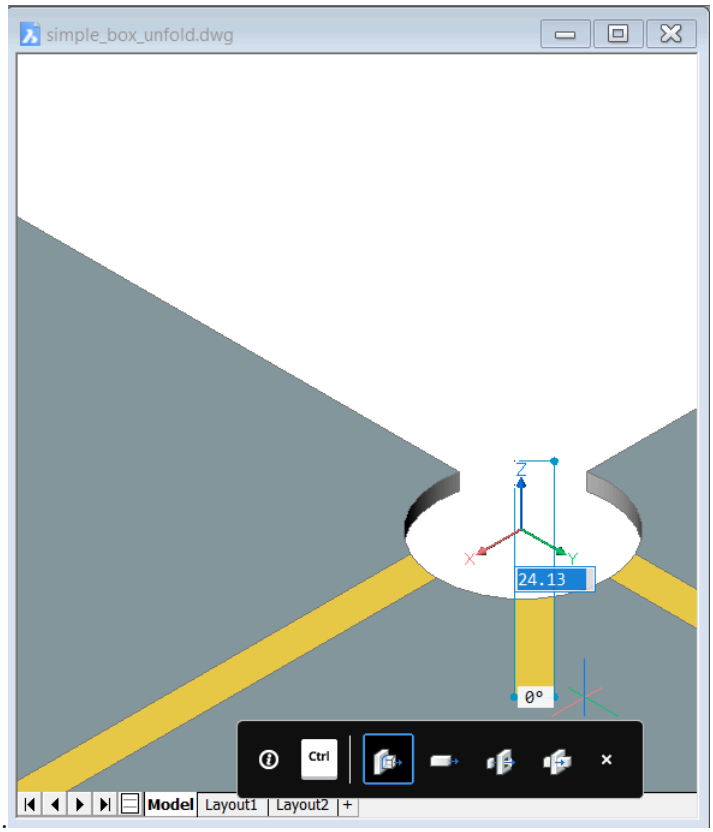
Nota: Nel **Navigatore Meccanico**, quando si seleziona la **geometria appiattita**, l'elenco delle caratteristiche è disabilitato.

21.5.23.4 Modificare la Geometria

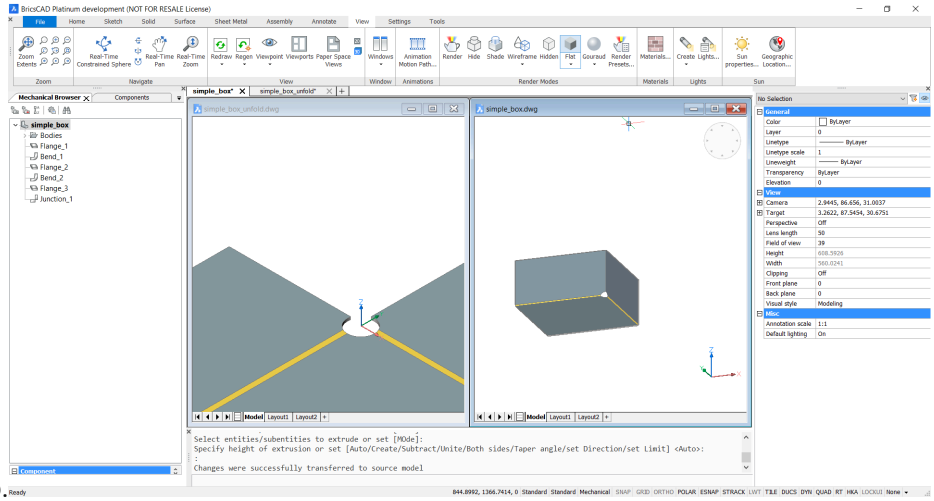
1 Lavorando con un documento appiattito in 2D, disegnare un cerchio nell'angolo.



2 Applicare il comando *DMEXTRUDE* al cerchio, tagliando il

volume. 

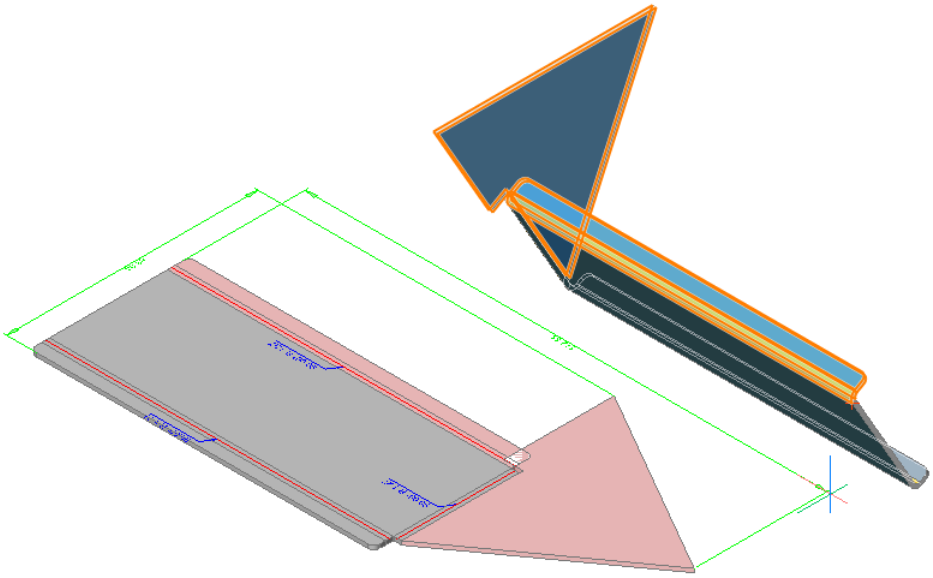
3 Cliccare di nuovo sul documento originale per renderlo

attivo. 

4 Il modello verrà aggiornato e verrà visualizzata l'inclinazione personalizzata dello scarico d'angolo. L'applicazione riporta nella barra dei comandi: Le modifiche sono state trasferite con successo al modello sorgente.

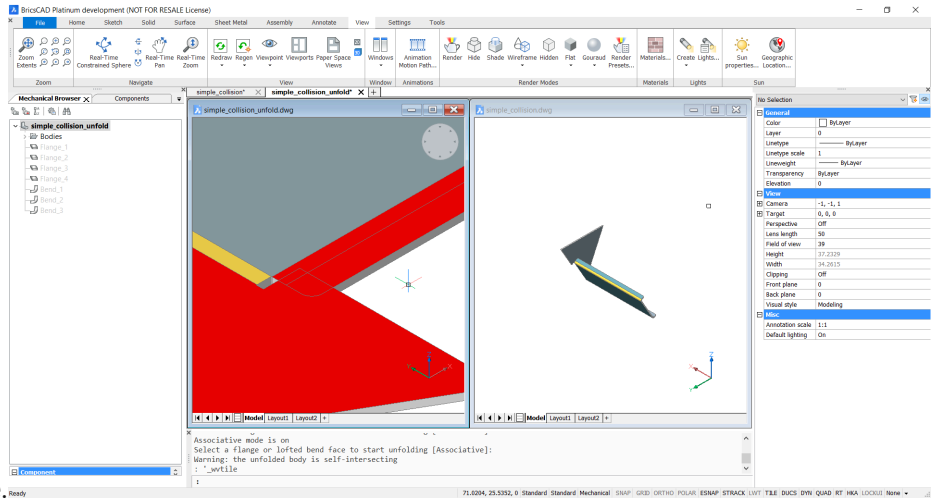
21.5.23.5 Evitare Collisioni nel Modello Spiegato in 2D

Un'altra utile applicazione dello spiegamento associativo è la risoluzione delle collisioni di materiale non ovvie.



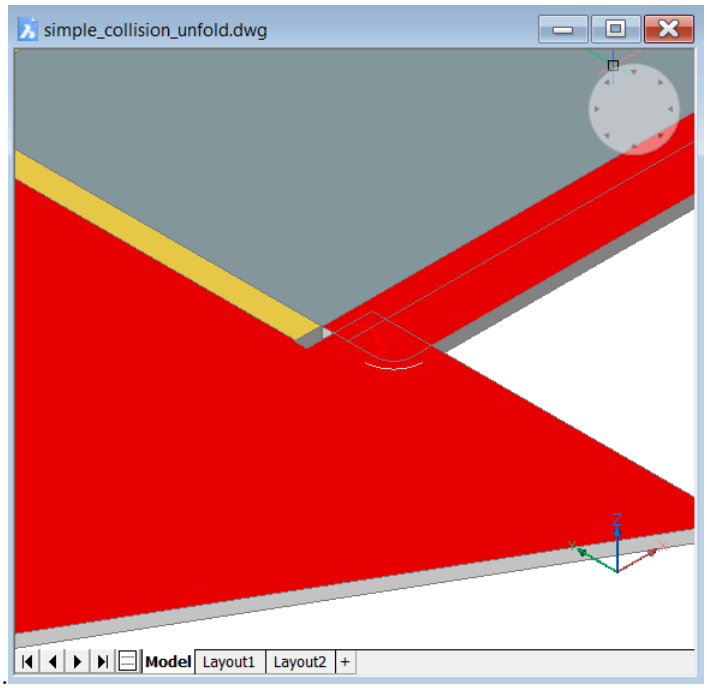
In questo esempio, non sappiamo quali modifiche applicare al modello 3D per evitare la collisione quando viene spiegato. Permettici di rielaborare il modello con lo Spiegamento Associativo.

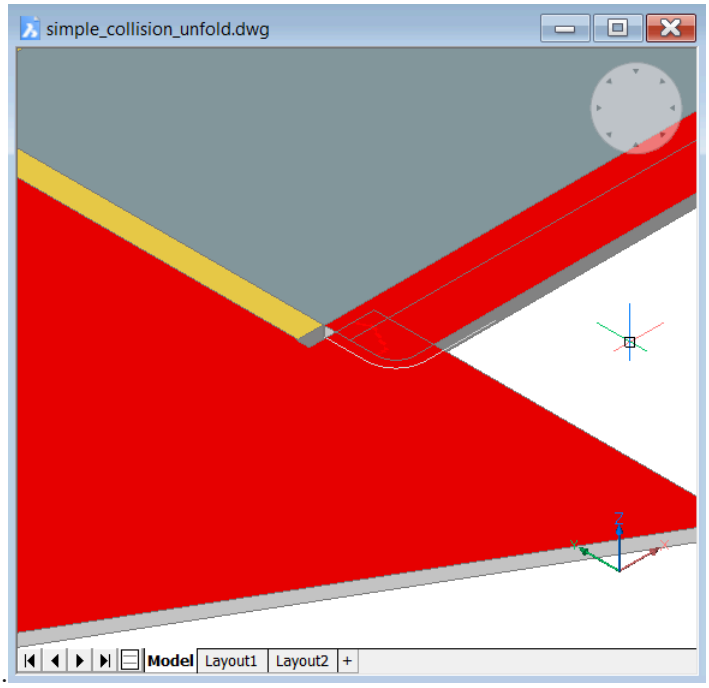
1 Dalla sezione **Lamiera** della barra multifunzione, selezionare l'icona **Sviluppo Associativo del Corpo** (), selezionare la faccia di una **Flangia**, aprire e fare clic sul pulsante **Affianca Verticalmente** () sulla barra degli strumenti o sulla barra multifunzione per affiancare i documenti aperti verticalmente e fare lo zoom sull'area della

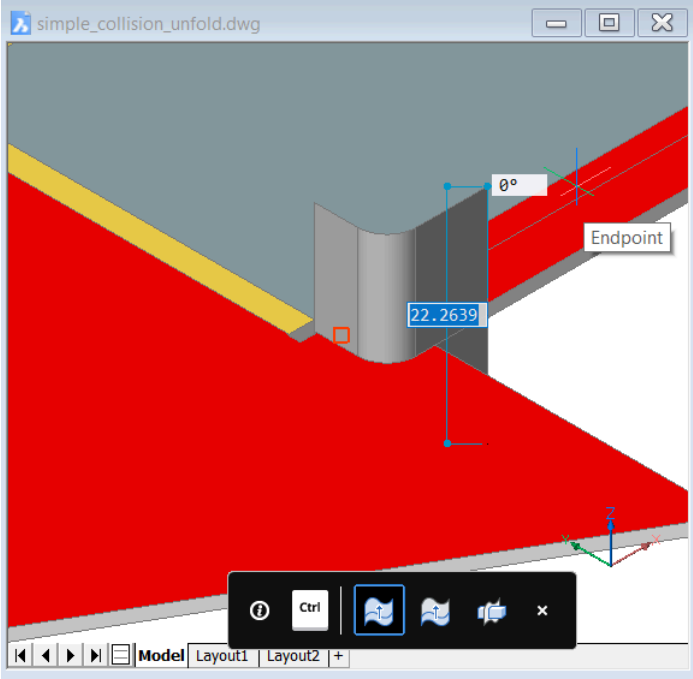


collisione.

2 Selezionare il bordo circolare del contorno e realizzare un offset di

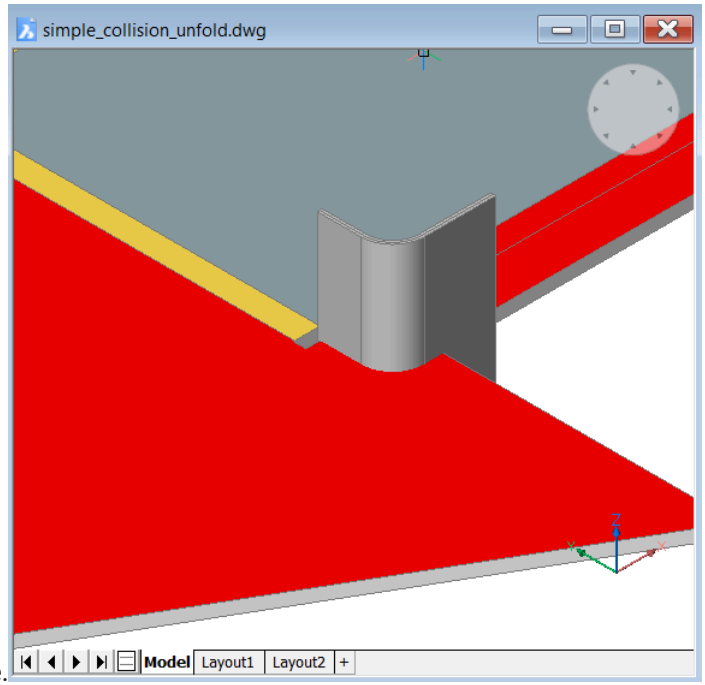
1. 

3 Disegnare linee dritte aggiuntive adiacenti al bordo e uniscile. 



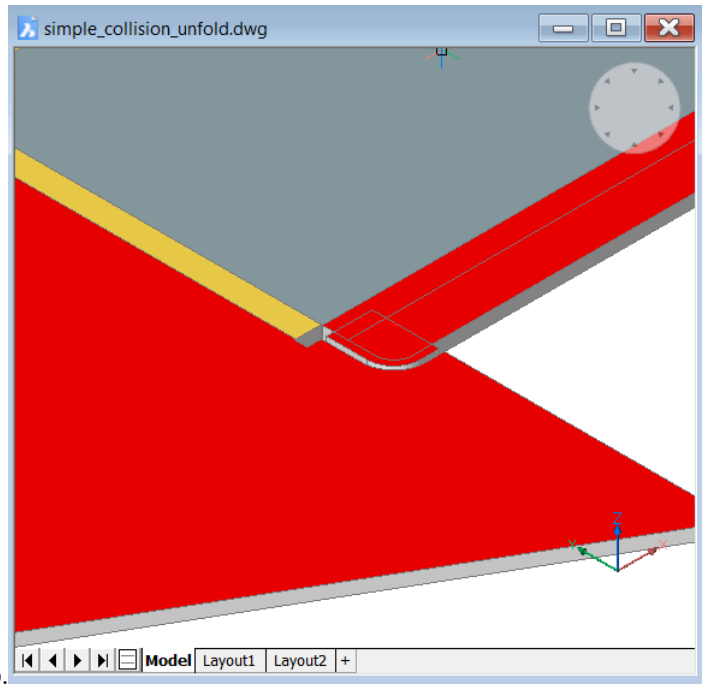
4 *DMEXTRUDE* la polilinea su entrambi i lati.

5 *DMTHICKENG* la superficie di un valore maggiore.

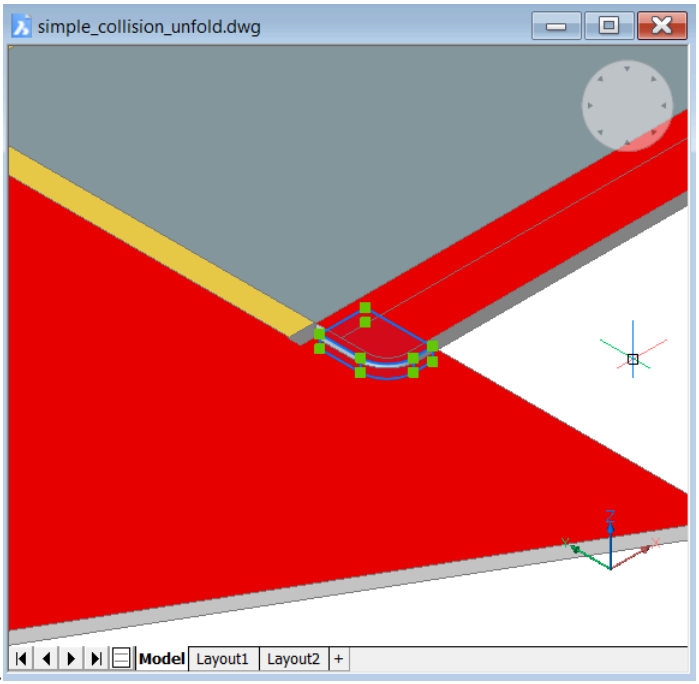


6 *SOTTRAI* il corpo ispessito dal modello

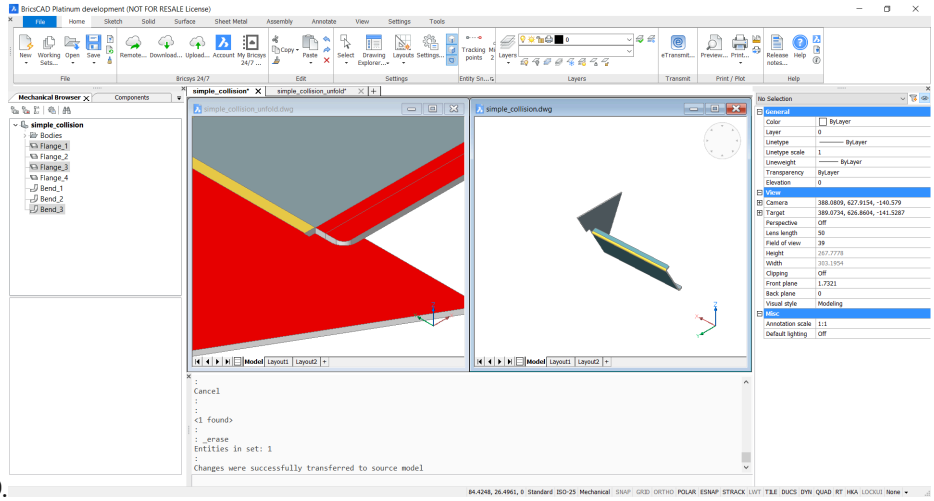
spiegato.



7 *CANCELLA* i pezzi rimanenti.



8 Ritorna al documento con il modello

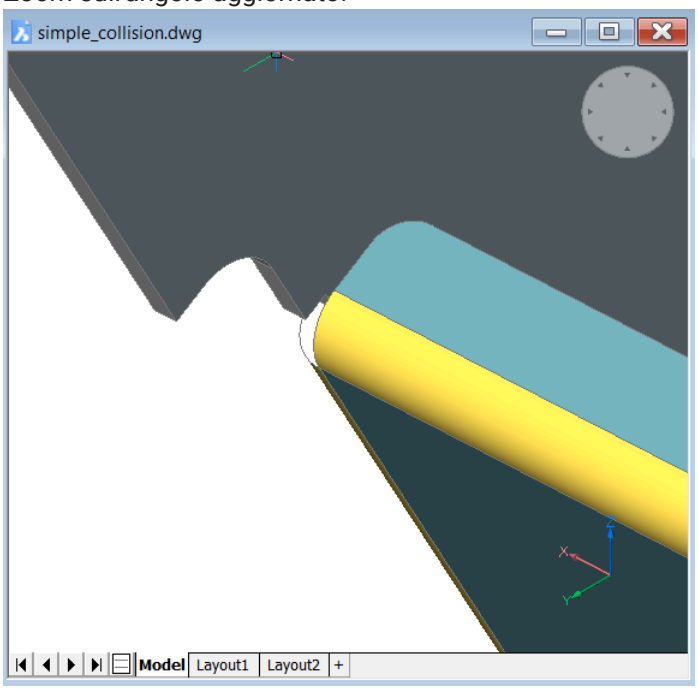


3D.

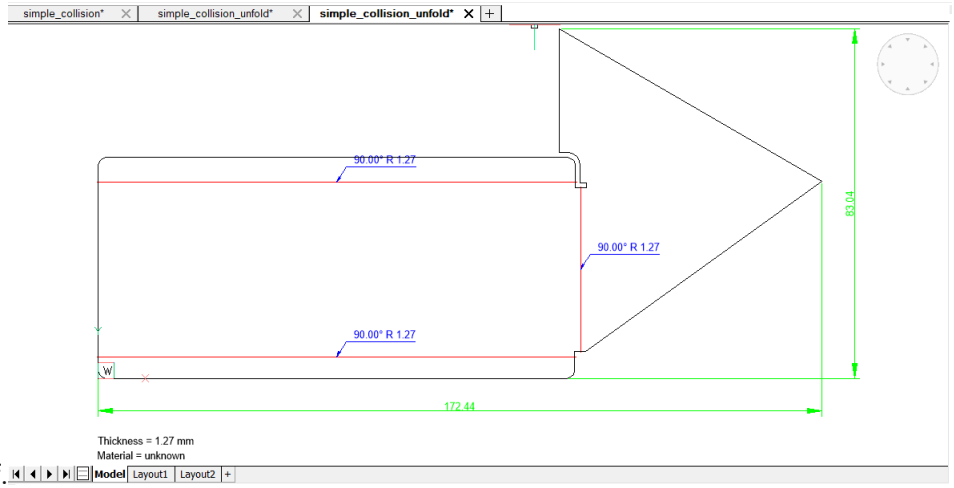
9 Assicurati che le modifiche siano state propagate.

Le modifiche sono state trasferite correttamente al modello di origine.

Zoom sull'angolo aggiornato.



10 Per verificare lo spiegamento 2D, esportare il file



come .dxf.

21.5.24 Caratteristiche Intelligenti

21.5.24.1 Panoramica

La tecnologia Caratteristiche Intelligenti gestisce l'aggiornamento intelligente locale della geometria dopo l'esecuzione dei comandi di Lamiera. Il comportamento è controllato tramite le preferenze dell'utente di **SmSmartFeatures.**

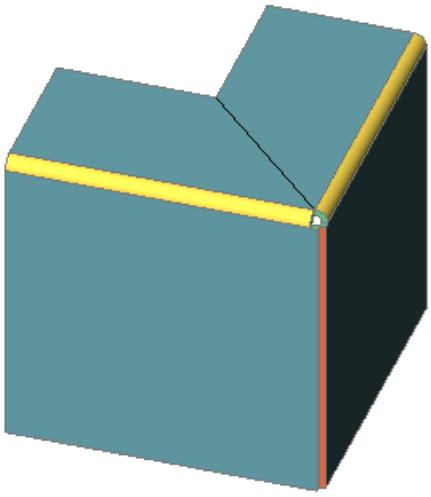
Sono disponibili 3 opzioni:

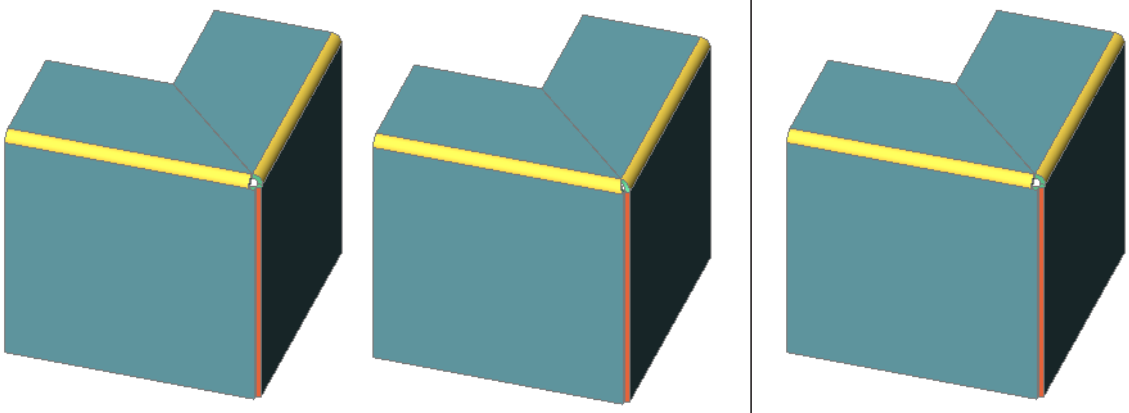
- Consenti ricostruzione delle caratteristiche della lamiera:
  - Gli scarichi d'angolo vengono trasformati in 2 scarichi dopo l'esecuzione di *SMSPLIT* su una flangia.
  - Gli scarichi non necessari vengono rimossi.
- Consenti impronta automatica degli spigoli dopo la ricostruzione: impronta automatica degli spigoli per le facce dello spessore.
- Consenti creazione automatica di giunzioni dopo aver creato curve.

Nota: Attiva sempre le opzioni Caratteristiche Intelligenti, a meno che non si verifichi un comportamento imprevisto.

21.5.24.2 Trasformazione di uno scarico d'angolo in gola di scarico

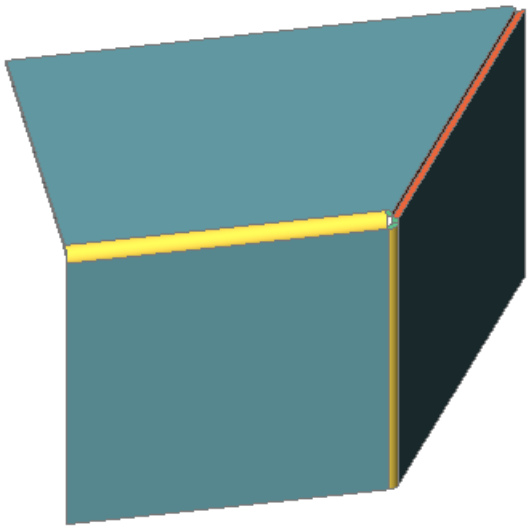
Quando si divide una flangia superiore, mediante una linea, in 2 parti separate, lo scarico d'angolo viene trasformato in 2 gole di scarico.



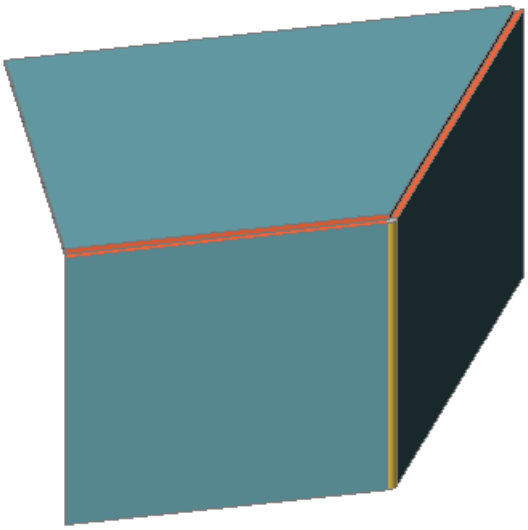
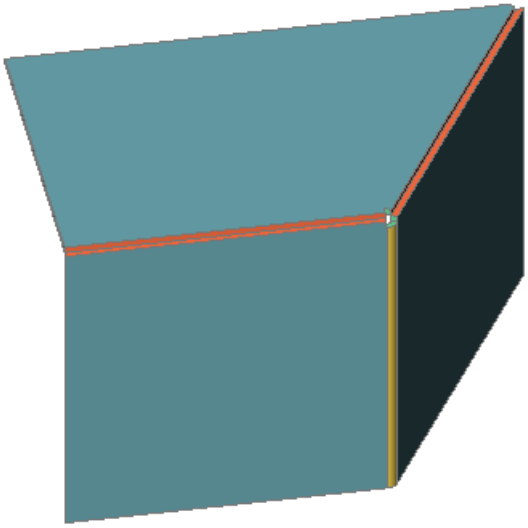
	
Le caratteristiche intelligenti creano automaticamente gole di scarico, che possono quindi essere commutate in modalità uniforme.	Senza le caratteristiche intelligenti lo scarico d'angolo è mantenuto e non può essere cambiato.

21.5.24.3 Rimozione di tagli non necessari

Nel modello sottostante esiste un taglio di scarico angolare, necessario per evitare che il materiale venga allungato.



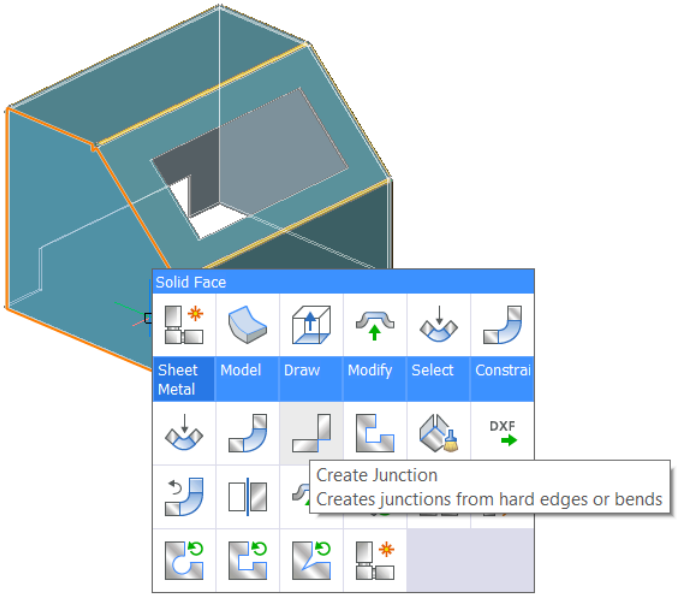
Quando la curva superiore viene convertita in una giunzione, si ottengono parti separate. Le funzioni intelligenti rimuovono automaticamente lo scarico ora non necessario.

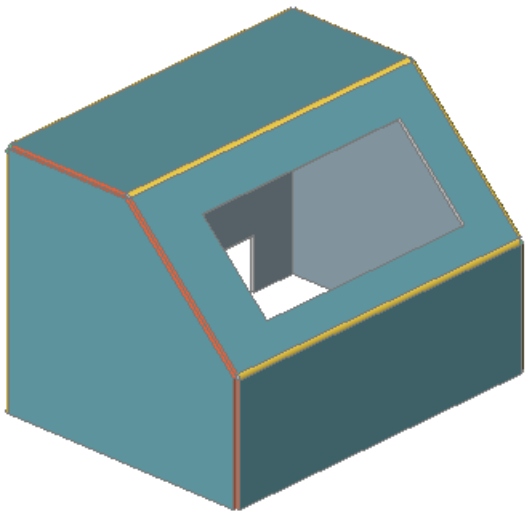
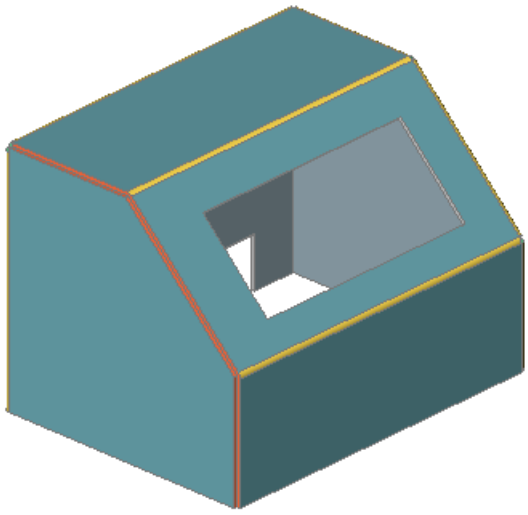
	
Le caratteristiche intelligenti riempiono automaticamente i tagli non necessari, semplificando così la geometria.	Senza le caratteristiche intelligenti, lo scarico d'angolo esiste ancora.

21.5.24.4 Impronta automatica degli spigoli

Gli spigoli impressi sulle facce di spessore li separano in aree che possono essere rielaborate automaticamente. Ciò significa che le giunzioni localizzate possono essere convertite in piegature. Senza spigoli impressi automaticamente è necessario utilizzare il comando *SMIMPRINT*.

Nell'esempio seguente, separeremo una flangia utilizzando giunzioni. Attivare le **Funzionalità di ricostruzione lamiera**, quindi utilizzare **l'Impronta automatica dei bordi dopo la ricostruzione**.

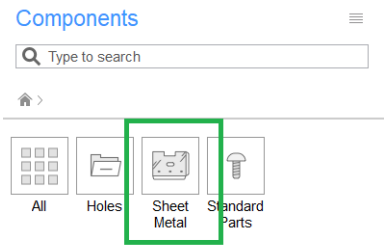


	
Consenti impronta automatica degli spigoli dopo la ricostruzione = ON. Le facce di giunzione sono divise correttamente.	Consenti impronta automatica degli spigoli dopo la ricostruzione = OFF. Le caratteristiche di giunzione si sovrappongono l'una all'altra.

21.5.24.5 Creazione automatica di giunzioni

In certe situazioni, quando spigoli vivi si incontrano all'angolo e si crea una piega su tutti tranne uno, che deve diventare una giunzione per rendere la parte dispiegabile.

Si supponga di creare pieghe sulla faccia superiore di una scatola in lamiera:



Quindi i rimanenti spigoli vivi adiacenti alla faccia devono diventare giunzioni. Quando **SmSmartFeatures** = ON, questa operazione viene eseguita automaticamente:

Per essere precisi, l'area di creazione automatica delle giunzioni si trova nel vertice comune degli spigoli vivi (o dello scarico angolare durante la creazione delle pieghe). L'opzione non ottimizza il layout della parte spiegata.

21.6 Disegni Generati

21.6.1 Viste di Disegno

21.6.1.1 Panoramica

BricsCAD Mechanical fornisce un gruppo di strumenti per generare viste di disegno da modelli 3D di parti e assiemi. Le vista di disegno vengono create nello spazio carta dello stesso file .dwg. Il modello 3D viene creato nello spazio modello. Le viste di disegno possono essere annotate utilizzando quote, direttrici, testo e tabelle. Le viste e le annotazioni vengono aggiornate quando viene modificato il modello 3D associativo.

Gli strumenti per generare, aggiornare e annotare viste di disegno si trovano nella scheda **Annotazione** dell'**Area di Lavoro Meccanico**:

Figura 152: Annotazione

Tabella 45: Annotazione

Vista di Disegno	Descrizione
Vista Base	Genera viste associative assonometriche ed ortogonali di un modello solido 3D in un layout dello spazio carta.
Viste Proiettate	Genera ulteriori viste proiettate da una vista di disegno esistente.
Vista di Sezione	Crea una vista sezione trasversale basata su una vista di disegno standard generata dal comando *VISTABASE* in un layout dello spazio carta.
Visualizza Dettagli	Crea una vista di dettaglio di una porzione di un disegno standard generato con una scala più grande.
Modifica Vista	Modifica la scala e la visibilità delle linee nascoste delle viste di disegno.
Aggiorna Viste	Aggiorna una selezione di viste di disegno ottenute da *VISTABASE* e *VISTASEZIONE* quando *VIEWUPDATEAUTO = 0*.
Stile Sezione	Stili predefiniti e definiti dall'utente.
Stile Dettaglio	
Esporta Viste	Esporta il contenuto delle viste di disegno, ottenuto con i comandi *VISTABASEe VISTASEZIONE*, nello spazio modello del disegno o in un nuovo disegno.

21.6.1.2 Viste di disegno in qualità bozza

Per impostazione predefinita, BricsCAD genera viste di disegno precise (di alta qualità) per qualsiasi modello 3D. Gli spigoli di tali viste (inclusi gli spigoli della silhouette) sono rappresentati da entità 2D; linee, cerchi, archi e spline. La generazione di viste precise per un numero elevato di entità 3D può richiedere un lasso di tempo consistente. Questo perché ogni spigolo di contorno deve essere approssimato con una curva 2D analitica.

È possibile ridurre il tempo necessario per generare viste di disegno cambiando l'impostazione *DRAWINGVIEWQUALITY* in *0*. Ciò consente di ottenere viste di disegno di qualità bozza. Queste viste di disegno sono rappresentate internamente come solidi 3D, il che rende impossibile inserire annotazioni sui bordi di contorno. Tuttavia, sono molto simili a una vista di disegno precisa (di alta qualità) e puoi usarle per creare rapidamente layout.

È possibile passare a una vista disegno di alta qualità utilizzando la sua proprietà *Geometria* (modificarla da *Bozza* a *2D*).

21.6.2 Annotazioni dei Disegni

Per creare e modificare le annotazioni, consultare l'Introduzione agli strumenti di Quotatura.

Quote e Multidirettrici presenti nelle viste di disegno vengono automaticamente aggiornate in caso di modifica del modello 3D sottostante.

I disegni degli Assiemi possono essere annotati con i Richiami BOM (pallinatura). Ogni richiamo si riferisce a una riga in una tabella dei materiali BOM generata dall'assieme.

La tabella dei materiali BOM può essere posizionata nello spazio carta (layout) con la vista di disegno di un assieme.

Le Rappresentazioni Esplose degli assiemi possono essere utilizzate per generare viste di disegno e annotarle con i richiami.

22. Communicator per BricsCAD

22.1 BricsCAD Communicator

BricsCAD Communicator is an optional plug-in for Pro and higher license levels of BricsCAD that imports and exports 3D data between major CAD formats. Its high-quality conversions facilitate the re-use of 3D CAD data to improve productivity and reduce project costs.

22.1.1 Supported export file formats

Windows

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **CATIA v5:** .CATPart, .CATProduct
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vd
- **XCGM:** .xcgm

Linux

- **3D PDF**: .pdf
- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

Mac

- **ACIS**: .asab, .asat, .sab, .sat
- **CATIA v4:** .model
- **IGES:** .iges, .igs
- **STEP:** .step, .stp

  The following export protocols can be specified; AP203, AP214, AP242
- **VDA-FS:**.vda

Nota: When the EXPORTPRODUCTSTRUCTURE system variable is set, you can export a BricsCAD assembly with mechanical components as a file with product structure.

22.1.2 Supported import file formats

Windows

- **ACIS:** .asab, .asat, .sab, .sat
- **Autodesk Inventor:** .iam, .ipt
- **CATIA v4:**.model
- **CATIA v5/v6:**.CATPart, .CATProduct, .CGR
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **NX:** .prt
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Linux

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **Siemens:** .jt
- **Solid Edge:** .asm, .par, .psm
- **SolidWorks:** .sldasm, .sldprt
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda
- **XCGM:** .xcgm

Mac

- **ACIS:**.asab, .asat, .sab, .sat
- **CATIA v4:**.model
- **Creo Elements / Pro Engineer:** .asm, .prt
- **IGES:** .iges, .igs
- **Parasolid:**.xmt_bin, .xmt_txt, .x_b, .x_t
- **STEP:** .step, .stp, .stpz
- **VDA-FS:** .vda

Nota:

Depending on the Communicator settings:

- You can import an assembly file, as a plain geometry (a nonhierarchical set of entities) or map its product structure into native blocks or mechanical components.
- PMI (Product and Manufacturing Information) is supported for the following file formats (Communicator license is required): ACIS, CATIA V5, NX, Pro/E / Creo, SolidWorks, XCGM. PMI annotations are imported as graphical elements grouped in blocks.
- If the assembly contains hidden parts, you can decide how to represent them in BricsCAD: 'import and hide', 'import and set visible' or just 'do not import'. This feature is currently supported on CATIA v5 only.
- You can set import options to automatically repair, stitch and simplify imported geometry.
- The IGES format has its own options to stitch and simplify. If you import a 3D model from SolidWorks, it makes sense to set an option to map Y axis into Z (because in SolidWorks the vertical axis is Y).
- For SolidWorks and other native CAD formats, it is possible to specify an alternate search path. This will be used to look for parts absent in the imported assembly folder.

22.2 BricsCAD Communicator settings

The **Communicator** section in the **Settings** dialog box allows you to customize the behavior of the related commands:

- setting environment options.
- control the import/export of data.
- toggling post-processing operations.

Settings are applicable when the EXPORT and IMPORT commands engage one of Communicator file formats.

22.2.1 Import settings

22.2.2 Product structure

Name: ImportProductStructure

Description: Determines how the imported entities will be structured and whether it will form a soup of entities, a block hierarchy or an assembly structure.

Values:

- **[0] None**: Creates a flat structure without blocks in the modelspace of the target database, regardless of whether the imported data have an assembly structure or not.
- **[1] As blocks**: The imported data have an assembly structure, which will be converted into a hierarchy of plain blocks; so the structure will be preserved, while the assembly metadata are lost. If the imported data is composed of entities only, they are placed in the modelspace of the target database.

- **[2] As mechanical components**: In this mode, data will be translated into BricsCAD assembly data, the structure and its properties (physical materials – BLMATERIALS command). If the imported file has no assembly data, a mechanical component will be created in the BricsCAD target document root anyway.

22.2.3 Translate colors

Name: ImportColors

Description: Determines how the colors of the imported entities will be assigned: either as true colors or as matching the current color palette.

Values:

- **[0] To RGB**: All entity colors will be converted to RGB, regardless of the current palette.
- **[1] To RGB if no matching palette index**: If the entity color is found in the palette, then the entity gets an index color. Otherwise, it is given a true color.
- **[2] To nearest palette index**: For any true color of the imported entity, the nearest match is searched in the palette and this index color is assigned to the entity.

22.2.4 Product and manufacturing information

Name: ImportPMI

Description: Determines whether product manufacturing information is imported. Currently such information is imported as exploded data (lines, text, ...) instead of compound entities (e.g. annotations).

Values: **on/off** checkbox.

22.2.5 Perform stitching

Name: ImportStitch

Description: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported.

Nota:

- Stitch operations are time-consuming when importing large files.
- Check the IMPORTIGESSTITCH setting, which can set an override for the IGES file format.

Values: **on/off** checkbox.

22.2.6 Repair model on import

Name: ImportRepair

Description: Applies operations, similar to the DMAUDIT command, in order to improve the quality of the imported geometry. Geometry modeled in CAD systems which use a kernel different from ACIS, often needs to be healed because of possible flaws.

Values: **on/off** checkbox.

22.2.7 Perform simplification

Name: ImportSimplify

Description: Applies operations similar to the DMSIMPLIFY command:

- Convert imported splines into canonic surfaces.
- Simplify topology (remove imprinted edges) if possible.

Nota: Also check the ImportIgesSimplify setting, which can set an override for the IGES file format.

Values: **on**/**off** checkbox.

22.2.8 Hidden parts

Name: ImportHiddenParts

Description: Defines how hidden (invisible) entities in the imported geometry are processed.

Values:

- **[0] Import and hide**: All entities are imported; invisible entities are hidden. Note that currently there are no user tools to make these hidden entities visible again.
- **[1] Import and set visible**: All entities are imported and visible with regardless of the visibility in the source file.
- **[2] Do not import**: Hidden entities in the source file are not imported.

22.2.9 Creo options

Alternate search paths

Name: ImportCreoAlternateSearchPaths

Description: This setting is taken into account when Creo files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

Values: string; if empty, the search is performed in the folder of the imported file only.

22.2.10 IGES options

1 **Perform stitching**

Name: ImportIgesStitch

Description: In some cases, imported geometry represents solid geometry as a set of separate surfaces. In order to be able to work with solid operations on the imported geometry, the DMSTITCH command has to be applied. If this setting = ON, the DMSTITCH command is executed automatically when the geometry is imported. This setting determines whether a stitch operation has to be applied to imported entities when the format of the source file is IGES.

Nota:

- Stitch operations are time-consuming when importing large files.
- For file formats other than IGES, the ImportStitch setting applies.

Values: **on**/**off** checkbox.

2 **Perform simplification**

Description: Determines whether a DMSIMPLIFY operation has to be applied to the imported geometry when the source file format is IGES

Nota: For other formats, see the ImportSimplify setting.

Values: **on/off** checkbox.

3 **Inventor options**

Alternate search paths

Name: ImportInventorAlternateSearchPaths

Description: Setting is taken into account when Inventor files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

Values: string; if empty, the search is performed in the folder of the imported file only.

4 **NX options**

Alternate search paths

Name: ImportNxAlternateSearchPaths

Description: Setting is taken into account when NX files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

Values: string; if empty, the search is performed in the folder of the imported file only.

5 **SolidEdge options**

Alternate search paths

Name: ImportSolidEdgeAlternateSearchPaths

Description: Setting is taken into account when SolidEdge files are imported. The value is a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

Values: string; if empty, search is performed in the folder of the imported file only.

6 **SolidWorks options**

- **Map SolidWorks Y to BricsCAD Z axis**

  Name: ImportSolidWorksRotateYZ

  Description: Corrects the difference between the SolidWorks and BricsCAD coordinate systems. If unchecked, imported geometry in BricsCAD is rotated by 90 degrees compared to its position in SolidWorks.

  Values: **on/off** checkbox.

- **Alternate search paths**

  Name: ImportSolidWorksAlternateSearchPaths

  Description: Setting is taken into account when SolidWork files are imported. The value is

a set of semi-colon (;) delimited, fully qualified paths used to search for missing assembly references during product structure import operations.

Values: string; if empty, search is performed in the folder of the imported file only.

7 **STEP options**

Map Y to BricsCAD Z axis

Name: ImportStepRotateYZ

Description: Corrects the difference between coordinate systems in different coordinate systems, i.e. SolidWorks and BricsCAD. This can be helpful to prevent undesired rotation when the origin CAD of STEP file is known, i.e. if it is SolidWorks.

Values: **on**/**off** checkbox.

22.2.11 Export settings

22.2.12 Product structure

Name: ExportProductStructure

Description: Determines whether the assembly structure will be preserved in the target document.

Values:

- **[0] No product structure**: Exports a flat structure without components in the target document, whether the BricsCAD document has a product structure or not.
- **[1] Export product structure**: Exports the BricsCAD product structure data (if existing) to the target document.

22.2.13 Hidden parts

Name: ExportHiddenParts

Description: Determines if invisible entities are exported or not. Entities can be invisible because of:

- the result of the HIDEOBJECTS command.
- sitting on a hidden layer.
- owned by an invisible component.

Values:

- **[0] Export and hide if possible**: Exports hidden entities. If the target format supports hidden entities, hidden entities in the source document will be hidden also in target document.
- **[1] Do not export**: Invisible entities are skipped.

22.2.14 STEP options

STEP export format version

Name: ExportStepFormatVersion

Description: Allows you to select the available specifications of the STEP format.

Values:

- **[0] AP203**: Selects AP203 protocol.
- **[1] AP214**: Selects AP214 protocol.
- **[2] AP242**: Selects AP242 protocol.

23. BricsCAD Shape

23.1 Introduction to BricsCAD Shape

23.1.1 What is BricsCAD Shape

BricsCAD® Shape is the new conceptual modeling tool from Bricsys. It's made to help architects and engineers streamline their 3D schematic design work. Shape is the fastest way to get designs out of your head and in front of your clients. It's CAD-accurate, using a powerful direct modeling engine.

You can use it as a live presentation tool, too. Potentially best of all, Shape offers a clean, simple user interface that you can learn in 30 minutes or less.

23.1.2 BricsCAD Shape lets you create without limits

Shape's smart modeling tools make it easy to capture in 3D. The multifunctional "Shape" tool can be used to create walls, slabs, structural elements and more. Direct modeling features let you drag, connect, push/pull and extrude as you like. Wall connections are automatically mitered for you.

BricsCAD Shape also offers everything you need to present your designs. It comes with a library of materials, textures and pre-built 3D components. You can use the included collection of doors and windows, or create your own custom versions easily. Make your model look the way you want using the Visual Styles panel in Shape.

23.1.3 How is BricsCAD Shape different from other tools?

First, BricsCAD Shape is CAD-accurate from the start. Under the simple UI is the same fast and proven parametric, 3D direct modeling engine that's in BricsCAD Platinum. Shape creates solid models, stored in industry-standard DWG – just like BricsCAD® BIM. Unlike competing products, Shape's solid models are fully accurate – never approximated. Every element that you create in Shape can be modified deeply, anytime, on the fly. Your concept models open directly in BricsCAD BIM. You'll never lose time starting over again. Make the smart move – directly from concept to BIM with BricsCAD Shape with no loss of detail.

23.1.4 BricsCAD Shape accelerates your BIM workflow

The Bricsys BIM workflow starts in 3D with Shape, and stays in 3D – speeding 2D construction documentation with the world's best drafting tools. No breaks or transitions in your workflow, all in DWG, with a friendly, familiar user interface that you'll appreciate. It won't take long for you to be productive in BricsCAD Shape or BricsCAD BIM.

23.1.5 Stop sketching. Start shaping!

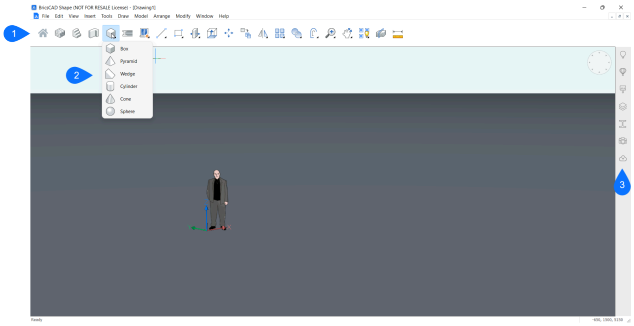
Get your free copy of BricsCAD Shape! BricsCAD Shape delivers the tools you need, and none that you don't. Even the learning materials are clean and simple – check out "10 Minutes to BricsCAD Shape" on YouTube, Bricsys channel.

23.2 BricsCAD Shape user interface

The toolbar at the top of your screen (1) gives you fast access to the most used tools. Left-click on any icon in the toolbar to use the tool.

Many top-level tools offer sub-menus (2). Hover over the icon, left click and hold. The sub-menu will fly out to show these additional tools.

The icons on the right side of your screen (3) give you access to a set of collapsible panels. The top-most icon opens the **Tips panel**. You'll find more details on how to use the tools in Shape here.



Nota: Many of the modeling tools that are used in BricsCAD and BricsCAD BIM are also available in Shape. You can get an overview of all available commands in Shape by typing in COMMANDS in the Command line.

23.3 Basic concepts of modeling in BricsCAD Shape

23.3.1 Selection methods

Highlighting and selecting objects

Entities in BricsCAD Shape are solids, which are made up of edges and faces.

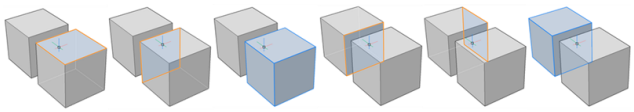
When hovering over an entity, a part of the entity is highlighted. To select the highlighted part of the entity, left-click the edge or face.

Nota: To select or highlight the whole entity, hold down the **CRTL** key and left-click.

For some operations, it's not necessary to select entities. When a surface or edge is highlighted, the **Quad** opens and several commands can be accessed.

Nota: You can only highlight one entity at a time, while it is possible to select multiple entities at once by left-clicking them one-by-one, or using a window selection method.

Sometimes, the entity you want to select is hidden behind other entities. Place the cursor over the obscured entity, then repeatedly press the **TAB** key. All (sub)entities under the cursor highlight one by one. Click to select the currently highlighted (sub)entity.



To clear a selection of objects, press the **ESC** button.

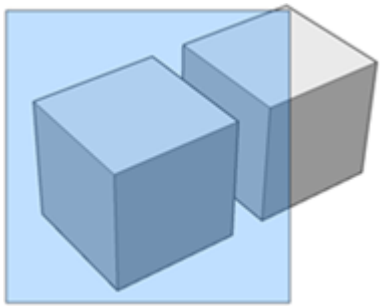
Selection windows

To select one or multiple elements at once, a selection window is used. Left click on an empty space in the model area. When you move the cursor a selection window displays.

Nota: The selection window selects entire solids by default. Press the **Ctrl** key to select faces, press the **Ctrl** key again to select edges and again to select solids. The current selection method is indicated in the **Hot Key Assistant** (HKA) at the bottom of the screen.

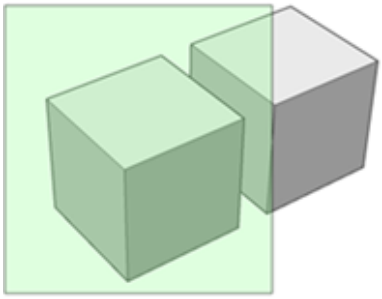
Blue selection window

Moving the cursor from left to right, a blue selection window displays: all entities that are completely inside the rectangle are selected. This selection method is called 'Window Inside'.



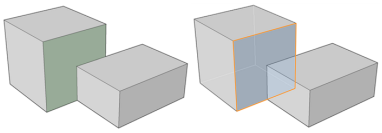
Green selection window

Moving the cursor from right to left, a green selection window displays: all entities that are completely inside or overlapping the selection rectangle are selected. This selection method is called 'Window Overlap'.



Boundaries vs Faces

Boundaries can be defined by a set of 2D entities in the XY field of the current coordinate system or on a 3D solid face or by partially overlapping faces of 3D solids. BricsCAD Shape automatically detects and highlights closed boundaries, as you hover over different model elements. When a boundary is detected, it will be highlighted. To select it, just left-click inside the highlighted boundary. Boundaries can be used to create new solids or making holes or offsets and recesses.



Nota: To select the entire solid face instead of a boundary, press the **TAB** key.

23.3.2 Quad

The Quad cursor menu, or Quad for short, is an alternative to the Command line, toolbars or ribbon, offering a rich set of tools while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs. It includes rollover tips, which display a limited set of properties when selecting or hovering over an entity.

The Quad displays when:

- You hover over an entity.
- A selection set exists.
- You right click on an empty space on the screen. This will open the No Selection Quad.

The initial content of the Quad when hovering over an entity is:

- Entity type.
- The icon of the most recently used command on this entity type.
- Relevant properties.

When you move the cursor to the Quad, it expands. On top you can find the most relevant or most recently used commands for the entity. Below you see a series of command tabs:

- Move the cursor over a tab to expand it.
- Place the cursor over an item to see a tool tip.
- Click an icon to launch a command.
- Click the Quad title bar to collapse, click again to expand.

23.3.3 Manipulator

The manipulator allows you to move, rotate scale and mirror selected entities and optionally create copies.

To display the Manipulator do one of the following:

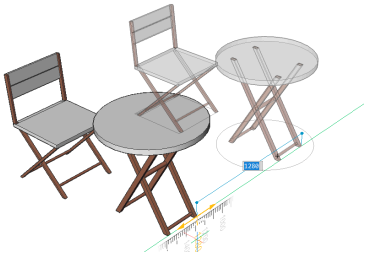
- Select the entity or subentity and choose **Manipulator** in the **Modify** tab on the **Quad**.
- Hold down the left mouse button a little bit longer when selecting.

Nota: The Manipulator aligns along the edge that you enter the entity by. Its origin sits at the closest endpoint of that edge.

Select an axis

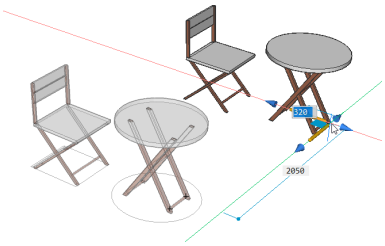
By selecting an axis of the manipulator you are able to move the entity. Specify the value of displacement by entering a value or picking a point. The step size of the ruler depends on the zoom factor. Zoom in to decrease the step size, zoom out to increase.

Nota: To copy the entity, hold down the **CTRL** key before clicking the axis.



Select a plane

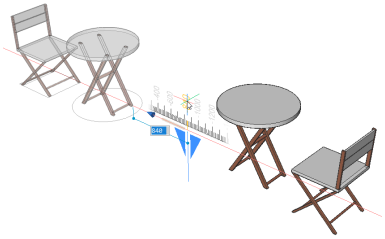
By selecting a plane of the manipulator you are able to move the entity in the selected plane after specifying the value of displacement by entering a value or picking a point.

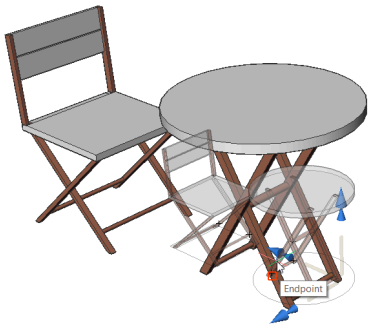


Select an arrowhead

By selecting an arrowhead of the manipulator you are able to scale or rotate the selected entity. By right clicking when hovering over the entity, a contexts menu with other options is displayed.There the function of an arrowhead can be changed between 3D mirror and 3D scale.

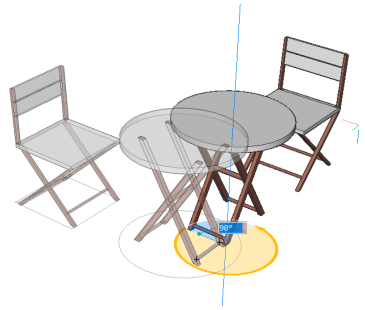
Select one of the arrowheads and specify the mirror axis or scale.





Select a rotation arc

By selecting an arc of the manipulator you are able to rotate the entity after entering the rotation angle value or specifying by clicking on a point in the drawing.



Select the anchor handle

By selecting the anchor of the manipulator you are able to relocate the manipulator.

23.4 Navigation in BricsCAD Shape

23.4.1 Panning

You can move the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space.

There are several ways to pan:

- Hold down the middle mouse button.
- Press and hold the **CRTL+SHIFT** and right mouse button.
- Type the *PAN* in the Command line.
- Type the *RTPAN* in the Command line.
- Click the **Real-Time Pan** icon in the toolbar.

To abort the *PAN* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

23.4.2 Rotating a view

You can rotate a view freely, in a constrained manner.

To rotate in real-time:

- Hold down the **SHIFT** and middle mouse button.
- Type *RTROT* in the Command line.
- Click the **Real-Time Constrained Sphere** icon in the toolbar.

To abort the *RTROT* command, do one of the following:

- Right click and choose **Exit** in the context menu.
- Press **Enter** key, **space bar** key or **Esc** key on the keyboard.

23.4.3 Zoom in/out

You can change the magnification of your drawing at any time by zooming.

Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail.

Changing the magnification of the drawing affects only the way the drawing is displayed. It has no effect on the dimensions of the entities in your drawing.

To zoom in/out:

- Scroll the mouse wheel.
- Press and hold the **CRTL+SHIFT** and left mouse button to zoom in or out.
- Type **ZOOM** in the Command line.
- Click the **Zoom In** or **Zoom Out** icons in the toolbar.

Nota: **Zoom Extents** displays the drawing to the extents of entities

23.4.4 Look From control

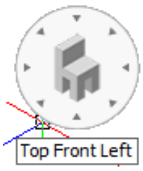
The **Look From** control allows you to select a number of preset views such as orthographic views and isometric views. By default the **Look From** control displays in the top right corner of the graphic screen.

1 Move the cursor to the **Look From** control.

The **Look From**control becomes active.

2 Position the cursor on the **Look From** control to choose a view orientation.

A preview image and a tooltip indicate the currently selected view orientation.



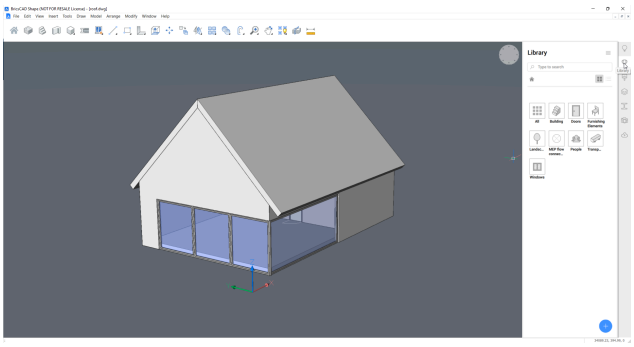
3 Click to confirm.

4 The view is updated.

23.5 Components in BricsCAD Shape

BricsCAD Shape includes a collection of components in the **Library** panel.

Open the library by clicking the airballoon icon on the right hand edge of the screen. Several categories are available.



Click the menu button at the top right corner of the panel to select or hide the available libraries:

- **Bricsys BIM library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Bim\Components*.
- **Bricsys 2D library**: content of *C:\Program Files\Bricsys\BricsCAD Vxx en_US\UserDataCache\Support \en_US\Blocks2D*.
- **User library**: content of *C:\ProgramData\Bricsys\Components*

Nota: Choose **Manage Libraries** in the menu to add folders to the COMPONENTSPATH system variable in the **Settings** dialog box.

23.5.1 To insert a component

1 Click the component in the library.

2 Move the cursor into the model.

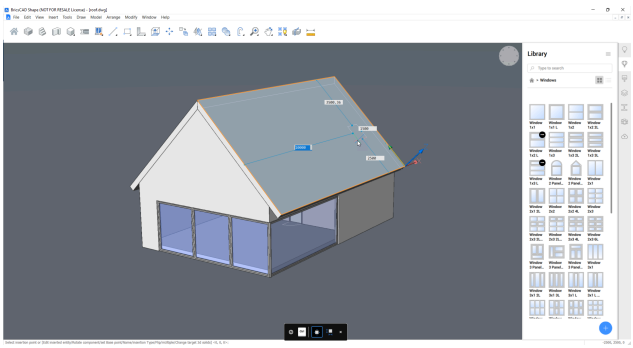
Nota: The selected component is attached to the cursor. The X and Y distances from the origin of the current coordinate system display dynamically.

3 Do one of the following:

- Click to place the component.
- Type a value in the X and Y distance fields.

Nota: Press the **TAB** key to jump to the other field. Press Enter to place the component.

23.5.2 To insert a window in the face of a solid



1 Click the window in the library.

2 Move the cursor into the model over the face of 3D solid.

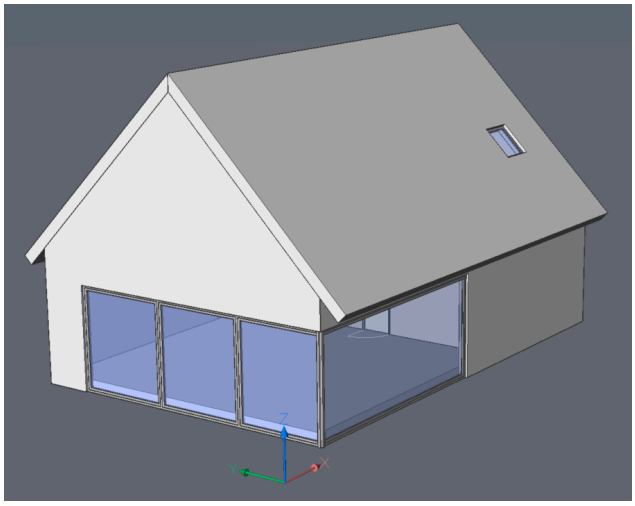
Nota: The window aligns to the face under the cursor. Distance fields from the sides of the face display dynamically.

3 Do one of the following:

- Click to insert the window.
- Type a value in distance fields.

  Nota: Press the **TAB** key to jump to the next field. Press Enter to place the component.

  The window automatically makes an opening in the solid.



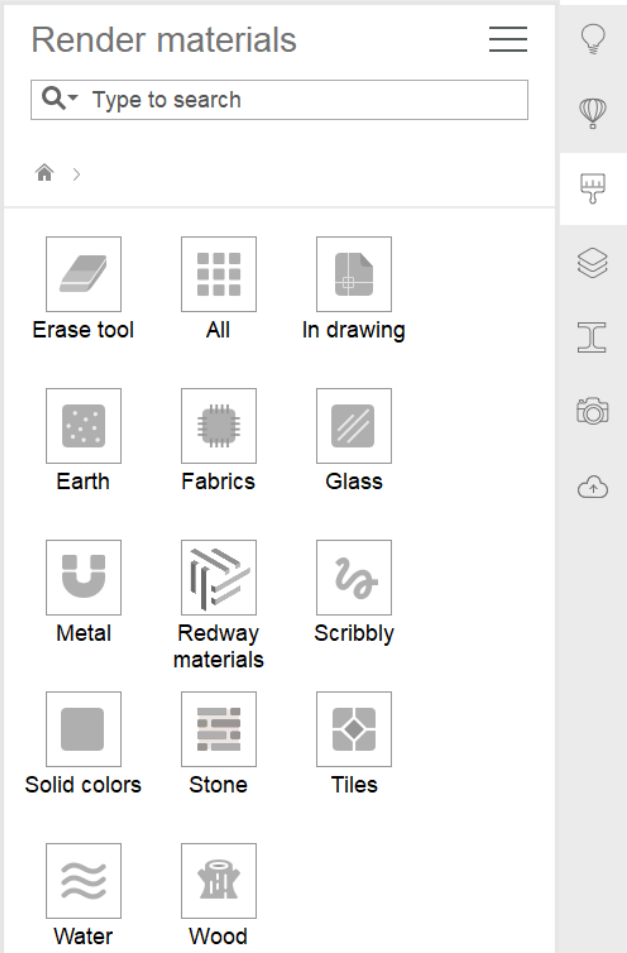
Nota: When you delete a window, the opening is removed.

23.6 Materials in BricsCAD Shape

BricsCAD Shape includes a collection of materials in the **Render materials** panel.

Render materials are stored in: *C:\ProgramData\Bricsys\RenderMaterials\UserMaterials*.

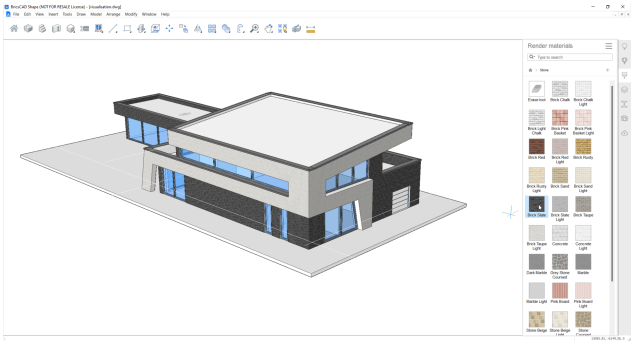
Open the **Render materials** panel by clicking the **paintbrush** icon on the right-hand edge of the screen. Several categories are available. To view all materials that are used in the drawing, select the **"In drawing"** category.



To assign a material to an entity in your model:

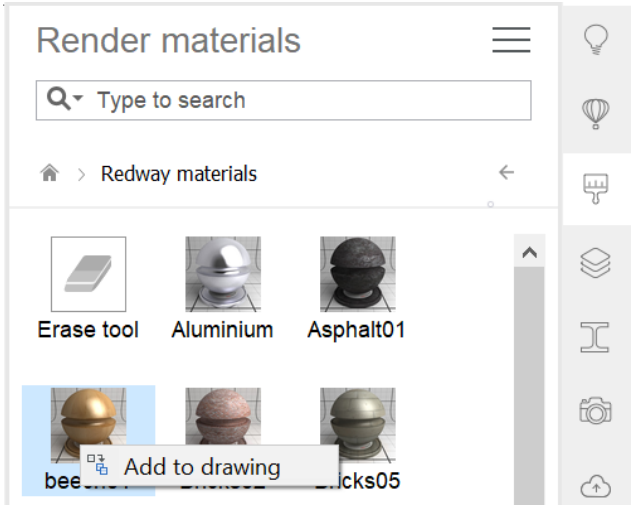
- Select the material in the **Render materials** panel.
- Select the entity you want to assign the material.

Nota: You can apply a material to multiple entities at once by preselecting them.



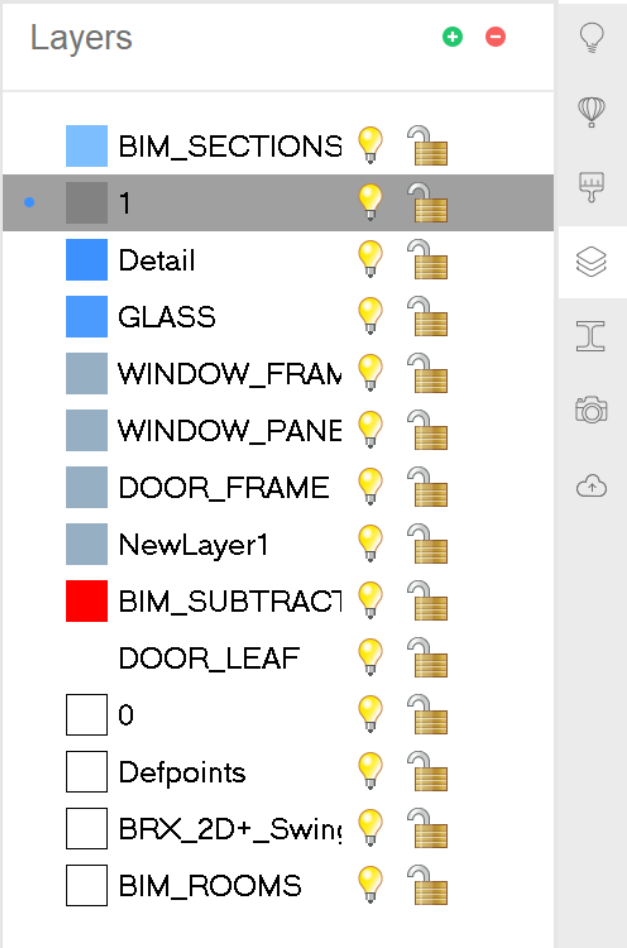
Nota: To remove a material from an entity, use the erase tool and click any entity to remove the material from.

It you want to add materials to the drawing to use later, right-click the material icon and select the "**Add to drawing**" option.



23.7 Layers in BricsCAD Shape

Layers can be used to group entities to organize your 3D model. The **Layer** panel can be accessed through the stacked sheets icon on the right side of the screen.



Nota: The current layer is indicated with the small blue dot. To change the current layer, click in the same column next to the layer you want to set as current layer.

23.7.1 Turn on/off

To turn on and of specific layers, toggle the lightbulb of the layer on and off. This way elements can be hidden or isolated. This can be of huge help with navigation.

- **On**: Displays entities that are on that layer.
- **Off**: Hides entities that are on that layer.

23.7.2 Lock/unlock

To lock or unlock layers, toggle the lock icon of the layer.

- **Unlocked**: Allows entities on that layer to be edited.
- **Locked**: Prevents entities on that layer from being edited.

23.7.3 Create/delete

To create a new layer, click the **green plus** sign. A new layer with a generic name is created. To delete a layer, make sure the layer to delete is selected and click the **red minus** sign.

23.7.4 Edit

Properties such as name and color of layers can be edited:

- To rename a layer, double-click the layer name or select **Rename** from the right-click menu.
- To change the color, select one or more layers and then click the current color for one of the selected layers. The **Select Color** dialog box is displayed. The color you select is applied to all selected layers.

23.8 Working with steel profiles

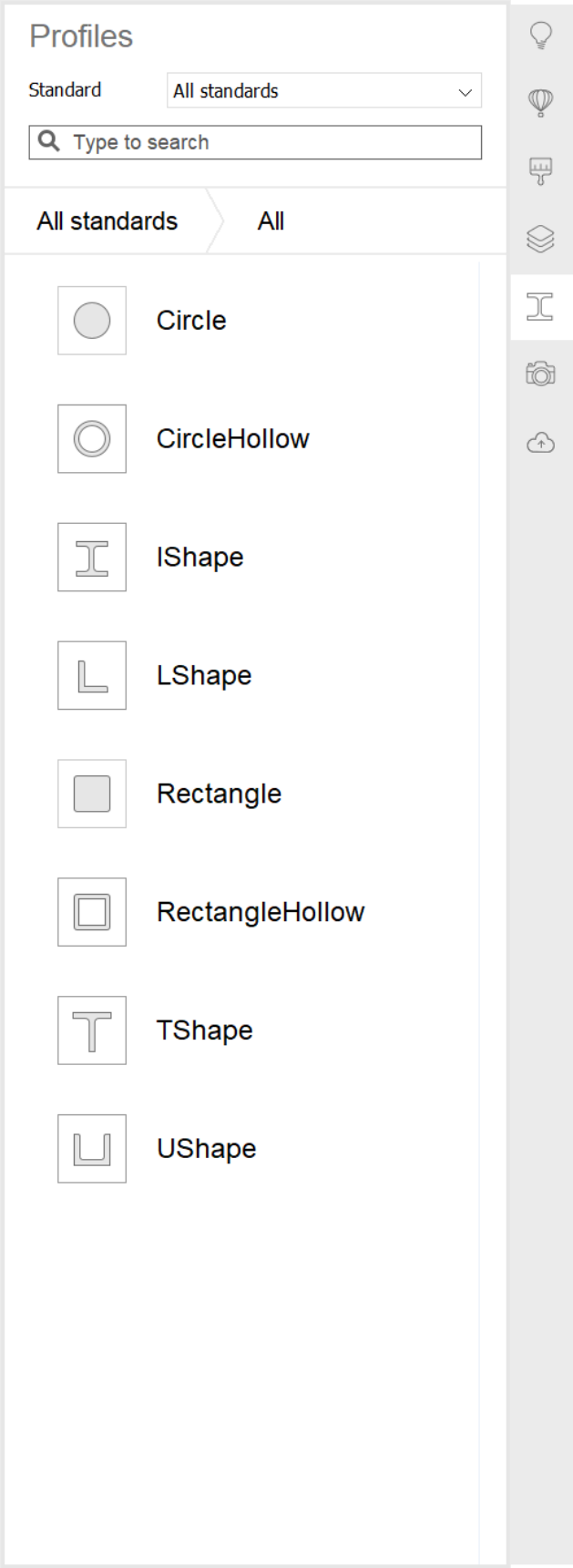
BricsCAD Shape offers standard structural steel profiles which enable you to easily create structural elements. Multiple standards are available, including American, British and European standards.

Steel profiles are created as linear solids.

23.8.1 Creating a steel profile

1 Click the **Profile** icon the vertical toolbar, on the right-hand edge of the screen.

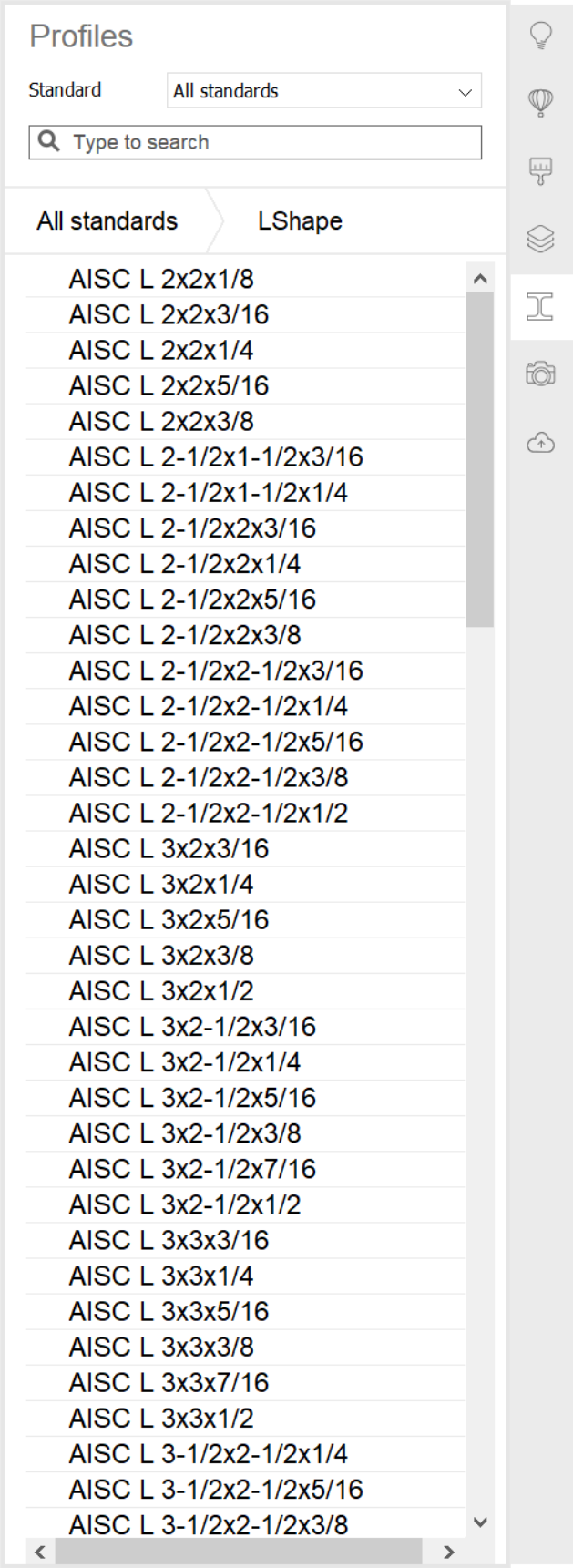
The **Profiles** panel displays.


Profiles
Standard
All standards
Type to search
All standards
All
Circle
CircleHollow
IShape
LShape
Rectangle
RectangleHollow
TShape
UShape

2 Optionally select a standard.

3 Select a profile shape.

4 A list of all profiles of the selected shape displays.


Profiles
Standard
All standards
Type to search
All standards
LShape
AISC L 2x2x1/8
AISC L 2x2x3/16
AISC L 2x2x1/4
AISC L 2x2x5/16
AISC L 2x2x3/8
AISC L 2-1/2x1-1/2x3/16
AISC L 2-1/2x1-1/2x1/4
AISC L 2-1/2x2x3/16
AISC L 2-1/2x2x1/4
AISC L 2-1/2x2x5/16
AISC L 2-1/2x2x3/8
AISC L 2-1/2x2-1/2x3/16
AISC L 2-1/2x2-1/2x1/4
AISC L 2-1/2x2-1/2x5/16
AISC L 2-1/2x2-1/2x3/8
AISC L 2-1/2x2-1/2x1/2
AISC L 3x2x3/16
AISC L 3x2x1/4
AISC L 3x2x5/16
AISC L 3x2x3/8
AISC L 3x2x1/2
AISC L 3x2-1/2x3/16
AISC L 3x2-1/2x1/4
AISC L 3x2-1/2x5/16
AISC L 3x2-1/2x3/8
AISC L 3x2-1/2x7/16
AISC L 3x2-1/2x1/2
AISC L 3x3x3/16
AISC L 3x3x1/4
AISC L 3x3x5/16
AISC L 3x3x3/8
AISC L 3x3x7/16
AISC L 3x3x1/2
AISC L 3-1/2x2-1/2x1/4
AISC L 3-1/2x2-1/2x5/16
AISC L 3-1/2x2-1/2x3/8

5 Click a profile in the list, then move the cursor to the model.

6 The outline of the selected profile is attached to the cursor.

7 Specify a point.

8 A linear solid displays dynamically.

A length and angle field display the current length an angle of the profile.

When the current direction is close to the X, Y or Z direction of the current coordinate system a colored tracking line displays. Press the **Shift** key to lock the tracking line. Press again to unlock.

9 Do one of the following to create the profile:

- Specify a point.
- Type a value in the length field, then press **Enter** or press the **TAB** key to activate the angle field.

The next profile displays dynamically.

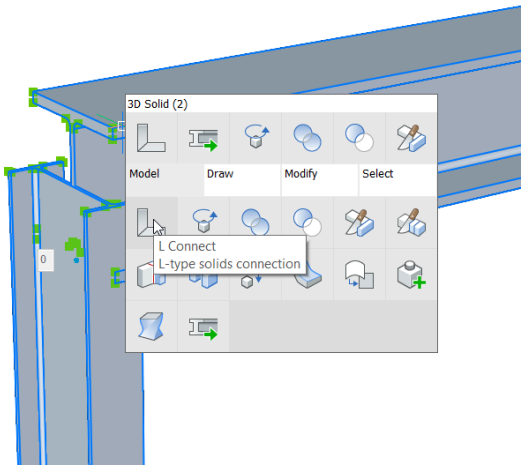
10 Repeat the previous steps to create more profiles or right click to stop.

23.8.2 Connecting Profiles

The L-Connect tool creates various types of L-connections between two selected profiles.

1 Select both profiles.

2 Click the L-Connect tool in the Model tab on the Quad.



- A bisector L-connection is created.
- The **Hotkey Assistant** (HKA) displays at the bottom of the screen.

3 Press the **Ctrl** key to modify the connection type.

The current connection type highlights in the **HKA**.

4 Right click to accept the connection type.

23.8.3 Assigning a different profile

1 Place cursor over the new profile in the **Profiles** panel. Do not click.

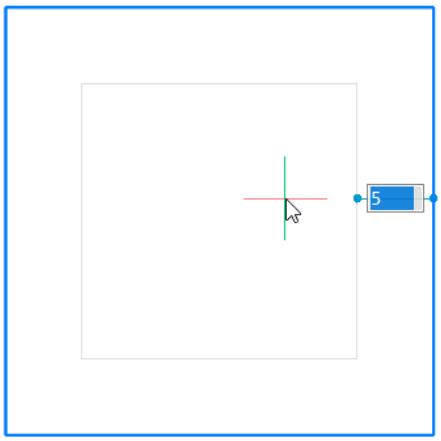
2 Hold down the left mouse button, then drag the profile onto the linear solid.

The selected profile is assigned to the linear solid.

23.8.4 Assigning a custom profile

1 Draw a custom profile shape using the tools in the **Draw** tab on the **Quad**:
   - A closed polyline.
   - A square or rectangle using the **Rectangle** tool.
   - A circle.
   - An ellipse.

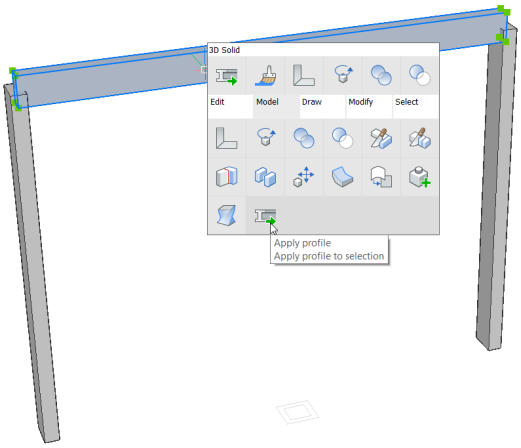
2 Optionally, use the **Offset** tool to create a hollow profile: select the outline, then specify the offset.



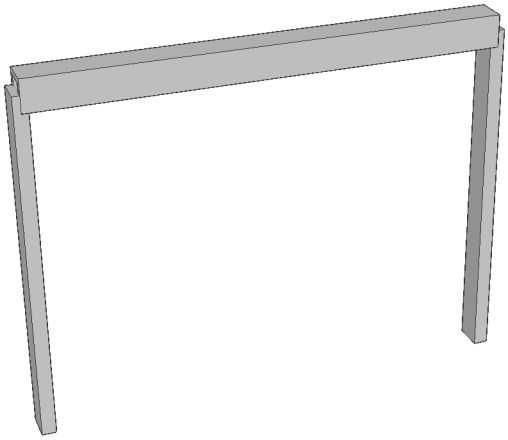
3 Hold down the **Ctrl** key, then select one or more linear solids. The linear solid(s) highlight(s).

4 Click the **Apply profile** tool in the **Model** tab on the **Quad**.

Prompts you: Select profile [selection options (?)]:



5 Do the following:
- Select the custom profile.
- Use the window overlap selection method to select the profile outline and the offset when applying a hollow profile.
- A preview of the result displays.
- Prompts you: Apply profile? [Ok/Quarter turn/Rotate/Delete clippings] <Ok>:



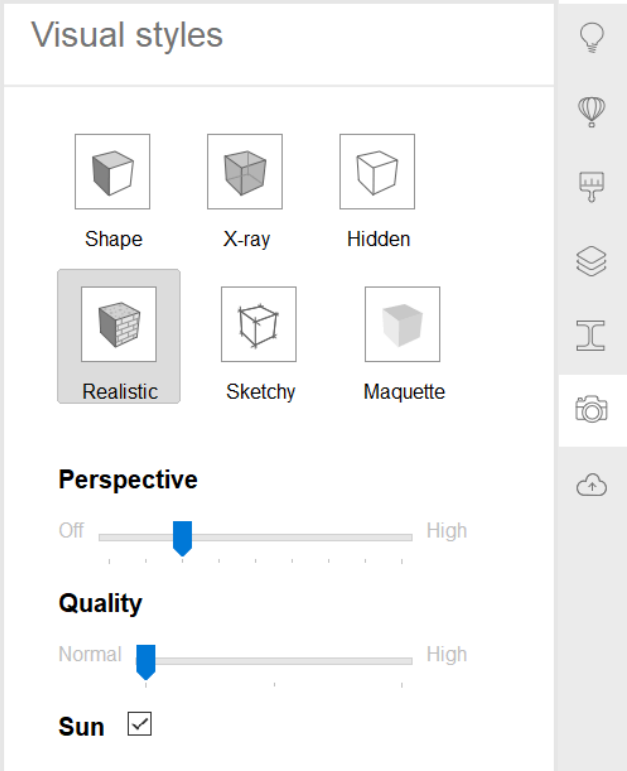
6 Do one of the following:
- Right click to apply the profile.
- Choose an option.
- Press the Esc key to quit.

23.9 Visualization in BricsCAD Shape

23.9.1 Visual styles panel

Visual styles control the appearance of your model.

The **Visual Styles** panel can be accessed by clicking the camera icon at the right side of the screen.



The look of the entire model will be changed when selecting another style.

The available visual styles are:

- Shape
- X-ray
- Hidden
- Realistic
- Sketchy
- Maquette

Perspective

You can change the perspective view by moving the **Perspective** slider.

Nota: Moving the perspective slider all the way to the left will turn the perspective off.

Quality

To control the on-screen rendering quality of the model, move the **Quality** slider.

Sun

To make the shadows of the sun appear, toggle the **Sun** checkbox.

23.9.2 Hide/Isolate/Show entities

Hide

When model elements block your view, you can hide them easily.

To hide one or more elements:

- Select the entities you want to hide.
- Click the **Hide Entities** icon in the **Quad** or in the toolbar.

Isolate

The opposite of hiding an element is isolating it, which will hide all the non-highlighted elements in your model.

To isolate entities:

- Select the entities you want to isolate.
- Click the **Isolate Entities** icon in the **Quad** or in the toolbar.

Show

Hiding and Isolating are temporary functions that you can use to simplify your model while working on it.

When you want to see all the elements in your model:

- Select the **Show Entities** icon in the “No Selection” Quad or in the toolbar.

23.9.3 Sections

You can use Sections to cut through your model dynamically.

To create a section:

- In the “No Selection” Quad, select the **Define Section** tool, found under the **Model** tab.
- Select any solid face in your model to create the section, parallel to the selected solid face.

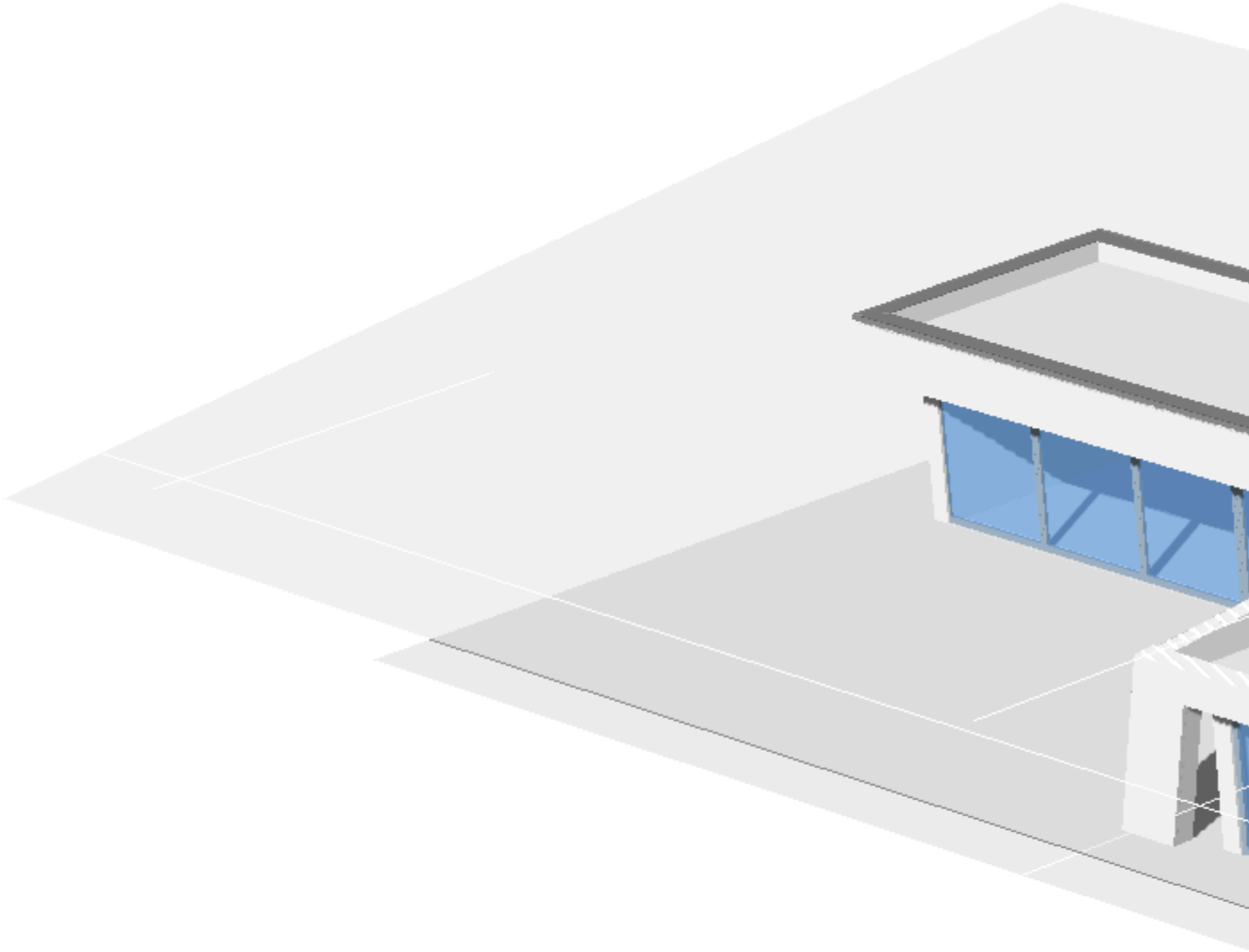
Nota: If you don’t choose a surface to attach the Section, a horizontal Section plane will be created.

You can toggle a section on and off by:

- double-clicking its handle
- highlighting the section plane and using **Clip Display** tool from the **Quad**.

A Section element has three grip edit points. Selecting the Section’s handle turns the grips on for editing:

- You can move the start and end grips to change the orientation and width of the Section.
- You can move the midpoint grip to reposition the section plane.
- If you click the arrow displayed near the midpoint grip, you will flip the clipping direction of the Section.



23.10 Solids in BricsCAD Shape

23.10.1 Drawing a box

1 Start creating a box by accessing the *BOX* command through the command line or by clicking the **Box** icon in the Quad or toolbar.

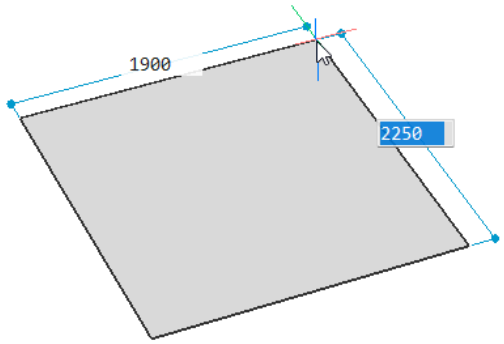
2 Specify a corner for the base of the box.

3 Specify the opposite corner for the base of the box to apply both the length and width.

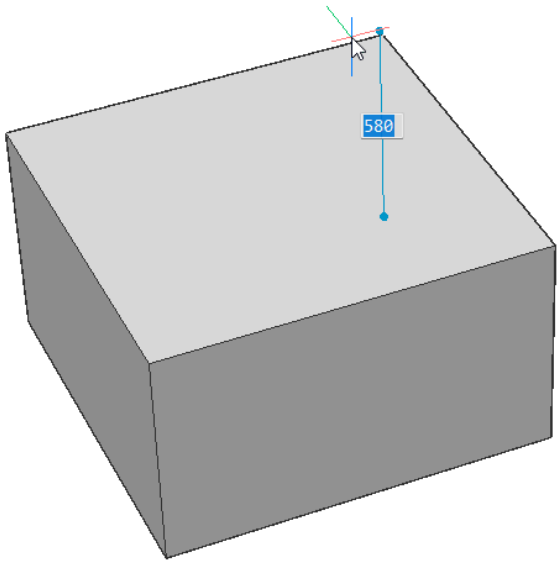
4 You can do this dynamically, by hovering the cursor at the opposite corner, or by entering the desired values in the dynamic input fields.

The box is created parallel to the x and y axes.

Nota: The blue dynamic field is the active one.



5 Specify the height of the box.



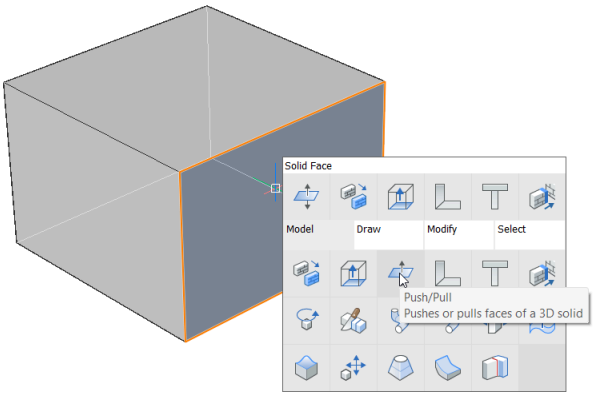
23.10.2 Manipulate a solid face

To manipulate a solid face:

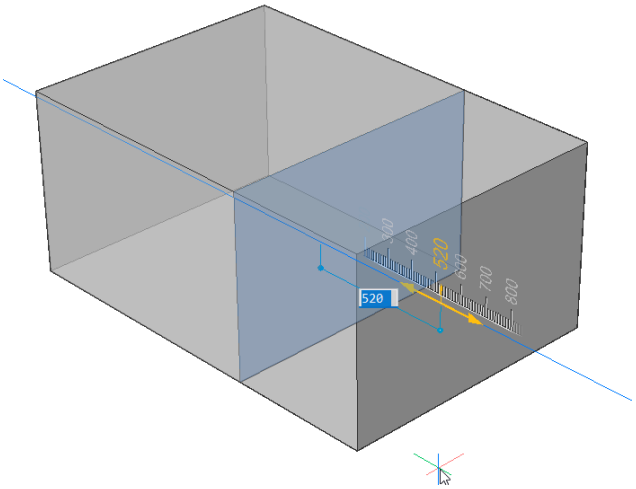
1 Hover the cursor over a face until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *DMPUSHPULL* command can be accessed.



4 Specify the displacement value or move with your cursor to define the end position of the face.



Nota: When selecting multiple faces and accessing the command, both faces will be pushed or pulled over same distances.

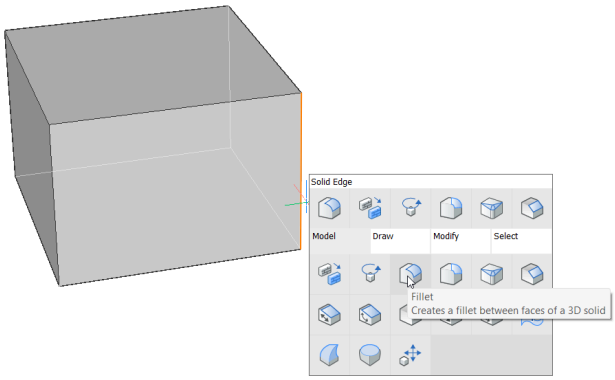
23.10.3 Manipulating a solid edge

To manipulate a solid edge:

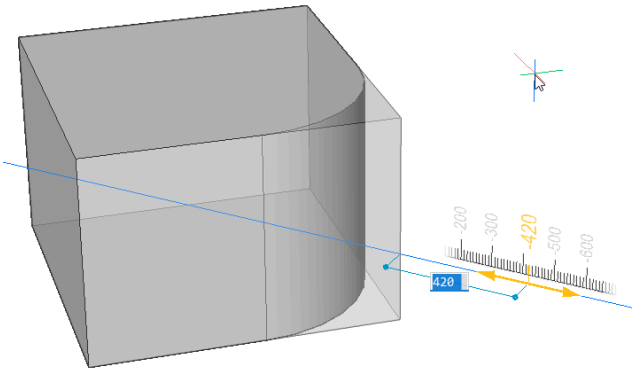
1 Hover your cursor over an edge while holding the **CRTL** key until highlighted.

2 Hover over the appearing icon to open the **Quad**.

3 Under the **Model** tab, the *FILLET* command can be accessed.

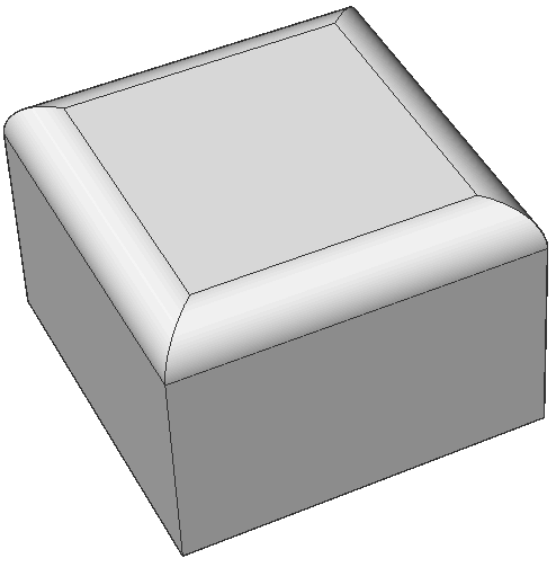


4 Specify the displacement value or move with your cursor to define the end position of the face.



Nota: When selecting multiple edges and accessing the command, all selected edges will have a fillet of the same radius.

Nota: When using a selection window to select edges, make sure you press the **CRTL** key twice so the edge selection icon is selected at the bottom of your screen.



23.10.4 Extrude solids

Selected boundaries can be extruded to create solids or surfaces. For example, a floor slab can be extruded between existing walls.

The extrude command can be accessed by the Quad or in the toolbar.

Nota: The extrude command can also subtract geometry also.

23.11 Creating walls in BricsCAD Shape

There are two ways to create a simple wall:

- Using the *POLYSOLID* command.
- Using the *BIMQUICKDRAW* command.

Nota: For a detailed explanation about *BIMQUICKDRAW*, read the article Quickdraw in BricsCAD Shape article.

23.11.1 About the Polysolid tool

The *POLYSOLID* command allows you to create wall solids by selecting a start point, an endpoint and entering a height.

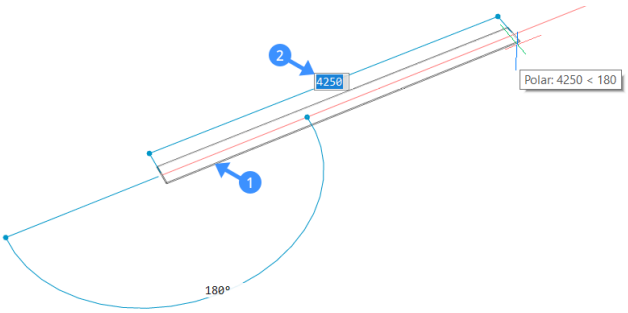
23.11.2 Procedure: Create a simple wall

1 Launch the *Polysolid* command:
   - Click the **Polysolid** icon

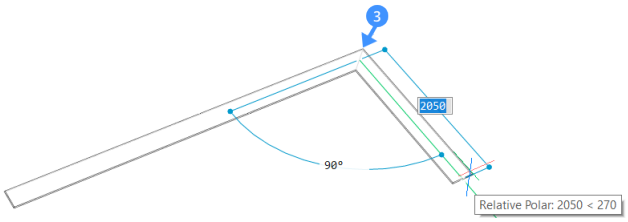
     in the toolbar
   - Type *POLYSOLID* in the command line.

2 Select a start point, the footprint (1) of the wall is displayed automatically.

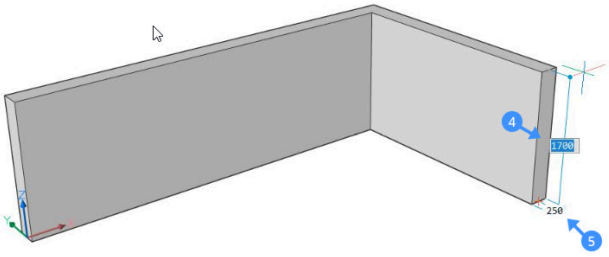
3 Move the cursor in the desired direction, the current length value is displayed in the length dynamic entry field (2).



4 Select an endpoint or enter a value in the dynamic input field and press **Enter** key. The footprint (3) of the adjacent wall is displayed. Select a new endpoint or enter values in the dynamic input field to create adjoining wall segments.



5 Press **Enter** once more or right-click. The height of the wall is displayed dynamically in the height field (4). Here, you can change the default height of the wall. To change the width of the wall, use the width field (5). You can switch between the two dynamic input fields by pressing **Tab** key.



6 To change the justification of the wall hit **CTRL** while the 'Polysolid justification Hotkey Assistant widget' appears at the bottom of the screen.

Nota: The width and height of the previous wall will be the new default values of the next Polysolid.

23.11.3 Procedure: Using an existing 2D plan to create walls

1 Launch the *POLYSOLID* command.

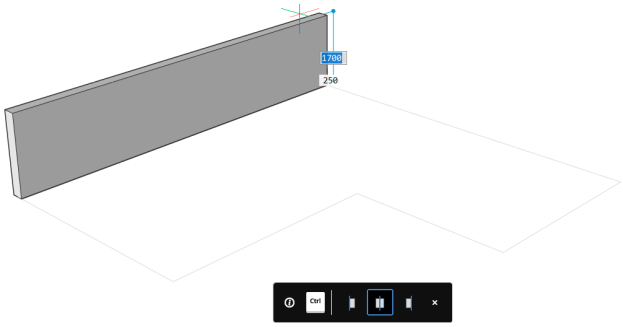
2 Type E in the command line or select **Entity** in the prompts menu.

3 Choose a 2D linear entity which will be the Polysolid base. Lines, open and closed polylines, arcs, circles, ellipses, elliptical arcs, and splines are accepted as a polysolid base.

4 Move the solid up or down and left-click to set the height of the wall, enter a value in the dynamic input field or right-click to accept the default value.

5 The current width of the wall appears in the **Width Field**. Press **Tab** switch between the Height and the Width field.

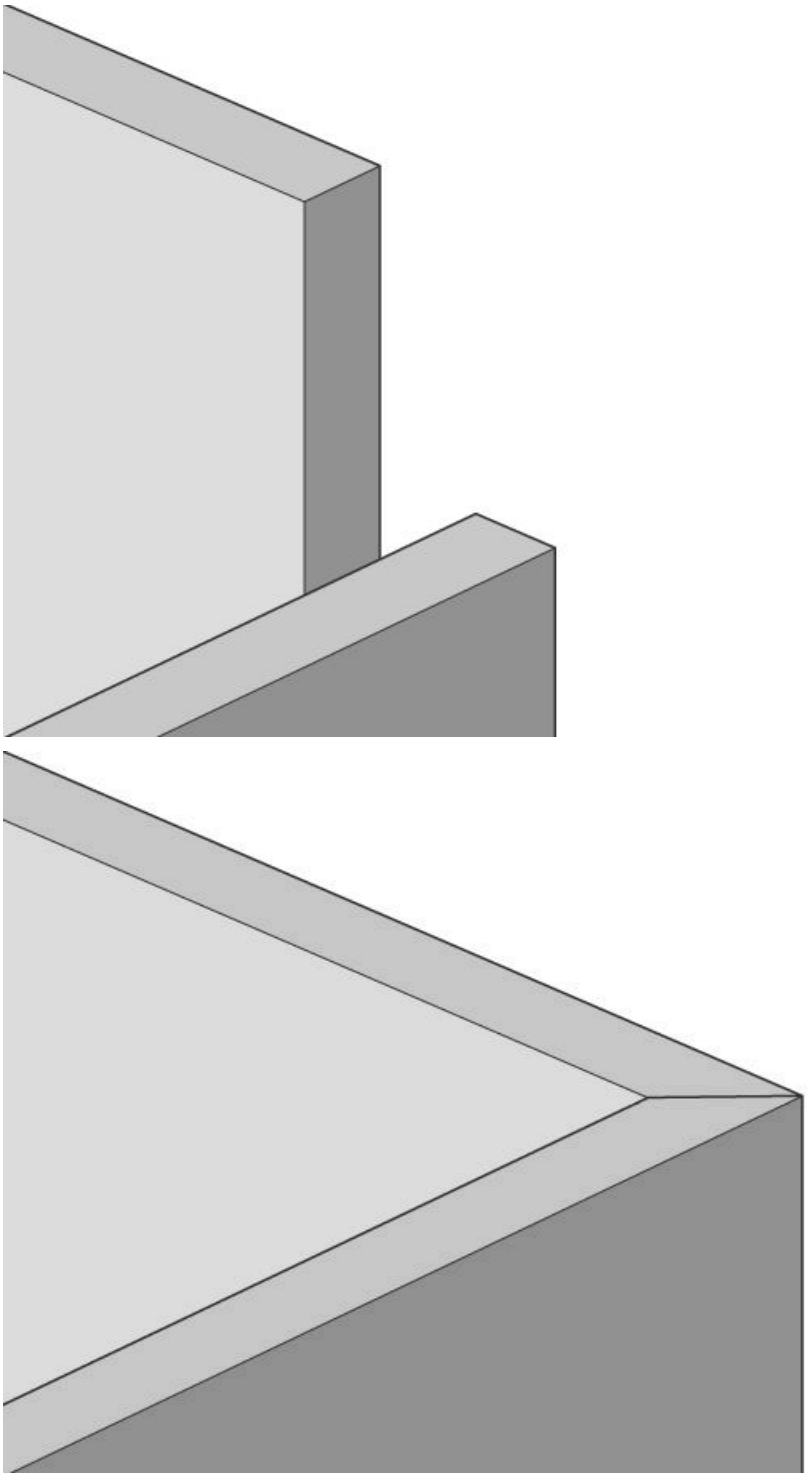
6 The **Hotkey Assistant** appears and displays the possible justification options. Press **CTRL** to cycle through the wall justification options.



23.11.4 Procedure: Creating L-connections between two walls

1 Launch the **LConnect** tool, by typing *LConnect* in the Command line and press Enter.

Select the two wall elements you want to connect.



Nota: When two walls intersect, a mitered connection will be created by default.

2 Change the layout of the wall connections using the 'LConnect HotKeyAssistant widget'.

3 The selected connection type is indicated by a blue frame.

The connection can be (from left to right) a bisector L-connection , L parallel type 1 , L parallel type 2 or a disconnection.

Press **Enter** to accept the current connection type or press **CTRL** to cycle through the possible connection options.

23.11.5 Procedure: Creating T-connections between two elements

1 Launch the *TConnect* command.

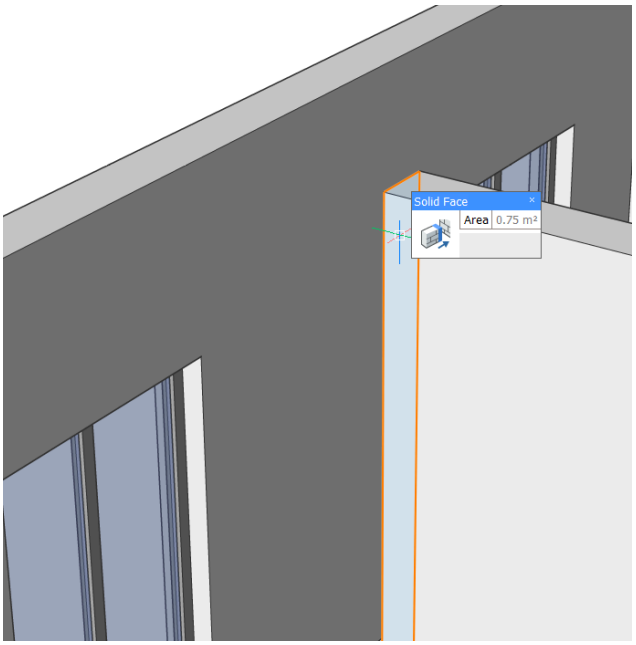
The tool allows you to connect the minor face of a wall to the major face of another element or wall. This tool can be used to connect the top face of a wall to a roof, the bottom face to a slab or side face of a slab to a wall.

2 Select the faces of the objects that you would like to connect and press **Enter**.

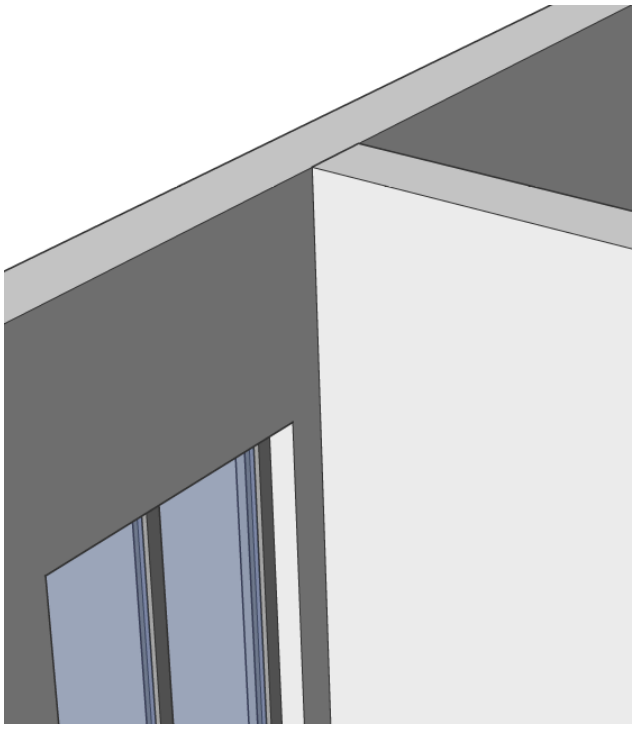
3 Select **Connect to nearest** in the prompt menu.

You can also use the **Quad** menu to create a T-connection between two walls or other elements:

- In the drawing area, highlight the face you want to connect to another wall.
- Use the **Connect with Nearest** tool from the **Quad**.



The Connect with Nearest tool automatically makes a T-connection between the two walls by extruding the face of the first wall to the face of the other wall.



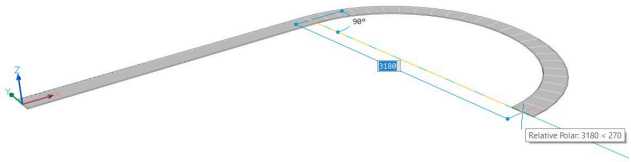
23.11.6 Procedure: Creating a curved wall

1 Launch the *POLYSOLID* command.

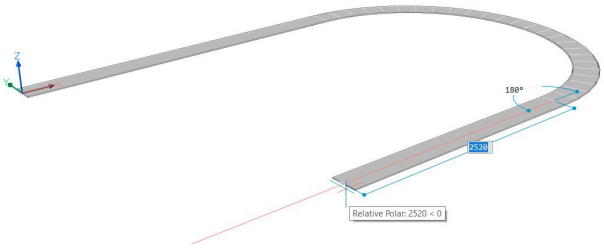
2 Select a start point and move the cursor to set the length or click to set the endpoint.

3 To create a curved wall, type A and press Enter to choose **Draw arcs** or select **Draw arcs** in the prompt menu.

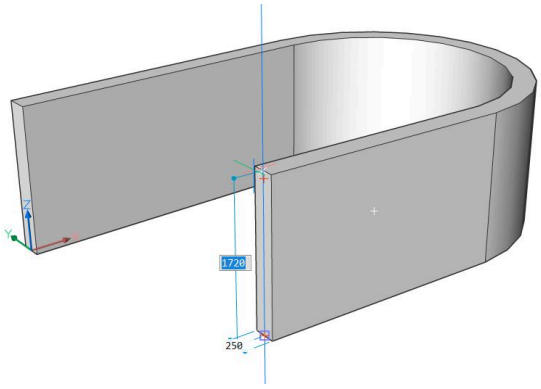
4 Now move the cursor in the desired direction to curve the wall. Enter a value in the dynamic input field to set the degree and length of the arc. Press **Enter**.



5 You can continue creating curved, adjoining wall segments by clicking or by entering values in the dynamic input fields. To go back to drawing straight wall segments, type L and press Enter or select **Draw lines** in the options dialog.

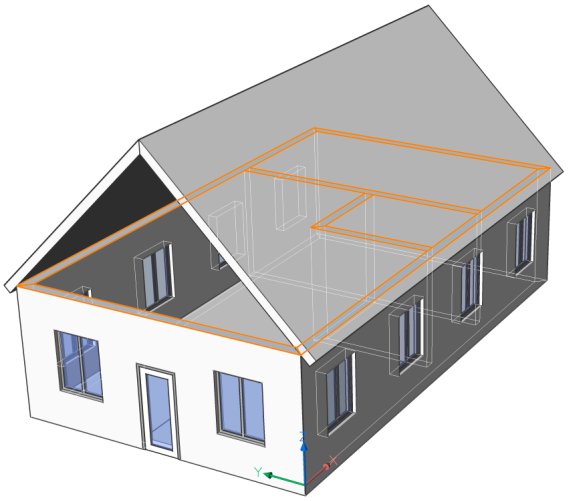


6 Press Enter once more or right-click. The height of the wall is displayed dynamically in the height field. Type a value to change the default height of the wall. To change the width of the wall, use the width field. You can switch between the two dynamic input fields by pressing Tab.

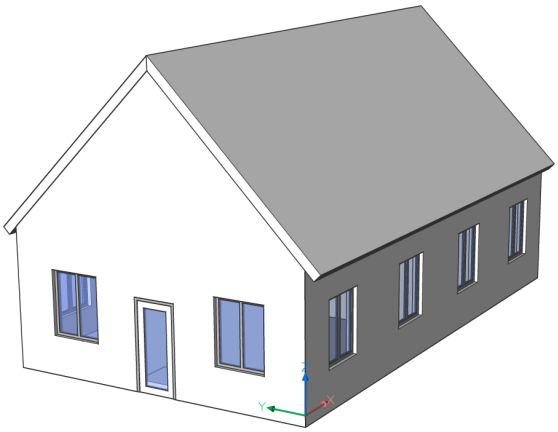


23.11.7 Procedure: Connecting multiple faces of the wall to the roof slab

1 To select all faces aligned to the current selection, highlight the top face of one of the walls and click the **Select aligned faces** tool in the **Select** tab of the Quad.



2 Launch the **Connect with Nearest** tool. The top faces of the wall are attached to the roof.



23.12 Quickdraw in BricsCAD Shape

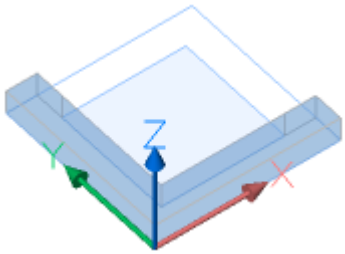
23.12.1 About the Quickdraw tool

The **Quickdraw** tool allows you to get started with conceptual modeling easily. You can use the **Quickdraw** tool to create a room or a whole building. Once in use, Quickdraw will remain active until you explicitly exit the command or until another command is executed.

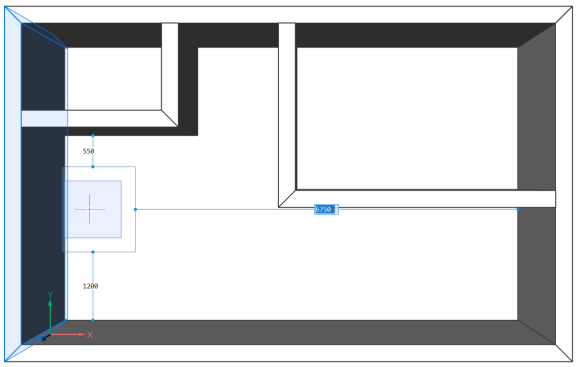
23.12.2 The magnetic snapping feature

With the magnetic snapping feature, you can easily align the QuickDraw cursor. When the command is active, move the cursor close to the object you want to align to. The QuickDraw cursor automatically snaps to the object. You have the following options:

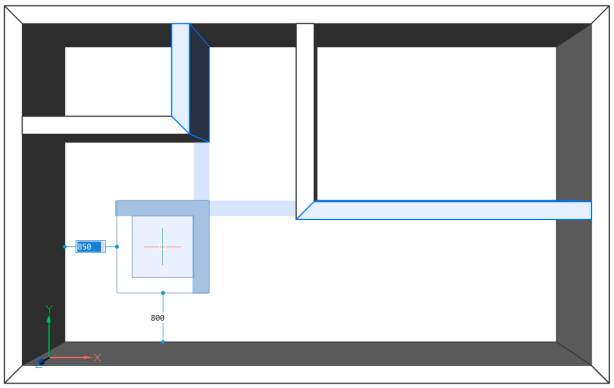
1 Align to the origin



2 Align to existing walls
   - To the wall itself:



   - The extends of the wall:



23.12.3 Procedure: Creating an L-shaped house

1 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** from the **Model** tab, click the **Quickdraw** icon in the ribbon or type BIMQUICKDRAW in the Command line.

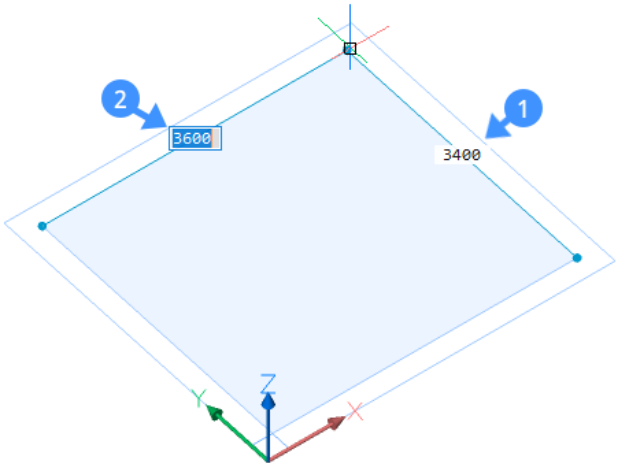
   You are prompted: Select first point or [floorHeight/wallWidth/slabThickness/autoAdoptsizes]:

   Nota: Select one of the options in the command bar to change the wall width, floorHeight and slab thickness of the roombeforeyou click and then specify the first corner of a building.

   Nota: When the **autoAdoptsizes** option is turned on, the dimensions of the walls and slab of the new room will be the same as the walls and slab you snap to. If you don't want this to happen, you can turn this option off in the command line.

2 Select any point in the drawing area to set the first corner.

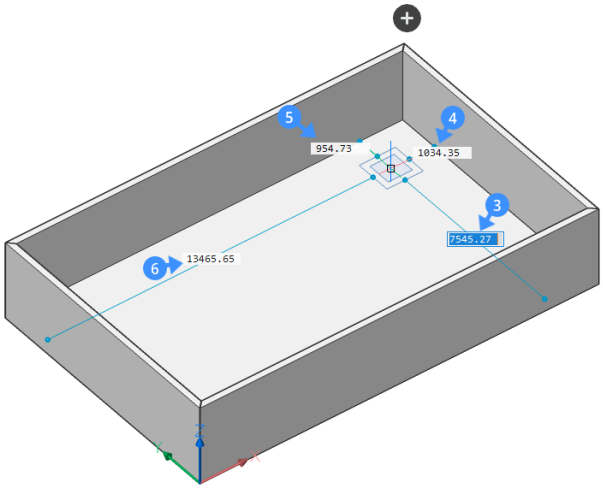
3 Drag the mouse cursor to define the footprint of the building.



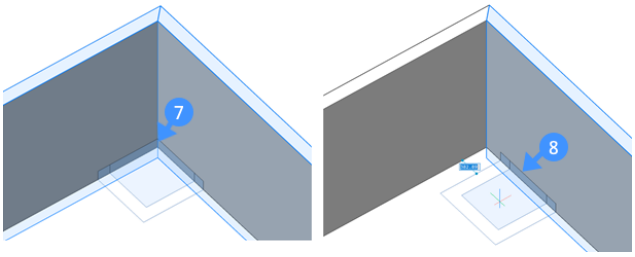
Specify the values of the room width and length in the Dynamic dimension field. Press the **Tab** key to switch between width and length and press **Enter** or just click freely. Press Enter or Escape to leave the QUICKDRAW command or you can continue to create more rooms.

4 Place the Quickdraw cursor inside the existing building to create more rooms.

5 When you place the Quickdraw cursor inside the existing building, the dynamic dimension fields display the distances from the Quickdraw cursor to the adjacent walls (3,4,5,6).



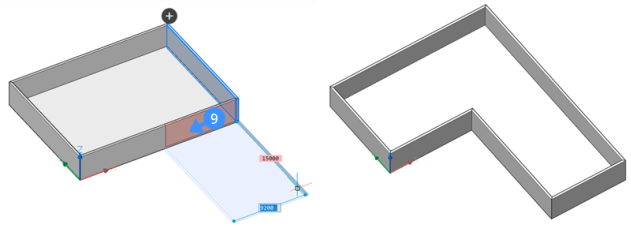
If the cursor aligns with one or two existing walls in the model, the layout of the cursor will adapt and depict the alignment.



To actually draw new walls, choose the distances from the already existing walls to specify your first point. Move your cursor or type the values in the dynamic dimension fields to do this. Draw walls in the same way as explained in steps 2 and 3.

6 To remove a section of a wall from the existing room, first, align the Quickdraw cursor with an existing wall at the inner side of the building. Then click and drag the cursor outside of the existing building.

7 The outer wall will turn red (9) to indicate the part of the wall that will be removed.

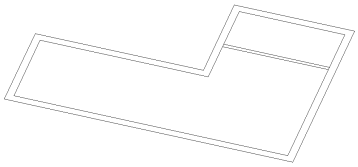


Nota: You can also align your cursor with an existing wall at the outer side of the building to remove a part of the slab. Then click and drag the cursor inside the existing building.

8 Select a second point. The wall highlighted in red will automatically be deleted and an L-shaped room will be created.

23.12.4 Procedure: using Quickdraw starting from a 2D layout

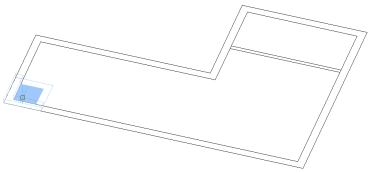
1 Open a 2D floorplan of a rectangular or L-shaped building.



Nota: It is also possible to draw a simple 2D floor plan with the POLYLINE command to start from. In this case, the default wall thickness of the BIMQUICKDRAW command will be used.

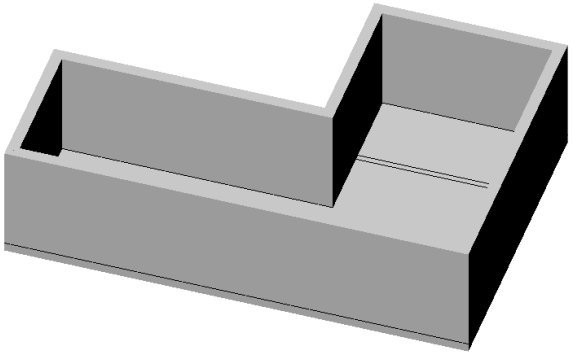
2 When the Quad cursor is in the 'No Selection' state (nothing is highlighted or selected), select **Quickdraw** in the **Model** tab, click the Quickdraw icon in the ribbon or type BIMQUICKDRAW in the Command line.

3 Hover over one of the outer walls with the cursor.

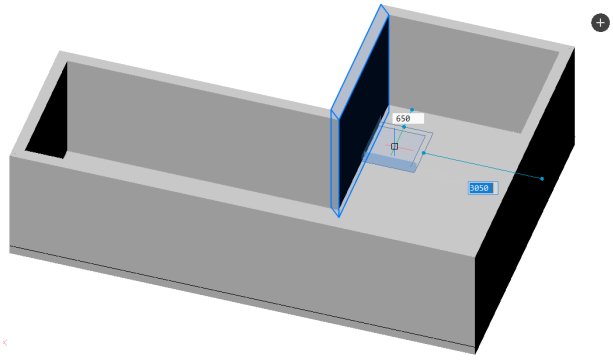


Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

4 Draw the outer walls, using the magnetic snapping feature to snap to the lines of the 2D drawing.

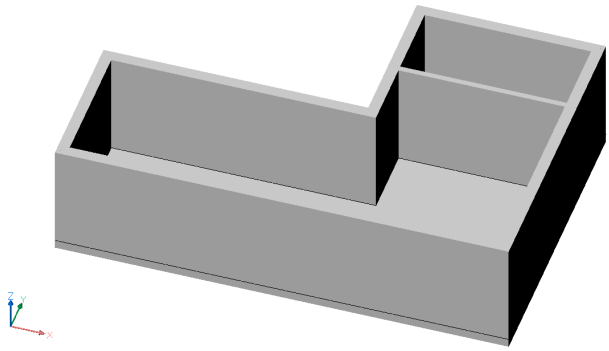


5 Hover over one of the inner walls with the cursor, using the magnetic snapping feature to snap to the lines of the 2D drawing.



Nota: The thickness of the wall automatically adapts to the distance between the parallel lines of the 2D drawing.

6 The 2D plan has been transformed into a 3D model.



23.13 Creating windows and doors in BricsCAD Shape

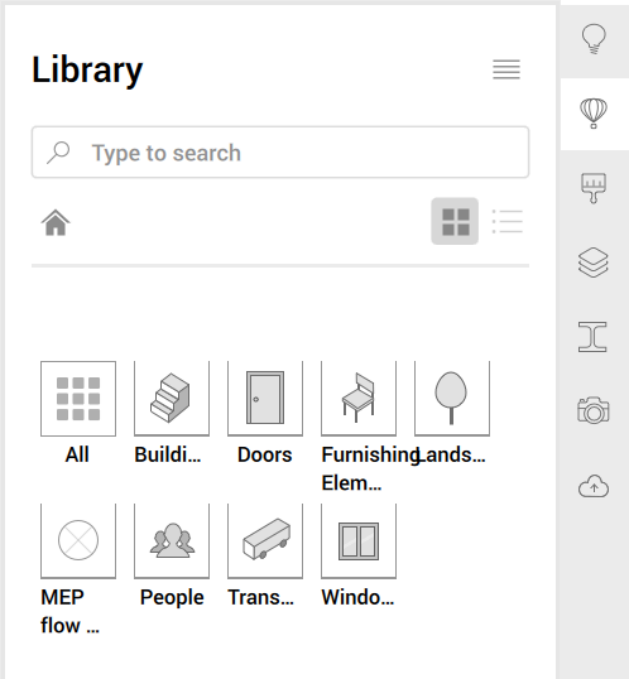
There are two ways to create windows or doors in BricsCAD Shape.

- Pick a door or window from the **Library** panel.
- Create your own custom window.

23.13.1 Procedure: Pick windows or doors from the library

1 Open the **Library** panel by clicking the airballoon icon on the right hand edge of the screen.

2 Click the **Doors** section in the **Library** panel.



3 Select a door from the panel.

4 Move the cursor into the model.

5 Click to place the component.

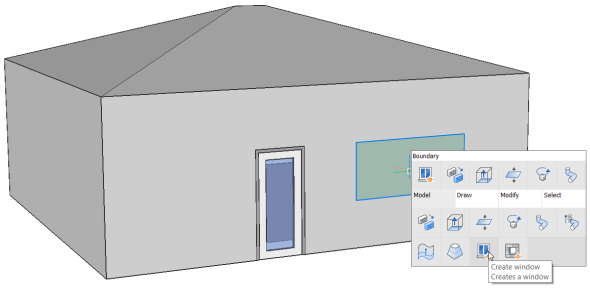
6 Press the **CTRL** key to toggle between door positioning and door size. Enter the desired values for the height and width of the door by pressing the **Tab** key. When you're done sizing and placing the door, press the **Enter** key.

Nota: This is the same for a window. Change Doors to Window section. Drag a window into the wall. You can change the size and location of a window just as you did with the door.

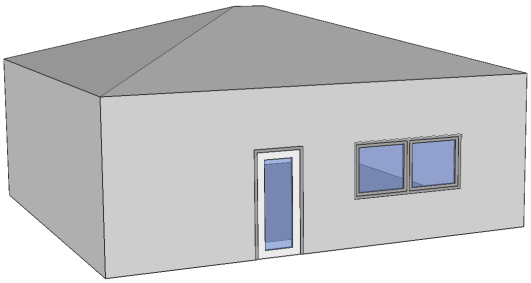
23.13.2 Procedure: Create a custom window

1 Use the either the **Polyline** tool, the **Polygon** tool or the **Rectangle** tool to draw the outline of the window on the wall face.

2 Move the cursor inside the window outline and, when the boundary is highlighted, choose **Create window** from the Quad.



3 The available window styles are shown in a dialog box. Move your cursor over the window design of your choice, and left-click it. The window is automatically created.



You can also create a custom window from a wall boundary:

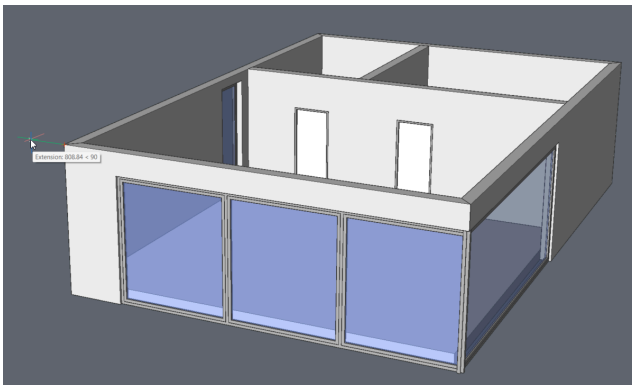
1 Select face of the outer wall and use the **Offset** tool from the Quad.

2 Move the cursor to re-size the new boundary, or key in a value and press Enter. Then, highlight the offset boundary and select **Create window** from the Quad.

23.14 Creating roofs in BricsCAD Shape

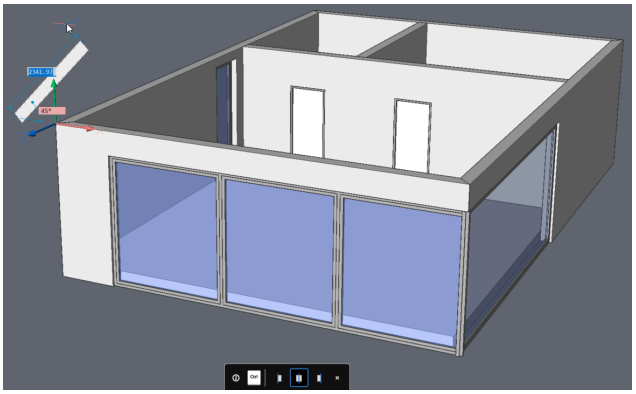
Launch the *POLYSOLID* command trough the toolbar or the "No selection" Quad.

1 To align the drawing lane to the wall surface, press the **SHIFT** button while hovering over the wall surface. The alignment of the slab can be changed by pressing the **CRTL** button.

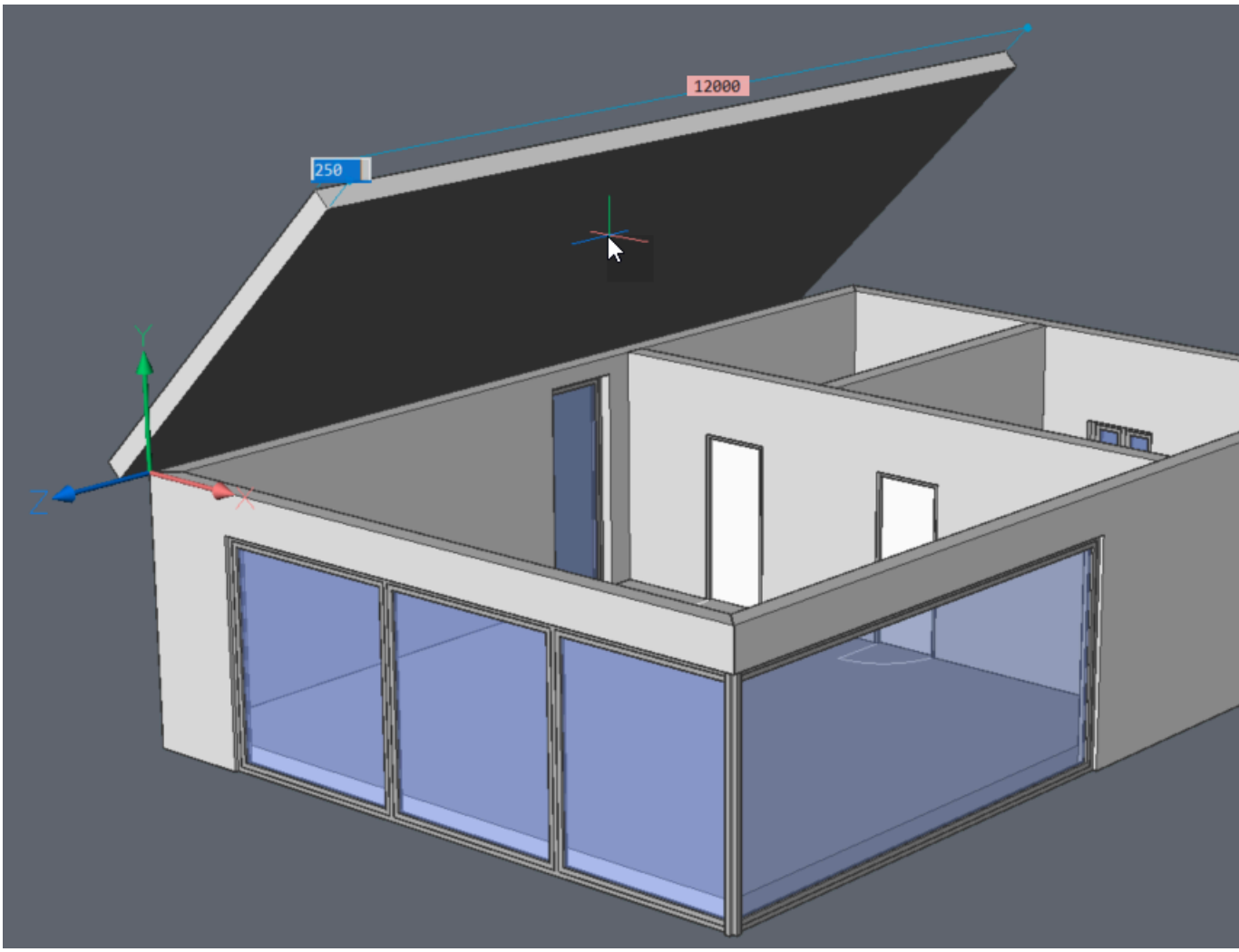


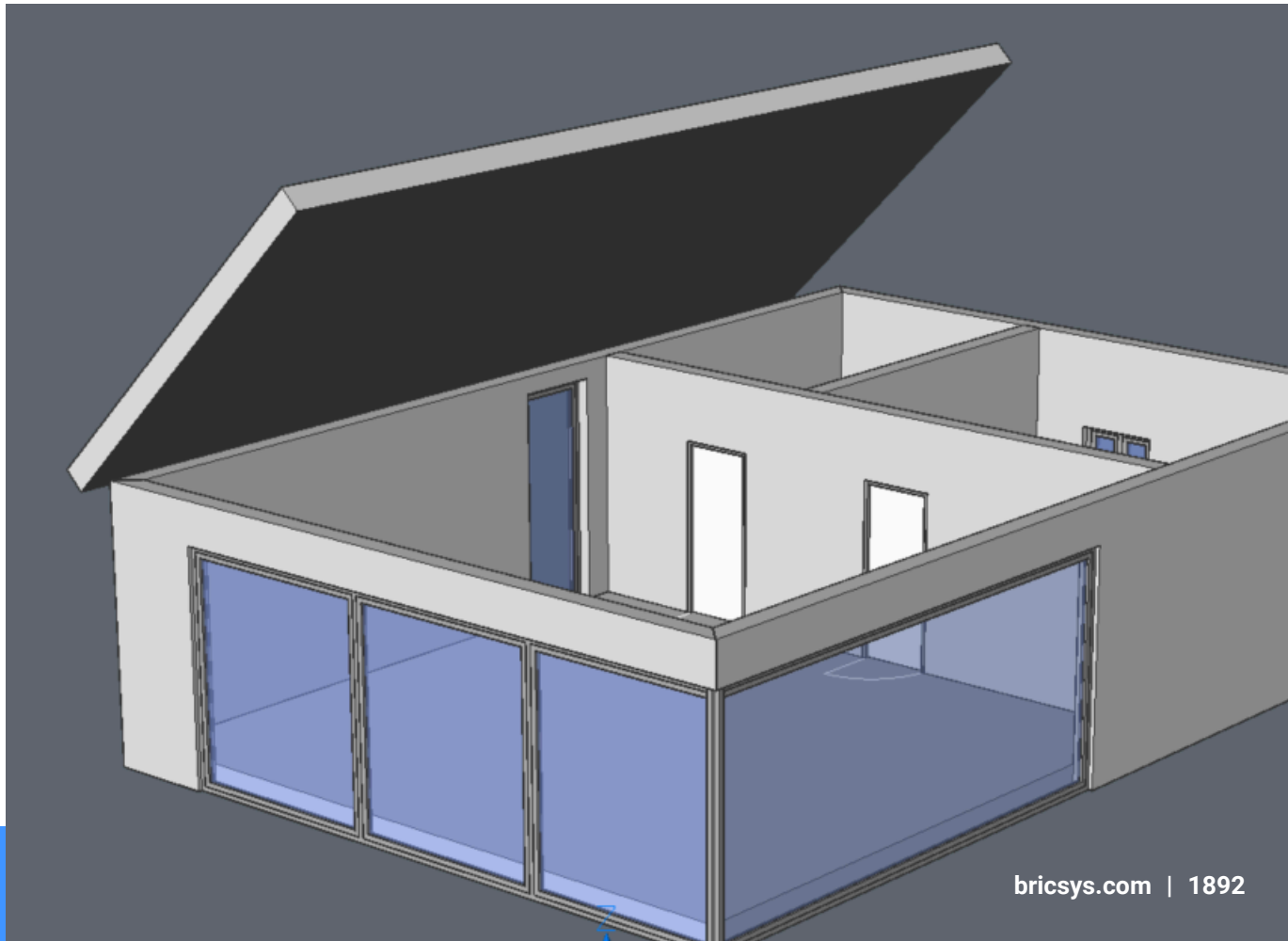
2 Pick an origin point to start drawing.

3 To lock the inclination, press TAB until the dynamic dimension of the inclination turns blue. Enter a value and press Enter.

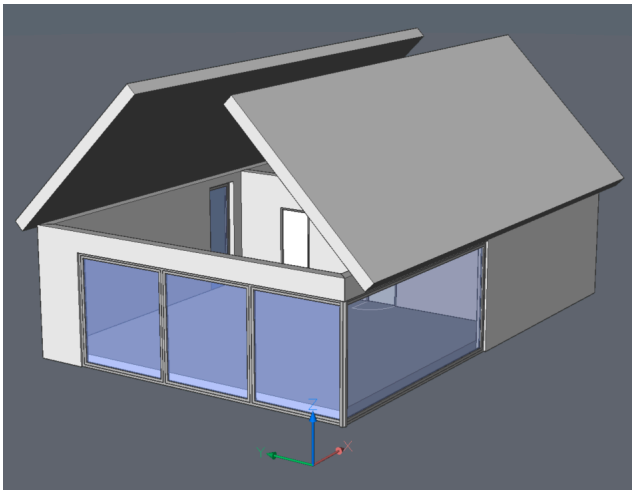


4 The profile thickness can be changes by pressing the TAB key and entering the desired value.

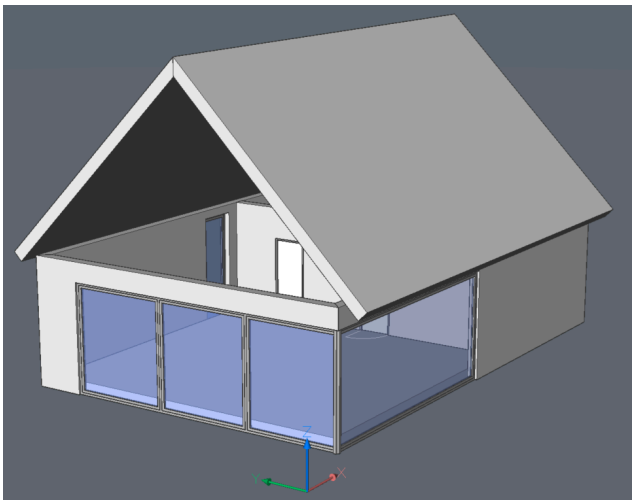

12000
250



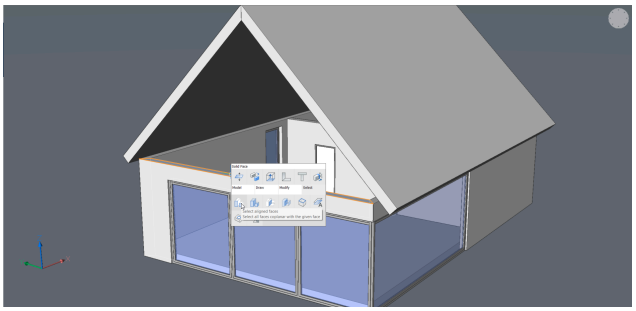
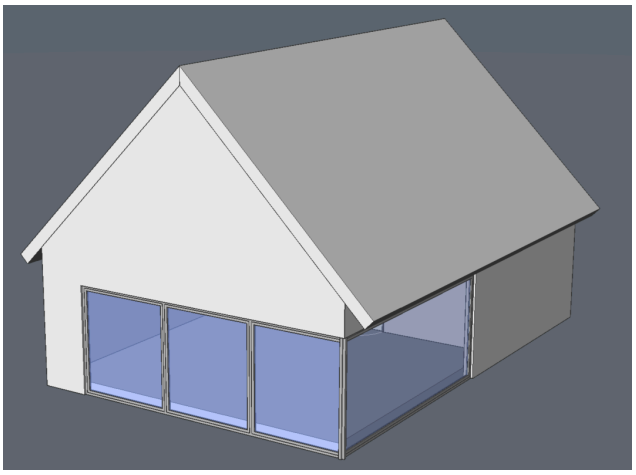
5 Repeat the same process on the other side of the building. Another way to have identical slabs on both sides is by 3D mirroring the first slab.



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.